

# Modeling of Turbulent Flow through Port/Valve Assemblies with an Algebraic Reynolds Stress Model

C.J.Deschamps and A.P.Watkins

*Department of Mechanical Engineering  
Thermodynamics and Fluid Flow Division  
UMIST  
PO Box 88, Manchester, M60 1QD  
UK*

## ABSTRACT

The present work is a numerical investigation of the turbulent flow through intake port and valve passages of IC engines with the employment of an Algebraic Second-Moment (ASM) turbulence model closure and is motivated by the importance that such a system has on engine operating and emission characteristics. Although idealised the axisymmetric flow geometry considered in the investigation still possesses important features present in practical assemblies. Previous analyses of the same flow geometry have shown that at large valve lifts eddy-viscosity models are unable to predict flow separation in the valve passage. This investigation is the first to include a second-moment closure for evaluation of the flow. Results obtained with both classes of turbulence models are compared and it is concluded that the ASM model gives improved results for all situations considered. This is associated with the capability of the ASM model to return lower levels of Reynolds stress in the vicinity of the valve passage, which leads the flow to separate more readily thereafter in accordance with experimental evidence.

## INTRODUCTION

Of much importance in internal combustion engines is the process of filling the cylinder with as much fresh air as possible and supplying a favourable flow field for combustion. The first requirement, also known as volumetric efficiency, can be maximised by avoiding restrictions to the flow and preventing flow separation in the system. Since the valve/port assembly is the most important flow restriction in the intake system, a good understanding of the fluid mechanics in such devices is critical to developing engine designs with better operating and emission characteristics.

Due to the importance of the subject a considerable number of works have been published in the literature. In recent years there has been a growing interest on the numerical prediction of flows in port/valve assemblies motivated in great part by the increased computational resources available and the appearance of new numerical techniques. The range of the investigations following this trend has been large, varying from predictions in

two-dimensions (1,2,3,4) and three-dimensions (2,5) flows under steady state condition up to the inclusion of the complete port and cylinder geometries under motoring conditions (6,7). However, it has been found that even for simple geometries eddy-viscosity models cannot predict accurately separated flow regions verified experimentally in the valve passage at large valve lifts, unless some ad hoc extra term is introduced into the equations (2,4). It is recognised that eddy-viscosity models are reasonably satisfactory in simple two-dimensional shear flows but also acknowledged are their deficiencies in situations of streamline curvature, acceleration and separation; all of them are present here. In this work a first step is given towards the inclusion of a second moment closure turbulence model in the analysis of the flow in port/valve assemblies and, as such, it seemed natural to use at this stage the Algebraic Second-Moment closure (ASM). For the assessment of the model performance the flow cases to be investigated are also modelled with an eddy-viscosity model and then the results obtained with both models are compared.

The present work investigates numerically the flow through an axisymmetric geometry of inlet port and valve under steady state condition, considered experimentally in the works of Gosman and Ahmed (1) and Bicen et al (8), shown in Fig. 1.

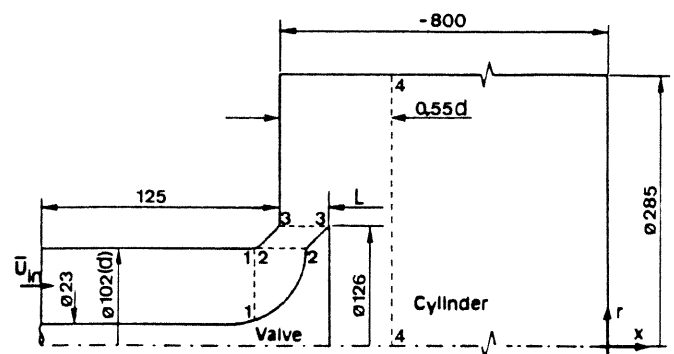


Fig. 1 Flow geometry

The same geometry has also been adopted in other numerical explorations (1,2,4) and used at UMIST for in-cylinder flow analysis with a Differential Second-Moment

closure (DSM) of turbulence (9). It is expected that future investigations within the framework of the present analysis can be done or extended using the results from those works.

