ARL-TN-38 AR-008-338

AD-A268 731

DEPARTMENT OF DEFENCE

DEFENCE SCIENCE AND TECHNOLOGY ORGANISATION

AERONAUTICAL RESEARCH LABORATORY

MELBOURNE, VICTORIA

Technical Note 38

APPLICATIONS OF THE PROGRAM UIFLOW-2D TO THE NUMERICAL SIMULATION OF AXISYMMETRIC FLOWS

DISTRIBUTION STATEMENT A
Approved for public release;
Distribution Unlimited

by

D. DAVIS
E. BRIZUELA

DTIC
S ELECTE D
AUG 3 0 1993
B

93-20044

Approved for public release.

© COMMONWEALTH OF AUSTRALIA 1993

JUNE 1993

93 8 26 072

This work is copyright. Apart from any fair dealing for the purpose of study, research, criticism or review, as permitted under the Copyright Act, no part may be reproduced by any process without written permission. Copyright is the responsibility of the Director Publishing and Marketing, AGPS. Enquiries should be directed to the Manager, AGPS Press, Australian Government Publishing Service, GPO Box 84, CANBERRA ACT 2601.

AR-008-338

DEPARTMENT OF DEFENCE
DEFENCE SCIENCE AND TECHNOLOGY ORGANISATION
AERONAUTICAL RESEARCH LABORATORY

Technical Note 38

APPLICATIONS OF THE PROGRAM UIFLOW-2D TO THE NUMERICAL SIMULATION OF AXISYMMETRIC FLOWS

by

D. DAVIS*
E. BRIZUELA

SUMMARY

The computer program UIFLOW-2D has been applied to the numerical simulation of turbulent axisymmetric flows.

In a preliminary test of the performance of the program it has been used to simulate four different types of flows, isothermal and reactive, with and without swirl. Numerical results are discussed and compared with published experimental data. Recommendations are made for future development and application to gas turbine combustors.

*Department of Mechanical Engineering, The University of Sydney, NSW 2006.

DSTO
AUSTRALIA

© COMMONWEALTH OF AUSTRALIA 1993

POSTAL ADDRESS: **Director, Aeronautical Research Laboratory,**
506 Lorimer Street, Fishermens Bend, 3207
Victoria, Australia.

TABLE OF CONTENTS

DTIC QUALITY INSPECTED 3

Accession For	
NTIS GRA&I	☑
DTIC TAB	☐
Unannounced	☐
Justification	
By	
Distribution/	
Availability Codes	
Dist	Avail and/or Special
A-1	

1. INTRODUCTION

The computer code UIFLOW-2D [1] has recently been acquired by ARL. This computational fluid dynamics code possesses many desirable features for the purpose of numerical simulation of combustion in conditions typical of gas turbine combustors. Hence, the code is being developed and adapted for this task.

An initial part of this effort is the installation of the code in one of the available local computers, adaptation of the code to the local system, and general code testing and debugging. This has been done by means of simulating non-reactive and reactive flows in ducts and assessing the results against comparable published data. It must be emphasized that the work reported herein constitutes only a preliminary assessment of the performance of the code and, being largely qualitative in nature, does not represent a full validation of the code, which is a continuing task.

The following four cases of flow have been computed:

- The turbulent flow of air in a smooth pipe;
- Turbulent non-reacting coflowing jets with no swirl in either of the inlet flows;
- As above plus swirl on the external (annular) jet flow; and
- Turbulent reacting coflowing jets (i.e., a gas diffusion flame) with swirl, under the above conditions.

2. THE UIFLOW-2D CODE

2.1 Description

The computer code UIFLOW-2D and its associated literature were obtained in 1992 from the Aero Propulsion and Power Directorate of the USAF Wright Laboratory, under the auspices of the Mutual Weapons Development Data Exchange Agreement, Annex 7017.

This code can simulate the flow of turbulent or laminar, isothermal or reacting flows in complex two-dimensional geometries (planar or axisymmetrical). It is based on the discretisation, using finite-volume techniques, of the steady-state, averaged conservation equations. Mass-weighted [2] averages are used, and equal species diffusivities are assumed.

The code employs body-fitted coordinates in a co-located grid (i.e., all variables are assumed located at the cell centre). Discretisation of the partial differential equations which govern the flow is done using the unconditionally stable hybrid differencing scheme. A multigrid scheme is included, to accelerate convergence.

System closure for turbulent flow is done using the standard $\kappa - \epsilon$ model plus wall functions near solid walls.

