Flow through catalytic converters - laser Doppler anemometry versus computational fluid dynamics

Girgis, N.S. , Benjamin, S.F. , Jasper, T.S. , Cuttler, D.H. and Davies, M.J.

Published version deposited in CURVE February 2014

Original citation & hyperlink:

Girgis, N.S. , Benjamin, S.F. , Jasper, T.S. , Cuttler, D.H. and Davies, M.J. (1993). Flow through catalytic converters - laser Doppler anemometry versus computational fluid dynamics. In *Proceedings of the Ninth International Conference on Systems Engineering.* University of Nevada, Las Vegas.

http://www.icseng.com/previous.php

PROCEEDINGS OF THE NINTH INTERNATIONAL CONFERENCE ON SYSTEMS ENGINEERING

July 14-16, 1993

Available From:
Howard R. Hughes College of Engineering
University of Nevada, Las Vegas
Box 454005
Las Vegas, NV 89154-4005

UNLV
UNIVERSITY OF NEVADA LAS VEGAS

Flow Through Catalytic Converters - Laser Doppler Anemometry versus Computational Fluid Dynamics

Dr N S Girgis, Dr S F Benjamin and
T S Jasper — Coventry University
Dr D H Cuttler — Jaguar Cars
Dr M J Davies — UKAEA Harwell

Abstract

Using laser Doppler anemometry (LDA) techniques, the flow velocities have been measured for a sudden pipe expansion into a square duct containing a catalyst situated downstream of the expansion. This geometry has been modelled using the computational fluid dynamics (CFD) code FLOW3D from Harwell.

The results of the LDA data have been compared to those predicted by the CFD code. The CFD simulations are close to the experimental data though the CFD results predict more mal-distribution than observed with the LDA technique.

Changing the turbulence modelling from the k-epsilon model to the Algebraic Reynolds Stress model improved the predictions significantly, with the magnitude of the recirculation matching the experimental data more closely. It was also found that the solution was not wholly grid independent, though variations were small.

Introduction

Over the past 20 years since the introduction of legislation requiring catalysts to be fitted to gasoline fuelled vehicles, researchers and engineers alike have been concerned about the flow distribution through catalysts. Researchers Lemme and Givens [1] and Howitt and Sekella [2] were early investigators into flow distributions, whilst Wendland and Matthes [3] provided flow visualisation data to demonstrate the poor distribution within a typical oval catalyst geometry.

It has been stated many times that the non-uniform flow patterns lead to under-utilisation of some parts of the catalyst and over-utilisation, with consequent early deterioration, of other areas. In recent years, designers have turned their attention to improving this area of catalyst performance through a variety of experimental and theoretical techniques.

Experimental analysis techniques include Pitot tube measurements of the dynamic head of the gas and LDA measurements of the velocities in particle seeded flows. Theoretical predictive techniques use computational fluid dynamics to calculate the velocity and pressure field under investigation.

This paper describes the comparison of the CFD predictions using the Harwell FLOW3D code to the LDA data measured using a square tube and catalyst arrangement. The experimental

technique is described in detail in the paper N Jorgensen, M J Davies *et al* [4] presented at ISATA in June 1992, and a schematic of the experimental layout is given in figure 1.
There are particular problems for CFD codes when modelling these catalyst type flows i.e. a large resistance just downstream of an expansion. For computational ease, the catalyst is usually modelled as porous media - using multiple computational cells per channel would lead to *very* large grids.

Such a porous media approach requires the ability to generate one-dimensional flow within the porous media and a flow resistance in that direction that is based on the laminar viscosity and channel velocity. Entrance effects are usually ignored, the channel properties approximated and modelling the turbulent to laminar transition using a porous media model is currently impossible. The technique for modelling catalyst flows using FLOW3D is detailed in the paper by Jasper *et al* [5] also presented at ISATA in June 1992.

Results from the LDA technique

The volume flowrate for the tests was 0.025 m^3/s which equated to a mean pipe velocity of 12.7 m/s and a mean square duct velocity of 2.5 m/s.

Figure 2 is a 3-D picture of the experimentally measured velocity vectors 50 mm and 5 mm upstream of the catalyst. Figure 3 shows the FLOW3D predictions of the velocity vectors for the same positions, as well as the velocity field as it emerges from the inlet pipe. Visually, the patterns look very similar at the two stations, with the higher grid density of the CFD data showing things more clearly.