## TURBULENCE MODELLING

The ASM model is derived from the differential form of the Reynolds stress equations:

$$\frac{D\overline{u_i u_j}}{Dt} - \frac{\partial}{\partial x_k} \left[ c_s \frac{k}{\varepsilon} \overline{u_k u_l} \frac{\partial \overline{u_i u_j}}{\partial x_l} \right] = \mathcal{P}_{ij} + \phi_{ij,1} + \phi_{ij,2} \quad (1)$$

$$+ \phi'_{ij,1} + \phi'_{ij,2} - \frac{2}{3} \delta_{ij} \varepsilon$$

where,

$$\mathcal{P}_{ij} \equiv - \left[ \overline{u_i u_k} \frac{\partial U_j}{\partial x_k} + \overline{u_j u_k} \frac{\partial U_i}{\partial x_k} \right] \quad (2)$$

$$\phi_{ij,1} \equiv -c_1 \frac{\varepsilon}{k} \left[ \overline{u_i u_j} - \frac{2}{3} \delta_{ij} k \right] \quad (3)$$

$$\phi_{ij,2} \equiv -c_2 \left[ \mathcal{P}_{ij} - \frac{2}{3} \delta_{ij} \mathcal{P} \right] \quad (4)$$

$$\phi'_{ij,1} \equiv c'_1 \frac{\varepsilon}{k} \left[ \overline{u_k u_m} n_k n_m \delta_{ij} - \frac{3}{2} \overline{u_k u_j} n_k n_i - \frac{3}{2} \overline{u_k u_i} n_k n_j \right] f_w \quad (5)$$

$$\phi'_{ij,2} \equiv c'_2 \left[ \phi_{km,2} n_k n_m \delta_{ij} - \frac{3}{2} \phi_{ik,2} n_k n_j - \frac{3}{2} \phi_{jk,2} n_k n_i \right] f_w \quad (6)$$

with  $\mathcal{P} = 1/2 \mathcal{P}_{kk}$  and  $f_w = (k^{3/2}/\varepsilon)/(c_l d_w)$ . The coefficients  $c_s$ ,  $c_1$ ,  $c_2$ ,  $c'_1$ ,  $c'_2$  and  $c_l$  take the values 0.22, 1.8, 0.6, 0.5, 0.3 and 2.44, respectively.

In the ASM treatment, the left side of equation (1) is replaced by Rodi's ASM transport hypothesis [10],

$$\frac{D\overline{u_i u_j}}{Dt} - \frac{\partial}{\partial x_k} \left[ c_s \frac{k}{\varepsilon} \overline{u_k u_l} \frac{\partial \overline{u_i u_j}}{\partial x_l} \right] = \frac{\overline{u_i u_j}}{k} [\mathcal{P} - \varepsilon] \quad (7)$$

The transport equations for the turbulence kinetic energy  $k$  and its dissipation  $\varepsilon$  needed to close the model are written, respectively, as follows:

$$\frac{Dk}{Dt} = \frac{\partial}{\partial x_j} \left[ c_s \overline{u_i u_j} \frac{k}{\varepsilon} \frac{\partial k}{\partial x_i} \right] + \mathcal{P} - \varepsilon \quad (8)$$

and

$$\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j} \left[ c_\varepsilon \overline{u_i u_j} \frac{k}{\varepsilon} \frac{\partial \varepsilon}{\partial x_i} \right] + c_{\varepsilon_1} \frac{\varepsilon}{k} \mathcal{P} - c_{\varepsilon_2} \frac{\varepsilon^2}{k}, \quad (9)$$

where  $c_\varepsilon$ ,  $c_{\varepsilon_1}$  and  $c_{\varepsilon_2}$  take the values 0.18, 1.44 and 1.92, respectively.

Walls have the effect of damping the intensity of velocity fluctuations normal to the surface considered and, hence, high levels of anisotropy are found there. In the second-moment closure such effects are modelled with the so called 'wall-reflection terms', equations (5) and (6). The terms were originally devised for channel flows with the intention of redistributing energy among the fluctuating velocity components so as to mimic levels found for the three components close to walls. Craft (11) verified, nevertheless, that in impinging flow regions the

term  $\phi'_{ij,2}$  gives in fact the wrong sign to the correction and instead of being damped the component normal to the wall is enhanced. Indeed, one of the main difficulties in devising wall-reflection terms is related to the amount of empiricism one has to exercise seeking to produce distributions verified experimentally in different flow situations. Current terms are formulated only for plane surfaces and distances to the wall  $d_w$  involved in equations (5) and (6), through the length scale function  $f_w$ , are generally found to be a parameter not easy to evaluate in complex geometries. For example, in the case of corners, the proximity of two walls leaves an open question concerning what form should the wall reflection assume there since there is no obvious way of interpreting the distance to the wall  $d_w$ .