The turbulent diffusion flame model is based on the one-step, fast chemistry model of Bilger [3]. At present UIFLOW does not cater for non-adiabatic flames, radiation effects or fuel sprays.

2.2 Applicability and desirable features

UIFLOW was designed specifically for the simulation of reactive flows in ducts. Hence, it incorporates many highly desirable features, such as

- mass-weighted (Favre) formulation, suitable for flows with large variations in density such as flames;
- hybrid differencing scheme which, at the low velocities typical of gas turbine combustors, becomes an implicit, central differences scheme, desirable for strongly recirculating flows;
- compliance with the equation of state (as opposed to other codes in which the absolute levels of pressure are not related to density); and
- use of boundary-fitted coordinates, necessary for complex curved boundaries such as exist in practical gas turbine combustors.

Another attractive feature is the code structure: the code is highly modular, comprising nearly 100 short subroutines each mostly free of special cases and IF statements. This greatly simplifies implementation of new models, and is computationally efficient.

2.3 Present Status

The applications reported here have exercised most areas of the code, the computation of premixed flames being a major exception (although not applicable to gas turbines). As a result, the code is now usable for the simulation of ducted gas diffusion flames assuming one-step reactions and fast chemistry. The code can handle swirling flows, high pressures and recirculation zones.

3. TURBULENT FLOW IN A SMOOTH PIPE

3.1 Test case

The geometry used in this case is that tested by Barbin and Jones [4]. This case was chosen because the published experimental results include both velocity profiles and contours. The duct is an axisymmetric pipe with a diameter of 0.2032 m and a length of 8.832 m (45 diameters).

A grid of 120 x 48 cells was used for this simulation, with two grid levels. Cell length along the x-axis was increased in a geometric progression, while the cell height (along the y-axis) was constant except for a local refinement near the pipe wall (Figure 1). This grid geometry provides a fine grid of low aspect ratio at the inlet and transition regions and at the boundary layer region.

Inlet conditions for the numerical simulation were assumed radially uniform and are listed below:

Velocity:	20 m/s
Pressure:	100 kPa
Temperature:	300 K
Turbulent kinetic energy (TKE):	0.04 m^2/s^2
Dissipation of TKE:	0.2 m^2/s^3

3.2 *Simulation results*

Selected results of the simulation are shown in Figures 2 and 4. Corresponding experimental results (adapted from [4]) have been included for comparison (Figures 3 and 5).

3.3 Comparison with experiment and discussion

The overall shapes of the velocity profiles (Figure 2) and variation of velocity (Figure 4) are not dissimilar to those of the experimental results (Figures 3 and 5 respectively). However, there are some discrepancies.

The first discrepancy is the peak in the velocity profile near the pipe wall at x/D=1.5 (Figure 2), which can also be identified in Figure 4 as a peak near the inlet. This can not be seen in the experimental results.

Predicted velocity values are also slightly higher, at intermediate radii, than those from the experimental results.

Both discrepancies are attributed to the fact that UIFLOW can not simulate transitional boundary layers. As the boundary layer is assumed turbulent even in the inlet region, the greater blockage results in higher axial velocities, first near the wall and later at all radii.

Allowing for the above considerations, simulation results are considered very satisfactory.

4. TURBULENT COAXIAL JETS

4.1 Numerical and Experimental Cases

Three numerical simulations were performed which allowed separate and progressive testing of the operation of the main models in the code, as follows:

- Axisymmetric coaxial jets of air without swirl and with a backwards facing step, to test the ability of the code to simulate recirculating mixing flows,
- The same with swirl, to test the swirl model, and
- A gas diffusion flame with swirl under the same conditions as above, to test the flame model.

The geometric and aerothermodynamic data used for these simulations was based on the experiments of Spadaccini et. al. [5], in which a central jet of methane fuel mixed and burned in a coaxial flow of air; results were reported in [5] with and without swirl in the annular coaxial air flow. Since these experiments always involved reaction and heat release the results are only loosely relevant to the two non-reacting numerical simulations ; nevertheless they were useful in assessing the plausibility of the numerical results, particularly near the inlets. For the non-swirling case the main differences were expected to be noted in the flame region, and to reside in that combustion provides a rise in temperature and thus expansion and an axial acceleration; densities must also be expected to be quite different. For the case with swirl additional differences were expected, although these are more difficult to envisage. Thus, the second simulation should also provide some insights into the effect of combustion on swirling flow fields.