Detailed analysis of the LDA data at a plane mid-way between the expansion and the catalyst front face shows that the velocities are extremely mal-distributed (see figure 4). The peak observed velocity of 11.2 m/s occurs just below the centre whilst the recirculation flow exhibits velocities of -3 m/s (cf. V_{bulk} 2.5 m/s). The contour plots show some asymmetry which, although the inlet tube and duct were carefully aligned, indicates how sensitive the flow is to geometry. The recirculation zones occur not just in the square corners, but about the entire periphery.

By the time the flow has moved to within 5 mm of the front face of the catalyst (figure 5), the velocity distribution is much flatter, though the peak velocity at nearly 6 m/s is still over twice the bulk velocity. The flow exhibits strong radial momentum here, with the radial and axial velocities of approximately the same magnitude. This indicates that the flow impinges on the front face of the catalyst at angles of 45 degrees or more.

Results from the FLOW3D modelling.

The CFD modelling entailed the use of a quarter section model with two symmetry planes, and a full three-dimensional model. The computational time and expense of running the 47500 cell full model meant that most of the development work was undertaken on the quarter section model.

Figures 6 and 7 illustrate the full section of the test geometry. Taking the plane mid-way between the expansion and the front face of the catalyst (figure 6), the CFD data can be compared to the LDA results. The peak velocity predicted by the CFD model is higher than the LDA data, though the contours of velocity posses the same shape. The magnitude of the recirculation (peak negative velocities of circa 3 m/s) compares well with the experimentally observed data. In figure 7, the velocity contours predicted are higher than the LDA data ($V_{max} > 6$ m/s), and even predict a small recirculation zone near the walls which, owing to the relatively coarse grid used, may not have been picked up by the LDA.

The secondary velocity vectors from the quarter section model 5 mm away from the catalyst are shown in figure 8. They show in more detail the recirculation patterns occurring here, with higher velocities ($V_{max} = 5.9$ m/s) than those measured by the LDA.

The full model used the k-epsilon model whilst the quarter section model compared the k-epsilon model with the Reynolds Algebraic Stress model. The results for the plane 50 mm upstream of the catalyst (figures 9 and 10) indicate that the area of recirculation is smaller for the Reynolds Stress model than the k-epsilon model. The contour of zero (0.0) velocity compares very favourably with that for the LDA. At the station 5 mm from the catalyst (figures 11 and 12), the Reynolds stress model has a lower peak velocity (just under 6.0 m/s) which compares extremely well with the LDA data (circa 6.0 m/s).

The maximum velocity predicted for the secondary flow at this station (figure 8) is just under 6 m/s, occurring mid-way between the centre line and the wall. This is well in excess of the axial velocity at this point, and somewhat higher than the experimentally observed values.

The velocity vector field along one of the planes of symmetry is shown in figure 13. It can be seen that the recirculation zone on the centre line only occurs in the second half of the expansion. The induced flow generated from the recirculation in the corners is in the direction of the bulk flow. At the wall (figure 14), the recirculation goes all the way back to the expansion wall and shows very large tangential velocities that are virtually perpendicular to the catalyst face

Modifying the grid on the quarter section model to improve resolution in the recirculation areas changed the velocity field slightly from the full model, indicating more work is needed to obtain a grid-independent solution. Limitations on computational time and memory meant that a better grid could not be tried.

Conclusions

The experimentally determined flow patterns exhibit significant flow mal-distribution, but the CFD predicted data indicates slightly larger areas of recirculation, with higher peak to trough values.

The k-epsilon turbulence model doesn't predict flow separation regions as well as the Reynolds Stress model. There is little experimental data with which to compare and validate the predictions but these results suggest that CFD may be used to model catalyst flows if the more sophisticated turbulence models are used.

Acknowledgements

The authors would like to acknowledge the Science and Engineering Research Council for funding this research at Coventry University through Grant No. GR/F/88797, Jaguar Cars Ltd and the UK Atomic Energy Authority for the experimental data.