Unfortunately, the present flow is a strong candidate to present the above difficulties in the application of wall reflection terms. For instance, after passing a relatively simple region in the port duct the flow starts deflecting along the curved surface of the valve towards the valve passage. Afterwards, in the in-cylinder region, the flow impinges against the cylinder wall, bringing about some features verified by Craft (11). For these reasons, the use of wall-reflection terms in the situation considered here becomes difficult to justify and, therefore, were not included in the numerical simulations.<sup>1</sup>

Another important aspect in the turbulence modelling is the description of solid boundaries. In the present flow situation, a problem arises since the employment of wall-functions based on the 'universal velocity-distribution law' are not really adequate to describe boundary-layers approaching separation. A second inconvenience is related to the numerical aspect and caused by the condition of minimum turbulence level that must be respected when using wall-functions (for practical reasons usually fixed as  $y^+ = 11.6$ ). Naturally, in flows where important features occur close to the walls it is quite difficult, if not impossible, to balance the needed grid resolution against the minimum value for  $y^+$ . Given the foregoing reasons and in order to predict the flow in port/valve assemblies it seems to be essential to avoid the use of wall-functions and to include the near-wall region in the calculations. However, the task of elaborating a second moment closure for low-Reynolds-number turbulence is particularly difficult and although much effort have been directed to the modelling of the near-wall region progress has not reached the point for the full benefit of flow in complex geometries as the one considered here. One way to tackle the problem, which has been used in recent works at UMIST (11,12), is to adopt a version of a low-Reynolds-number eddy-viscosity model in the near wall region and a standard form of second moment closure beyond that. In this analysis the one-equation model has been included for calculations of the region close to the walls and, in this way, comparisons between eddy-viscosity and second-moment closures are made with the use of two versions of two-layer model. The first version has the one-equation model and the standard High-Reynolds-Number  $k$ - $\varepsilon$  model (13) applied in the

<sup>1</sup>It is worth mentioning that Lea [9] in his in-cylinder numerical simulations found that the inclusion of wall-reflection terms made results worse in certain regions of the flow.

near-wall and core regions, respectively; this model will be referred to hereafter as Two-L- $k\varepsilon$ . The other version replaces the High-Reynolds-Number  $k-\varepsilon$  model with the ASM and will be identified as Two-L-ASM. It is opportune to state at this point that in this technique the location of the interface between the two turbulence models may affect to some extent numerical predictions and that this is an acknowledged limitation of the present calculations. However, the main objective here is to assess effects that substitution of the High-Reynolds-Number  $k-\varepsilon$  model by the ASM in the core region may originate in the numerical solution.

Basically, the one-equation model adopted here calculates the dissipation  $\varepsilon$  according to

$$\varepsilon = \frac{k^{3/2}}{l} \quad (10)$$

The eddy viscosity is obtained using the following relationship:

$$\nu_t = c_\mu k^{1/2} l_\mu \quad (11)$$

where  $c_\mu (= 0.09)$  is an empirical constant. The length scales  $l$  and  $l_\mu$  in the equations above given by

$$l = 2.4 y_e [1 - \exp(-A_D y_e^*)] \quad (12)$$

and

$$l_\mu = 2.4 y_e [1 - \exp(-A_\mu y_e^*)] \quad (13)$$

include 'damping functions' to give the correct flow field in regions where the molecular viscosity  $\mu$  is greater or comparable to the turbulence viscosity  $\mu_t$ . The parameter  $y_e$  is a geometry-determined effective length scale which here was considered to be the normal distance between the point considered to the nearest wall. Additionally,  $A_D$  and  $A_\mu$  are constants equal to 0.235 and 0.016, respectively, and  $y_e^*$  is a local turbulence Reynolds number indicating the turbulence intensity, and is defined as  $y_e^* = y_e k^{1/2} / \nu$ . The equation for the transport of the kinetic energy  $k$  is essentially the same as used in the High-Reynolds-Number  $k-\varepsilon$  model but with the molecular diffusion being taken into account.

## NUMERICAL METHODOLOGY

The set of equations governing the flow are transformed to an orthogonal curvilinear coordinate system and then solved based on a finite volume methodology developed at UMIST (14).

As far as boundary conditions are concerned, mean velocity and turbulent kinetic energy profiles at inlet of the port duct were interpolated from the experimental data of Ahmed (1). The distribution of the dissipation rate was estimated based on the assumption of turbulence-energy equilibrium (13). In the plane of symmetry, the normal velocity to the boundary is set to zero and the same for normal gradients of all other quantities. Concerning solid boundaries, calculations were extended up to the walls. The fluid exit was located far enough downstream in the in-cylinder region that a condition of parabolic flow could be assumed.

A great deal of effort was invested in tests to assess the numerical solution sensitivity to variables such as grid

refinement, convection scheme and boundary condition. Three grid levels (157x94, 157x112 and 193x94) were used, with refinement being mainly promoted in the port exit and valve passage regions. Of much help was some evidence of the discretization needed for the analysis and made available by other works (1,2,4). For numerical stability reasons, the effect of grid refinement on the numerical solution, documented in (15), was examined with use of the Power-Law-Differencing-Scheme (PLDS). After selecting the grid offering the best compromise between accuracy and computational economy, a further reduction on the truncation error was made possible by replacing in the simulations the PLDS with the higher order accurate convection scheme QUICK. Fig. 2 gives a partial view of the computational grid used in this work and which either exceeds or is comparable to the discretization level adopted in the aforementioned works. The full grid extended downstream in the cylinder region to the full extent of the geometry shown in Fig. 1.

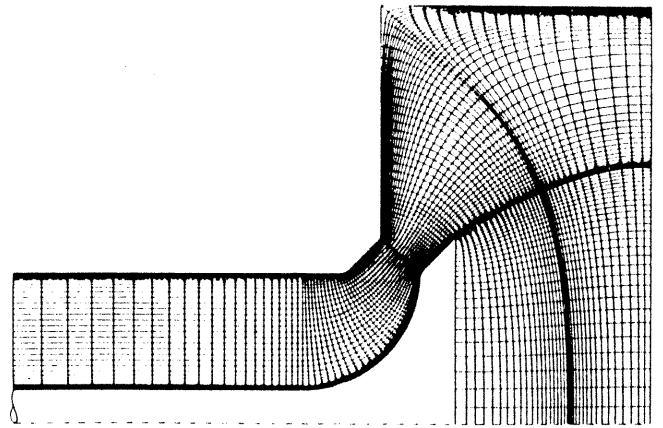


Fig. 2 Computational grid

## RESULTS

The main emphasis of this investigation is the prediction of flows at three large valve lifts ( $L/d = 0.22, 0.25$  and  $0.30$ ) since it is in this range of lifts, characterized by separated flow regions in the valve passage, where the greatest discrepancy between the numerical and experimental results has been found (1,2,4) when eddy-viscosity models were applied.

Results of predicted streamlines for the situation  $L/d = 0.25$ , investigated experimentally by Gosman and Ahmed [1] and with  $Re \approx 90,000$ , are shown in Fig. 3 for both versions of two-layer model. A particularly

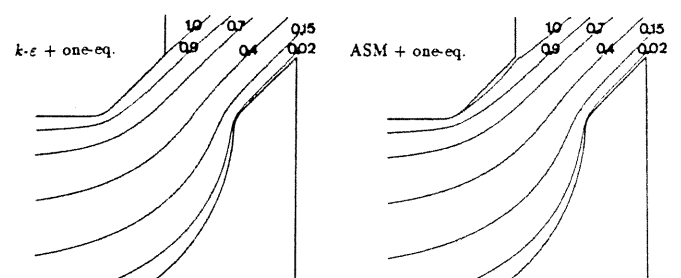


Fig. 3 Predicted streamline contours

interesting result that can be observed straight away is that the Two-L-ASM does predict a separated flow region on the valve seat whereas the Two-L- $k\epsilon$  does not show any sign of such trend.

Fig. 4 show results of velocity components  $U$  and  $V$  in the axial and radial directions, respectively, calculated with both models and compared with experimental data [1] at the inlet and exit of the valve passage (traverses 2 and 3 indicated in Fig. 1). Comparisons for other locations can be found elsewhere (15). It is clear from that figure that profiles calculated with the Two-L-ASM have improved markedly towards an agreement with the experimental data when compared to the Two-L- $k\epsilon$  results, particularly at the exit of the valve (traverse 3). This can be attributed to the separated flow region in the valve passage observed with the Two-L-ASM, already shown in Fig. 3.