References

[1] Lemme C. D. and Givens W. R., "Flow Through Catalytic Converters - An Analytical and Experimental Treatment", SAE 740243

[2] Howitt J. S. and Sekella T. C., "Flow Effects in Monolithic Honeycomb Automotive Catalytic Converters", SAE 740244

[3] Wendland, D. W. and Matthes, W. R., "Visualisation of Automotive Catalytic Converter Internal Flows", SAE 861554

[4] Jorgensen N., Davies M. J., Dewhurst S. J. and Goatcher T., "Visualising Flow Distributions Through Catalytic Converters", presented to the 25th International Symposium on Automotive Technology and Automation, ISATA, Florence, June 1992

[5] Jasper T. S., Benjamin S. F., Girgis N. S. and Cuttler D. H., "Computational Fluid Dynamics and its Application to Catalyst Exhaust Systems", presented to the 25th International Symposium on Automotive Technology and Automation, ISATA, Florence, June 1992

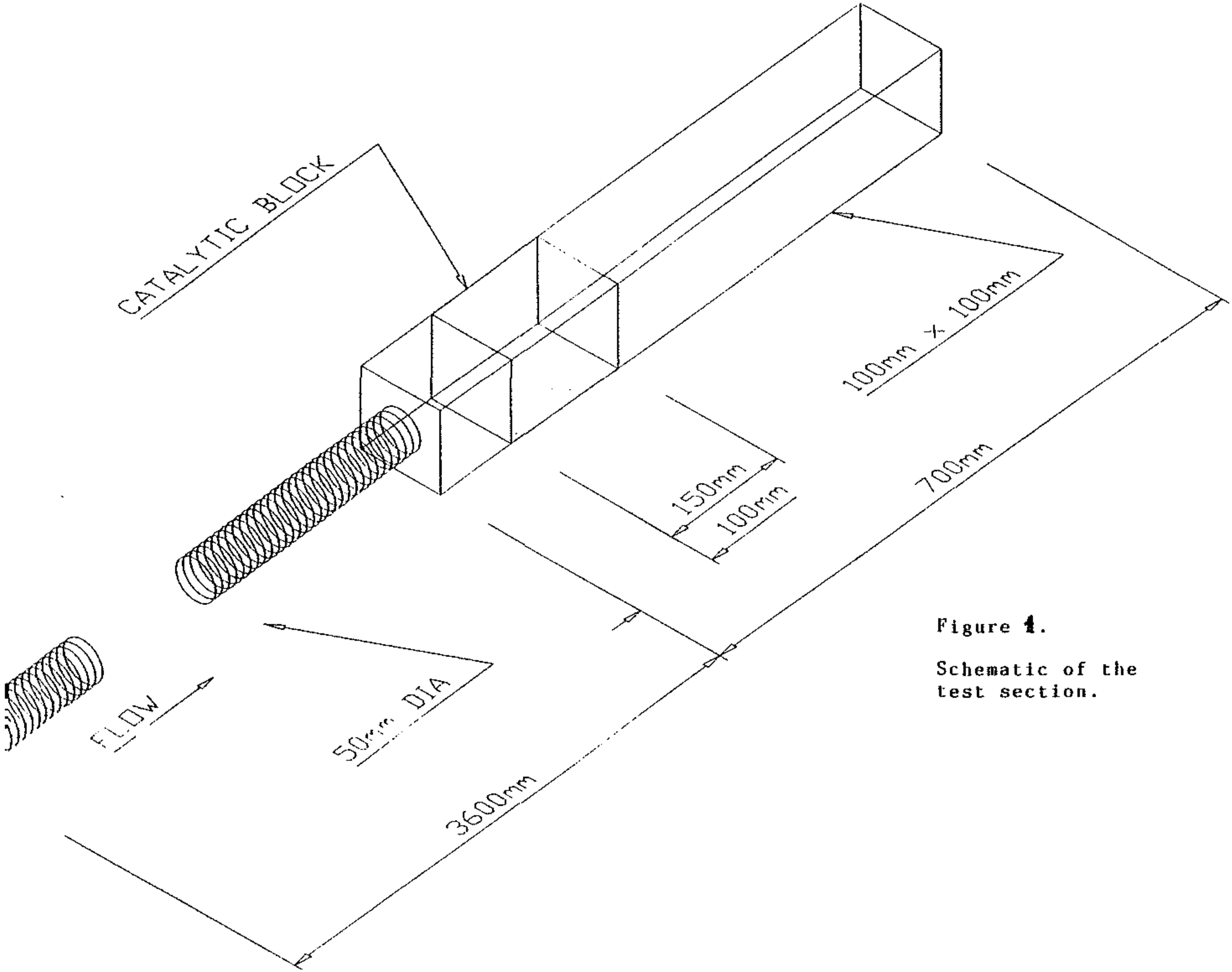


Figure 1.

Schematic of the test section.