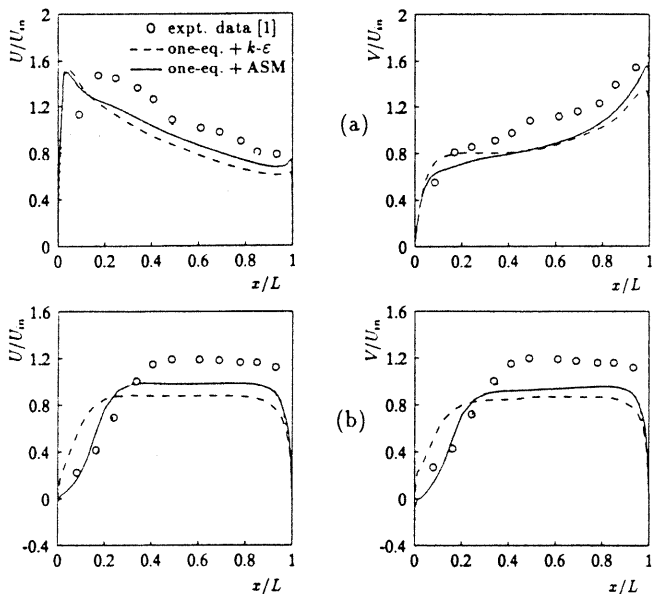


Fig. 4 Results of mean velocity profiles;  $L/d = 0.25$

- (a) Traverse 2  
(b) Traverse 3

The explanation for the distinct results predicted by the two models can be given with help of Fig. 5, where profiles of kinetic energy  $k$  and Reynolds shear stress  $\overline{uv}$  are plotted for traverses 1 and 2.<sup>2</sup> At the entrance of the valve, traverse 2, noticeable is the higher level of  $k$  and  $\overline{uv}$  returned by the Two-L- $k\epsilon$  in the core region and the better agreement with the experimental data displayed by the Two-L-ASM results. Without any doubt, the most significant variations occur for the Reynolds shear stress. Although on the valve surface both models calculate similar shear stress levels, close to the port

<sup>2</sup>The Reynolds shear stress  $\overline{uv}$  in the valve passage (traverse 2) is transformed to the directions parallel and normal to the valve and valve seat surfaces. For traverse 1, where no experimental data is available, from the valve up to the mid-point of the traverse  $\overline{uv}$  is transformed to the directions parallel and normal to the surface of the valve, and from the mid-point upwards in the axial and radial directions.

surface the Two-L-ASM model returns lower levels of  $\overline{uv}$  in the vicinity of the valve seat chamfer, which explains why in the case of the ASM model the flow separates more readily from the valve seat surface. This occurs due to the fact that in the ASM model the flow acceleration present close to the valve seat chamfer acts so as to decrease the shear stress level directly through its production term  $\mathcal{P}_{12}$ . Such effect of course cannot be accounted for in the eddy-viscosity formulation.

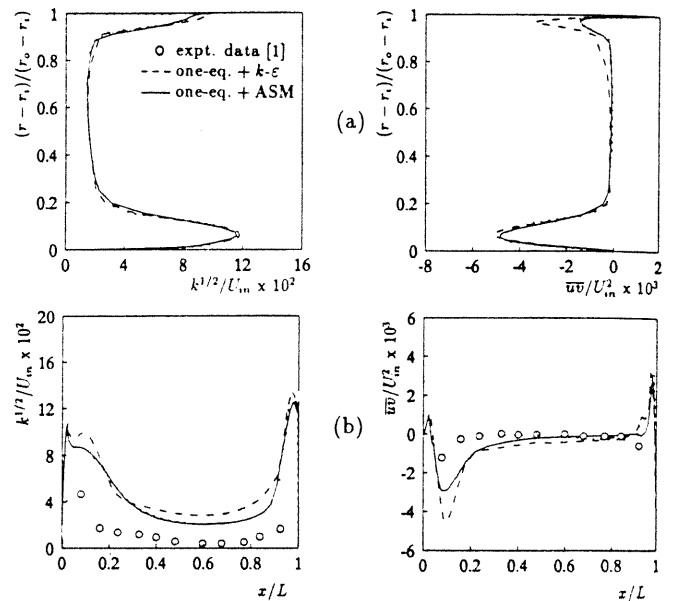


Fig. 5 Turbulence quantity profiles in the proximity of the valve seat chamfer;  $L/d = 0.25$

- (a) Traverse 1  
(b) Traverse 2

The two remaining flow situations have valve lifts  $L/d = 0.22$  and  $0.30$ ,  $Re \approx 7,000$ , and were investigated experimentally by Bicen et al (8). For these situations experimental data is available for velocity components ( $U$  and  $V$ ) and fluctuating velocities ( $\overline{u^2}$  and  $\overline{v^2}$ ) only from the plane at the exit of the valve passage onwards.

Comparisons of results with experimental data for both flow situations (Figs. 6 and 7) show that velocity profiles predicted by the Two-L-ASM at traverse 3 agree considerably better with the measurements. Nevertheless, the maxima of velocity are still underpredicted, and this is presumably due to a larger separated region observed experimentally.

One intriguing feature present in the numerical result for the  $V$  velocity component obtained with the Two-L-ASM is the change in the profile slope at  $x/L = 0.2$  in traverse 3 and which was traced to occur exactly at the interface between the one-equation model and the ASM model. As can be seen in Fig. 7 the same behaviour is verified even more markedly for the valve lift  $L/d = 0.30$ . The problem, observed only in that region, seems to be associated with substantial differences in the diffusion transport predicted there by both models and, therefore, could not be prevented.

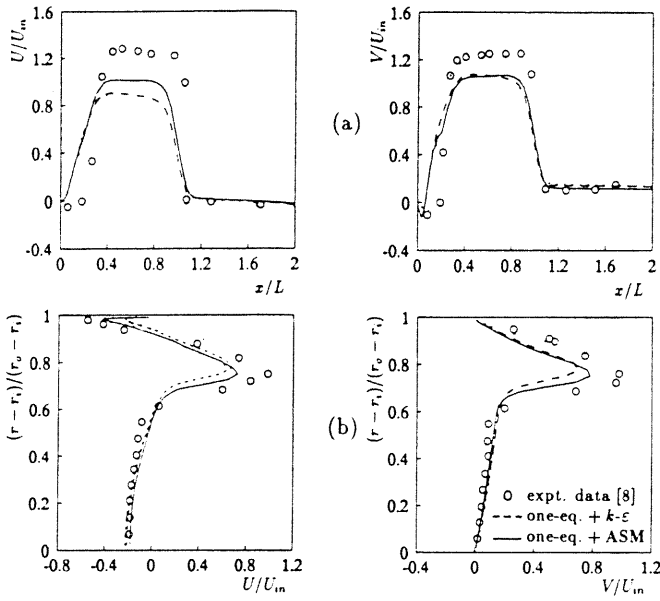


Fig. 6 Results of mean velocity profiles;  $L/d = 0.22$   
 (a) Traverse 3  
 (b) Traverse 4

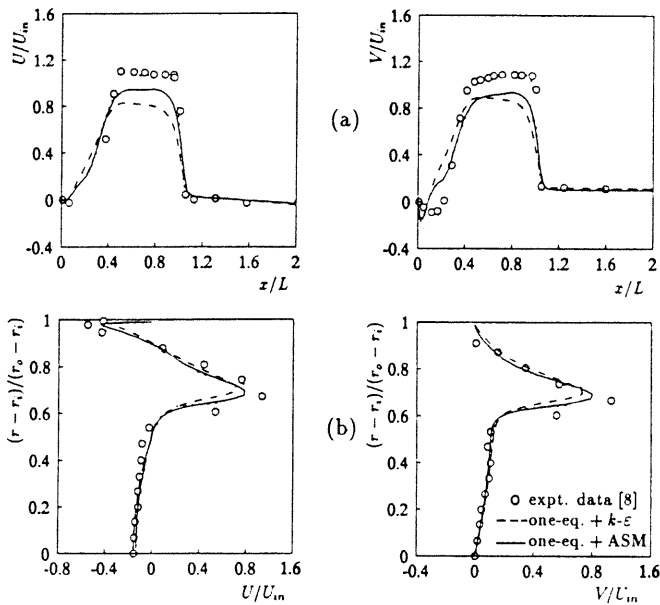


Fig. 7 Results of mean velocity profiles;  $L/d = 0.30$   
 (a) Traverse 3  
 (b) Traverse 4

For traverse 4, velocities components obtained by the Two-L-ASM are also in better agreement with the experimental data. The underprediction of the maxima of velocity calculated by both models at those traverses is a consequence of the reduced size of the separated flow region predicted in the valve passage. For radial positions

less than 0.5 at traverse 4, velocity profiles are shown to be very little affected by the different velocity profiles at the exit of the valve.