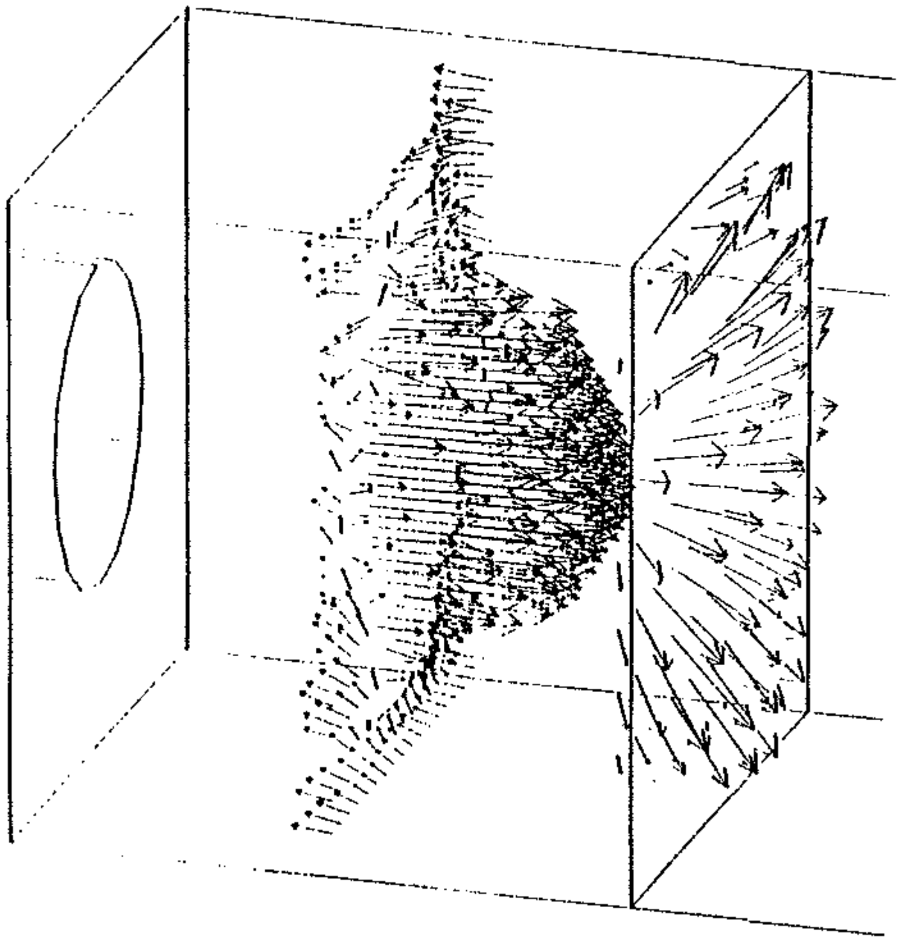


Figure 2. Laser Doppler measured velocity velocity vectors

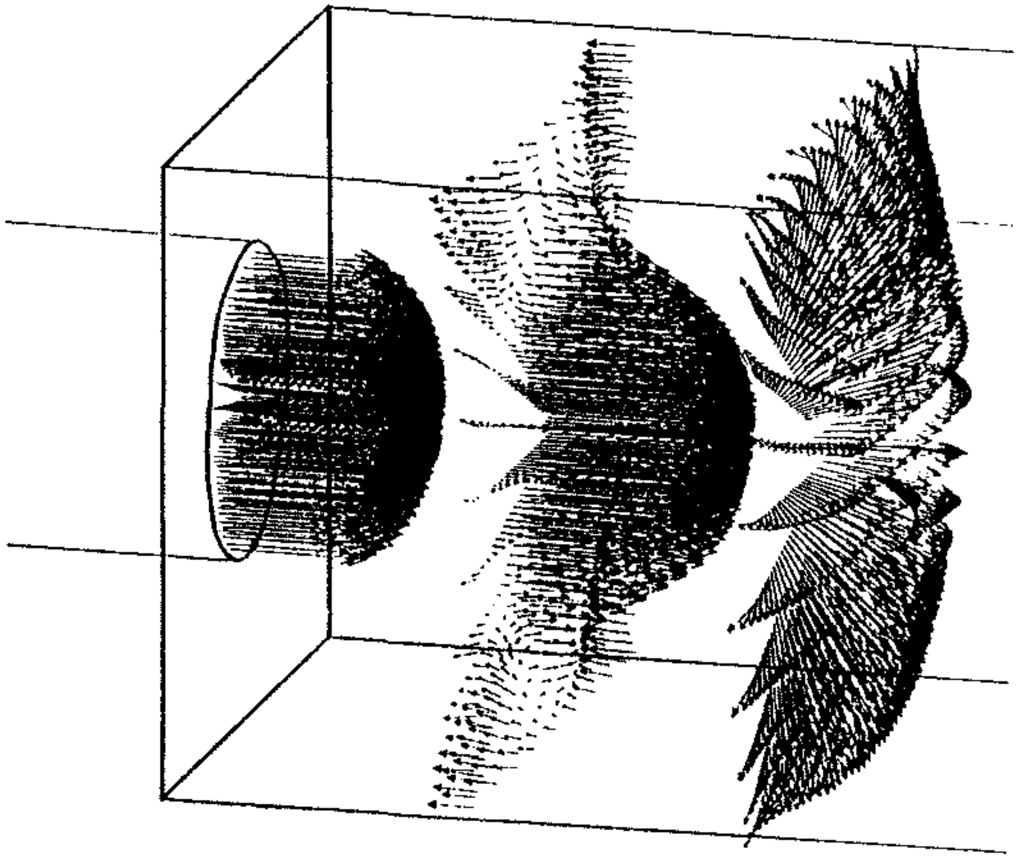