Turning attention to turbulence quantities, fluctuating velocities  $\overline{u^2}$  and  $\overline{v^2}$  given by the Two-L-ASM are seen to be closer to the measured levels for both traverses (Fig. 8). Nevertheless, it is disappointing to verify that in the recirculating region at traverse 4, the ASM model is unable to improve distributions of stresses towards agreement with the apparent isotropy condition indicated by the experimental data. The discrepancy, also observed for  $L/d = 0.22$ , could at first sight be thought to be related with shortcomings in the approximation of the stress transport in the ASM. However, Lea (9) found the same result using a Differential Stress Model (DSM), which does not use any simplification for the transport terms. The problem is more likely to be associated with the redistribution terms, equations (3) and (4), since it is precisely those terms which are responsible for redistributing energy between the fluctuating velocity components towards the isotropy condition.

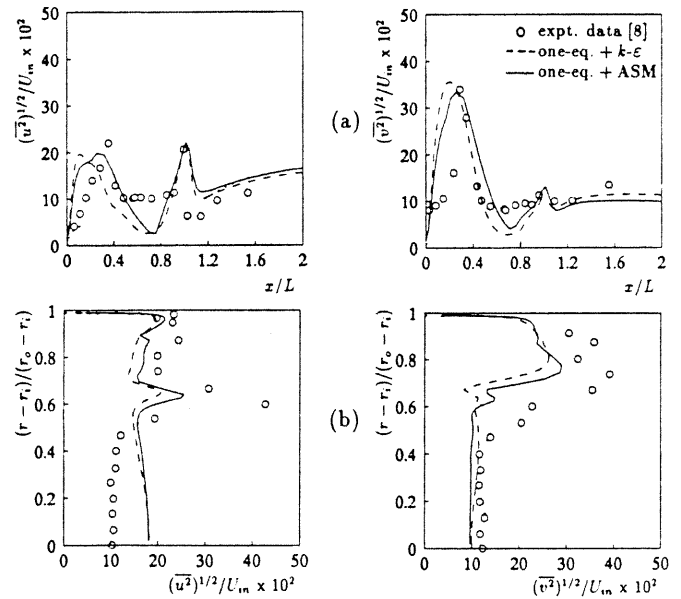


Fig. 8 Results of fluctuating velocity profiles;  $L/d = 0.30$   
 (a) Traverse 3  
 (b) Traverse 4

Finally, an assessment of the turbulence models in terms of global quantities is presented in Table 1, where values of discharge coefficient,  $C_d$ , calculated with the different versions of Two-Layer models are compared to experimental data. Overall the Two-L-ASM model is seen to give the best predictions of  $C_d$ , which is a consequence of its better capability to predict the separated flow region in the valve passage as observed experimentally.

### CONCLUSIONS

The ASM combined with the one-equation model in the near-wall region, denoted as Two-L-ASM, has been applied to the numerical analysis of the flow in port/valve

Table 1: Comparison between measured and predicted discharge coefficients  $C_d$ ; ASM and eddy-viscosity models.

	$L/d$		
	0.22	0.25	0.30
Exp. data (1,16)	0.53	0.50	0.45
Two-L- $k\varepsilon$	0.66	0.64	0.54
Two-L-ASM	0.63	0.60	0.50

assemblies for three large valve lifts;  $L/d = 0.22, 0.25$  and  $0.30$ . The assessment of the results obtained with the model was mainly carried out through comparisons with predictions obtained with the standard  $k-\varepsilon$  model in combination with the one-equation model, denoted as Two-L- $k\varepsilon$ , for the same flow conditions. The main conclusions from the analysis are:

i) The performance of the Two-L-ASM was verified to be superior to the Two-L- $k\varepsilon$  in all situations investigated.

ii) In the valve passage a clear improvement in the prediction of the separated flow region on the valve seat was verified with the Two-L-ASM, and was presumably originated by the lower level of Reynolds shear stress calculated by the model in the proximity of the valve seat chamfer surface.