Figure 3. Computational Fluid Dynamics predicted velocity vectors

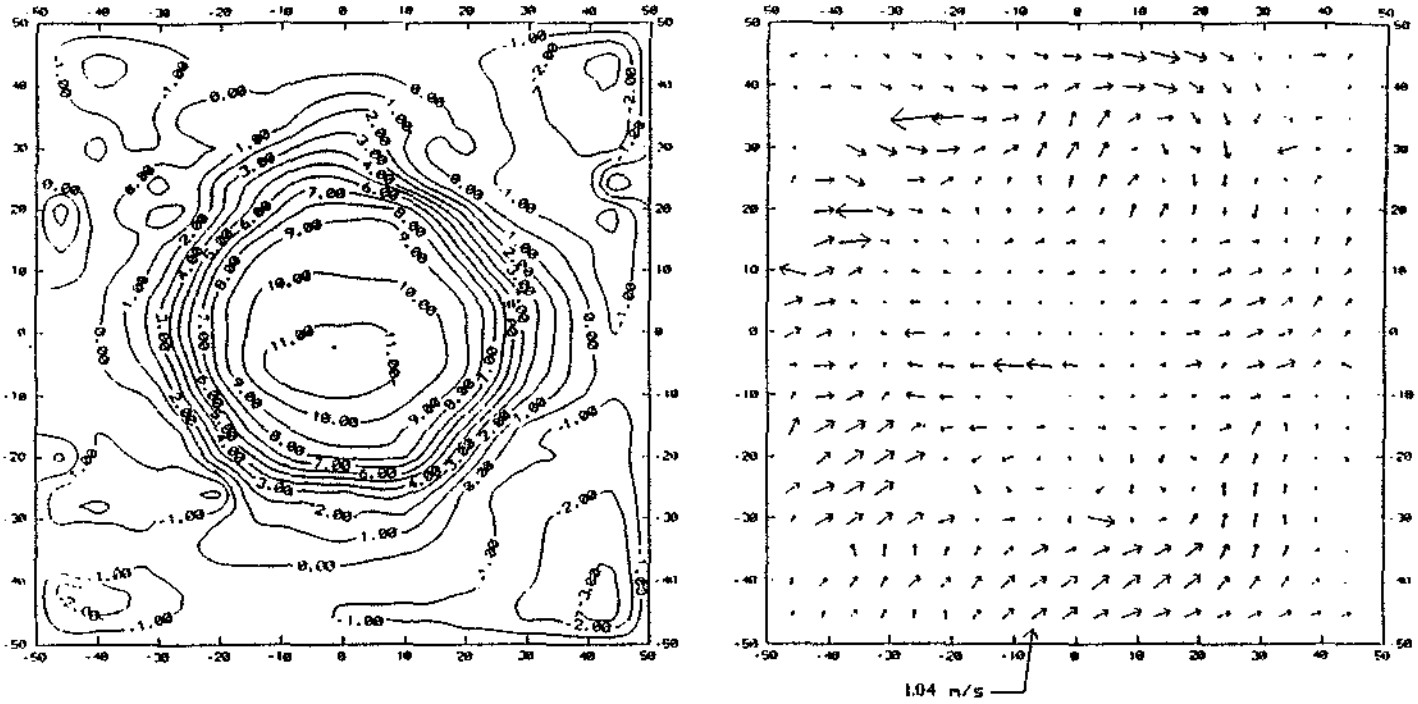


Figure 4. Axial and secondary velocities 50mm upstream of the converter.