iii) Values of discharge coefficient calculated with the Two-L-ASM when compared with those given by the Two-L- $k\varepsilon$  show to be in closer agreement with measurements and are a consequence of the improved results of the flow field in the valve passage verified with the former.

iv) The mean velocity and turbulence fields in the in-cylinder region, especially in the cylinder head region and further downstream along the cylinder wall, was found to be strongly affected by the velocity profile at the exit of the valve. On the other hand, in the large recirculating region beneath the valve levels of mean velocity and turbulence quantities showed very little sensitivity to the different flow conditions predicted at the exit of the valve by the models investigated in this work.

v) An acknowledged limitation of the present analysis is the necessity of an interface between two different turbulence models close to the walls. This practice can be expected to be particularly difficult to apply in the prediction of flows at reduced valve lifts or, in fact, in any situation where conditions accounted by both models prove to be far from each other in the proximity of the interface. A necessary step in further explorations with a second moment closure, if the full benefit of the more rigorous modelling is to be accomplished, should be the extension of the closure to the near-wall region.

## REFERENCES

- Gosman, A. D., and Ahmed, A. M. Y., "Measurement and Multidimensional Prediction of Flow in an Axisymmetric Port/Valve Assembly", SAE Paper 870592, 1987.
- Naser, J. A., "Flow Prediction in Valve/Port Assemblies", PhD Thesis, Dept. of Mech. Engng., University of London, 1990.
- Lilek, Z., Nadarajah, S., Peric, M., Tindal, M. J., and Yianneskis, M., "Measurements and Simulation of the Flow around a Poppet Valve", Eighth Symposium on Turbulent Shear Flows, Munich, 1991.
- Tsui, Y. Y., and Lee, S. Y., "Calculation of Turbulent Flow through Engine Inlet Ports", Int. J. Heat and Fluid Flow, Vol.13, No. 3, pp. 232-240, 1992.
- Aita, A. T., Munck, G., Montmayer, N., Takenaka, Y., Aoyagi, Y., and Obana, S., "Numerical Simulation of Swirling Port-Valve Cylinder Flow in Diesel Engines", SAE Paper 910263, 1991.
- Haworth, D. C., El Tahry, S. H., Huebler, M. S., and Chang, S., "Multidimensional Port-and-Cylinder Flow Calculations for Two- and Four-Valve-per Cylinder Engines: Influence of Intake Configuration on Flow Structure", SAE Paper 900257, 1990.
- Le Coz, J. F., Henriot, S., and Pinchon, P., "An Experimental and Computational Analysis of the Flow Field in a Four-Valve Spark Ignition Engine—Focus on Cycle Resolved Turbulence", SAE Paper 900056, 1990.
- Bicen, A. F., Vafidis, C., and Whitelaw, J. H., "Steady and Unsteady Air Flow through an Intake Valve of a Reciprocating Engine", Trans. of the ASME—J. Fluids Engng., Vol. 109, pp. 3-35, 1987.
- Lea, C. J., "Second-Moment Closure for In-Cylinder Flow Calculations", PhD Thesis, Faculty of Technology, UMIST, 1994.
- Rodi, W., "A New Algebraic Stress Relation for Calculating the Reynolds Stresses", Z. Ang. Math. und. Mech., Vol 56, p. 219, 1976.
- Craft, T. J., "Second Moment Modelling of Turbulent Scalar Transport", PhD Thesis, Faculty of Technology, UMIST, 1991.
- Iacovides, H., "Momentum and Heat Transfer in Flow through 180° Bends of Circular Cross Section", PhD Thesis, UMIST, 1986.
- Launder, B. E., and Spalding, D.B., Mathematical Models of Turbulence, Academic Press, 1972.
- Kadja, M., "Computation of Recirculating Flow in Complex Domains with Algebraic Reynolds stress closure and Body-Fitted Meshes", PhD Thesis, Faculty of Technology, UMIST, 1987.
- Deschamps, C. J., "Modelling of Turbulent Flow through I.C. Engine Inlet Port and Valve Passages", PhD Thesis, Faculty of Technology, UMIST, 1993.
- Vafidis, C., and Whitelaw, J. H., "Steady and Pulsating Air Flow through a Stationary Intake Valve of a Reciprocating Engine", Report FS/84/04, Dept. of Mech. Engng., Imperial College of Science and Technology, 1984.