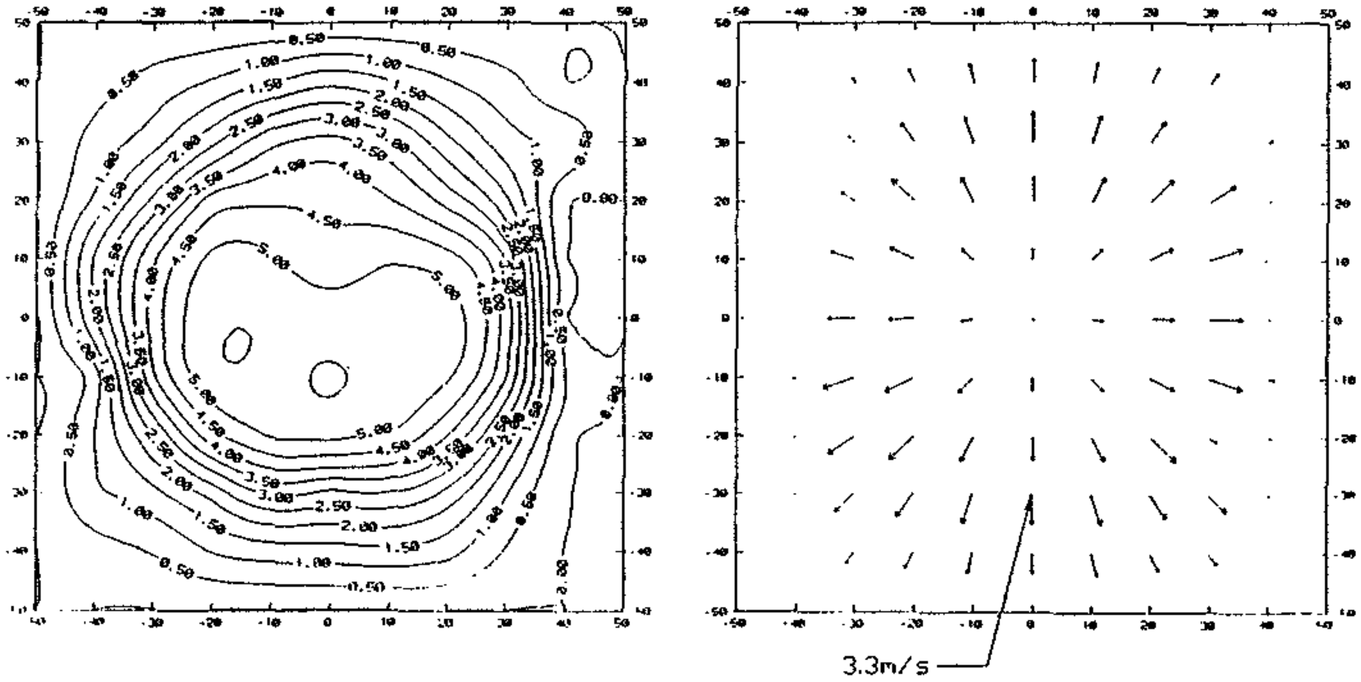


Figure 5. Axial and secondary velocities 5 mm upstream

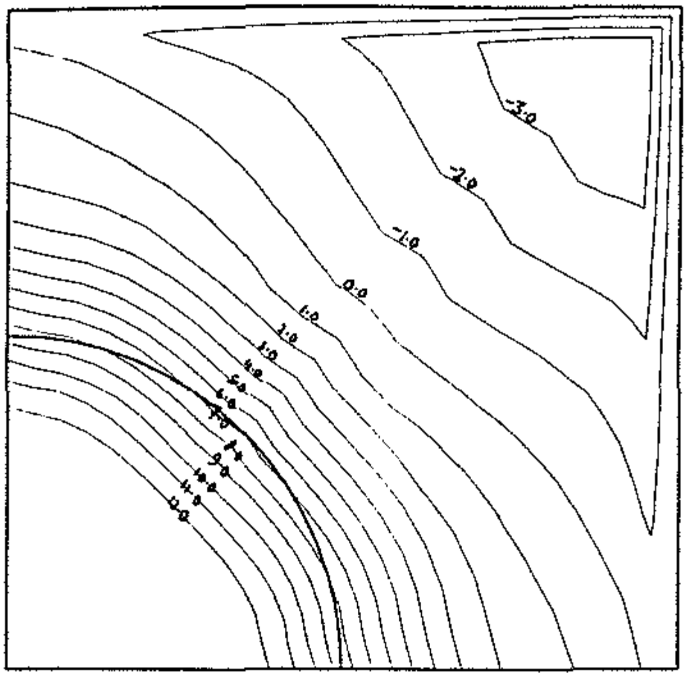


Figure 9 Axial velocity contours for the K-epsilon model 50 mm upstream of the catalyst (m/s)

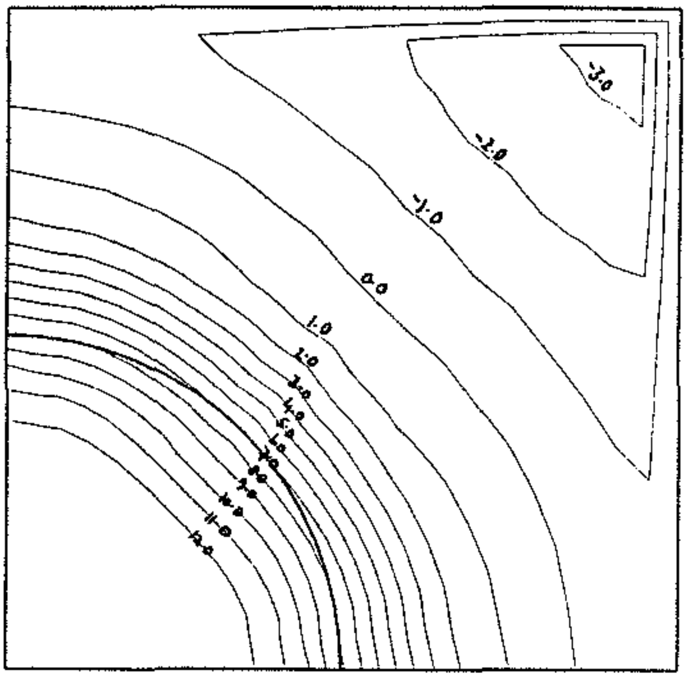
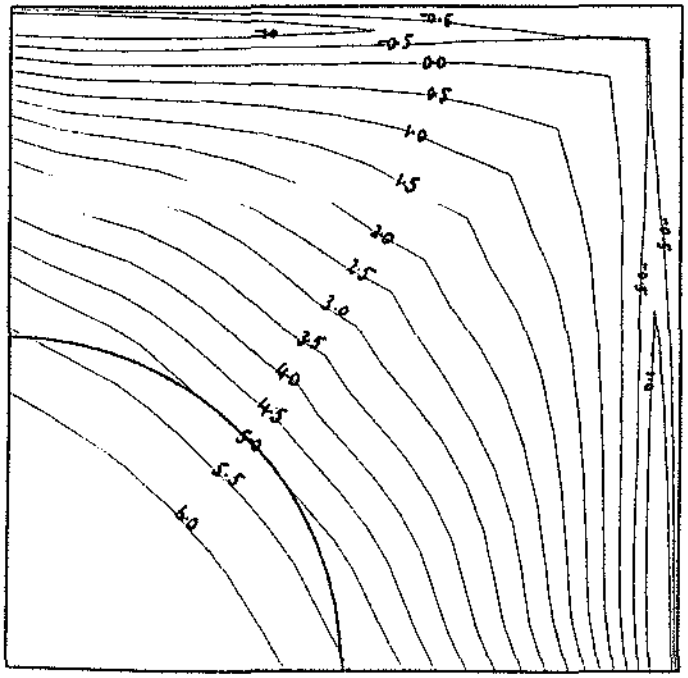
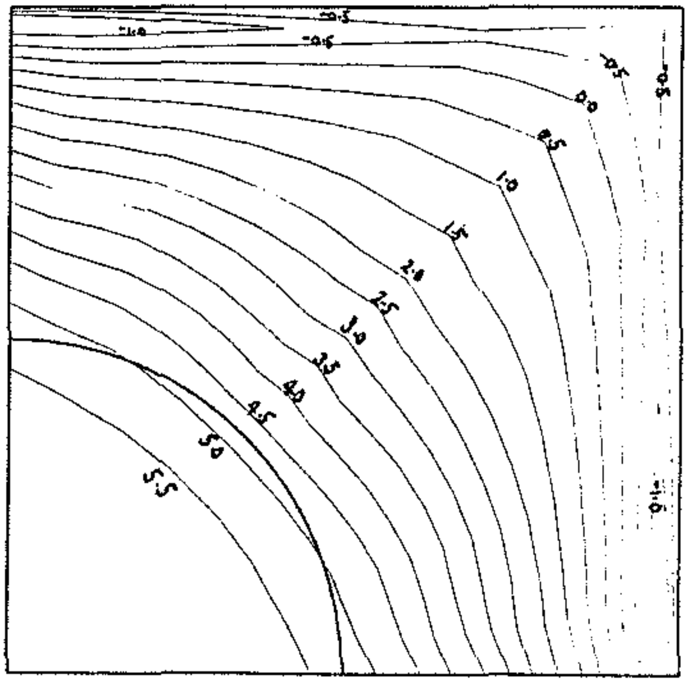


Figure 10 Axial velocity contours for the Reynolds stress model 50 mm upstream of the catalyst (m/s)





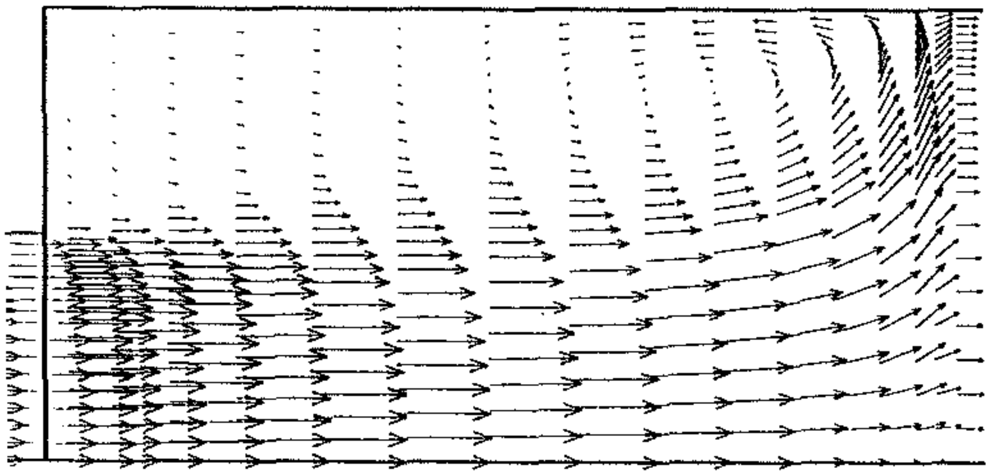


Figure 13 Velocity vectors for the Reynolds stress model along the centre plane of symmetry (m/s)

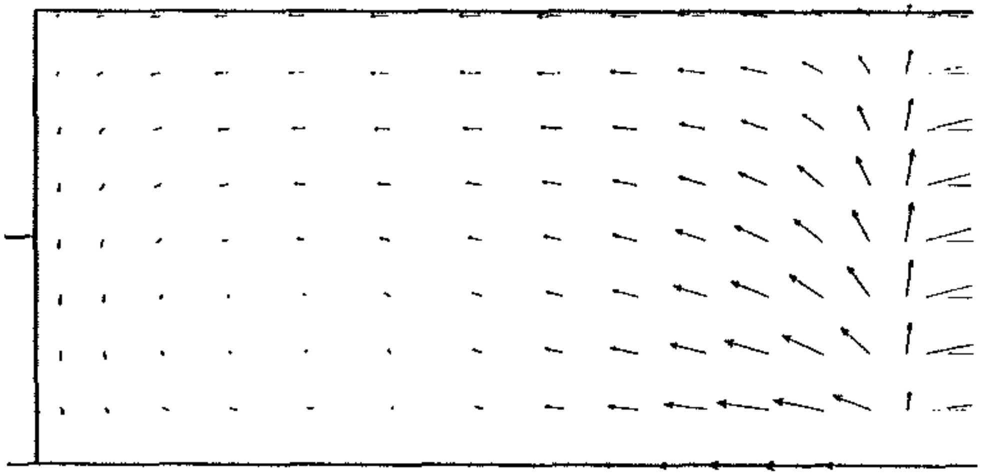


Figure 14 Velocity vectors for the Reynolds stress model 0.5 mm from the far wall (m/s)