The third simulation is directly comparable to the experiment.

The geometry used for the simulation is shown in Figure 6, whilst Figure 7 shows a diagram of the apparatus used in the experiment. The swirler blades were removable, and experimental data with and without swirl is reported in Ref [5]. Experimental inlet conditions comprised a central 1 m/s flow of methane gas to a radius of 0.032 m, an annular 20 m/s airflow between radii 0.032 m and 0.047 m and a downstream facing step from 0.047 m to the pipe wall at 0.061 m.

A 120 x 34 grid was used in all simulations, similar in construction to that described in Section 3.1.

4.2 Non-reacting jets without swirl

4.2.1 Test case

The thermodynamic state of the substitute air inner jet was specified to match the original fuel jet density and pressure. Inlet conditions are:

	Inner Jet	**Outer Jet**
Velocity:	1 m/s	20 m/s
Temperature:	600 K	751 K
Density:	2.29 kg/m^3	1.83 kg/m^3
TKE:	0.004 m^2/s^2	1.0 m^2/s^2
Dissipation of TKE:	0.27 m^2/s^2	300.0 m^2/s^2

4.2.2 Simulation results

Axial velocity contours and turbulent kinetic energy contours from the numerical simulation are shown in Figures 8 and 10; experimental results, adapted from [5], are shown in Figures 9 and 11. A velocity vector plot and mass flow rate contours of the numerical solution are given in Figures 12 and 13.

4.2.3 Discussion

Taking into account previous considerations regarding reacting and non-reacting flows, the numerical results compare reasonably well with the experimental data. Both recirculation zones have been satisfactorily predicted (Figures 8 and 9), and the location of the remaining velocity contours are qualitatively similar to those from the experiment. The non-reacting simulation exhibits lower acceleration along the centreline as expected.

The experimental results in Figure 11 are in the form of contours of rms fluctuation of axial velocity. Direct comparison with the numerical results is not possible, since turbulent fluctuations of one component of velocity are not readily available from the numerical solution. Nevertheless, contours of turbulent kinetic energy (the sum of the squares of axial and radial rms fluctuations), plotted in Figure 10. show that areas of high turbulence are predicted correctly in position and approximately in value.

Overall, the code performance in simulating this case is considered satisfactory.

4.3 Non-reacting jets with swirl

4.3.1 Test case

The test case used in the simulation of swirl is an extension of the case with no swirl (Section 4.2). In the experimental set-up swirl was generated by means of radial vanes (Figure 7); the swirl velocity thus generated was said to be of the order of

10 m/s, but inlet swirl profiles were not reported, which is unfortunate since swirl flows are known to be very sensitive to initial conditions. For the purposes of the numerical simulation the distribution of swirl velocity in the annulus was adjusted until results of swirl at downstream locations agreed with the experimental data. All other input parameters remained unchanged from Section 4.2.

4.3.2 Modifications needed to UIFLOW-2D for convergence

Very slow convergence was observed in the simulation of this case, compared with the previous one without swirl. This slowness was at first attributed to the inlet axial and swirl velocities having been specified as uniform in the radial direction. In an attempt to rectify this, the program was modified to read prescribed velocity distributions and interpolate for values at cell centres, in itself a facility worth adding. However, this provided only a marginal improvement in convergence .

The reason for the inability to reduce residuals was investigated by means of an analysis of intermediate results and a detailed study of the code listing. It was found that certain components of the source terms for the axial and radial momentum equations had been left out. The transport equation of a variable Φ is written in cylindrical polar coordinates as:

$$\frac{1}{r}\left[\frac{\partial}{\partial x}(\rho u r\Phi) + \frac{\partial}{\partial r}(\rho v r\Phi)\right] = \frac{1}{r}\left[\frac{\partial}{\partial x}(r\Gamma_\Phi\frac{\partial\Phi}{\partial x}) + \frac{\partial}{\partial r}(r\Gamma_\Phi\frac{\partial\Phi}{\partial r})\right] + S_\Phi \qquad (1)$$

where x, r are axial and radial coordinates, ρ is density, u and v are axial and radial velocity, Γ_Φ is the diffusivity coefficient (e.g., the turbulent viscosity divided by the Prandtl number) and S_Φ is the source term. When Φ stands for u or v these are the Navier-Stokes equations, and the source terms for an axisymmetric flow with swirl are:

$$S_u = -\frac{\partial p}{\partial x} + S^u \qquad (2)$$

$$S_v = -\frac{\partial p}{\partial r} + \frac{\rho w^2}{r} - \frac{2\mu_t v}{r^2} + S^v \qquad (3)$$

where w is the swirl velocity, p is the static pressure and μ_t is the turbulent viscosity. Terms S^u and S^v are the ones identified as missing and they are written as:

$$S^u = \frac{\partial}{\partial x}(\mu_t\frac{\partial u}{\partial x}) + \frac{1}{r}\frac{\partial}{\partial r}(r\mu_t\frac{\partial v}{\partial x}) \qquad (4)$$

$$S^v = \frac{\partial}{\partial x}(\mu_t\frac{\partial u}{\partial r}) + \frac{1}{r}\frac{\partial}{\partial r}(r\mu_t\frac{\partial v}{\partial r}) \qquad (5)$$

These terms were coded into the program structure; this resulted in a marked improvement in convergence.

It should be noted that the results reported herein for the previous cases of Sections 3 and 4.2 include this correction to the code.

4.3.3 Simulation results

Axial velocity contours and turbulent kinetic energy contours from the numerical solution are shown in Figures 14 and 16, while experimental results, adapted from [5], are shown in Figures 15 and 17. A velocity vector plot from the numerical solution is shown in Figure 18.

4.3.4 Discussion

It is clear from Figures 14 to 17 that there are substantial differences between the numerical (non-reactive) and experimental (reactive) results, in particular in relation to the recirculation zones. The numerical results show one such zone behind the step which, although not reported in the experimental data, may safely be assumed to exist. Numerical and experimental results (Figures 14 & 15) agree that, with swirl, the toroidal recirculation zone near the fuel inlet moves away from the centre of the pipe and becomes weaker, although there is some disagreement on velocity values. The minimum axial velocity as predicted by the numerical solution (Figure 14), of -2 m/s also agrees with the experimental result (Figure 15). However, the numerical simulation predicts another recirculation zone at approximately $x/D = 1.0$ which was not observed in the experiment.

As noted in Section 4.2.2 the experimental result of axial rms velocity (Figure 17) is approximately the square root of the turbulent kinetic energy. Comparing the turbulent kinetic energy contours from the numerical solution (Figure 16), it can be seen that near the inlet (0-0.25 diameters), there is some agreement between the results. However, there is little agreement further down the pipe.

4.4 Reacting Jets with Swirl

4.4.1 Test case

In view of the reasonable comparisons between the numerical (non-reactive) and experimental (reactive) data without swirl it was desired to determine whether the differences noted in the presence of swirl were attributable to the effects of combustion on the flow field or to defective performance of the swirl model within the code. The first was presumed on the basis that expansion and acceleration due to the flame near the centreline would not permit the appearance of the extra recirculation zone as computed. To prove this point the full case of combustion with swirl was computed. Case conditions were the same as before except for the following changes:

- Methane inlet centre jet, temperature 336K
- Reactive flow, complete combustion of methane in air with 10% excess air.

It was also necessary to modify the code for this case as the combustor was water cooled, i.e., non-adiabatic. An *ad-hoc* sink term was therefore added to the energy equation near the wall, and adjusted to reproduce the experimental temperature measurements.

4.4.2 Simulation results

Full details of the simulation of the experimental case of Reference [5] will be reported elsewhere. For the present purposes, Figure 19 shows the contours of axial velocity, which can be compared with Figures 14 and 15.

4.4.3 Comparison with experiment and discussion

Numerical simulation results with reacting flow are clearly more in agreement with the reported experimental data. The third recirculation zone apparent in the non-reacting case (Figure 14) has disappeared, and the general pattern up to about one diameter is much more similar to the experimental contours of Figure 15. The lower acceleration near the centreline predicted by the code can not be explained at this stage, although it is thought to be linked to the flame chemistry (e.g., incomplete combustion in the rich zone).

The code performance in simulating this case is considered satisfactory.

5. CONCLUSIONS AND RECOMMENDATIONS

The computer code UIFLOW-2D has been successfully installed in an ARL computer system. Minor coding errors and differences due to machine dependency have been corrected, as well as some more substantial shortcomings, in particular the omission of some source terms.

Cold and reacting flows in ducts were simulated and the results compared with published experimental data. Comparisons show that the UIFLOW-2D program operates satisfactorily in all the areas tested.

The present work can not be considered to fully validate the program; further evaluation of the functioning and performance of the program needs to be conducted against appropriate experimental results.

6. REFERENCES

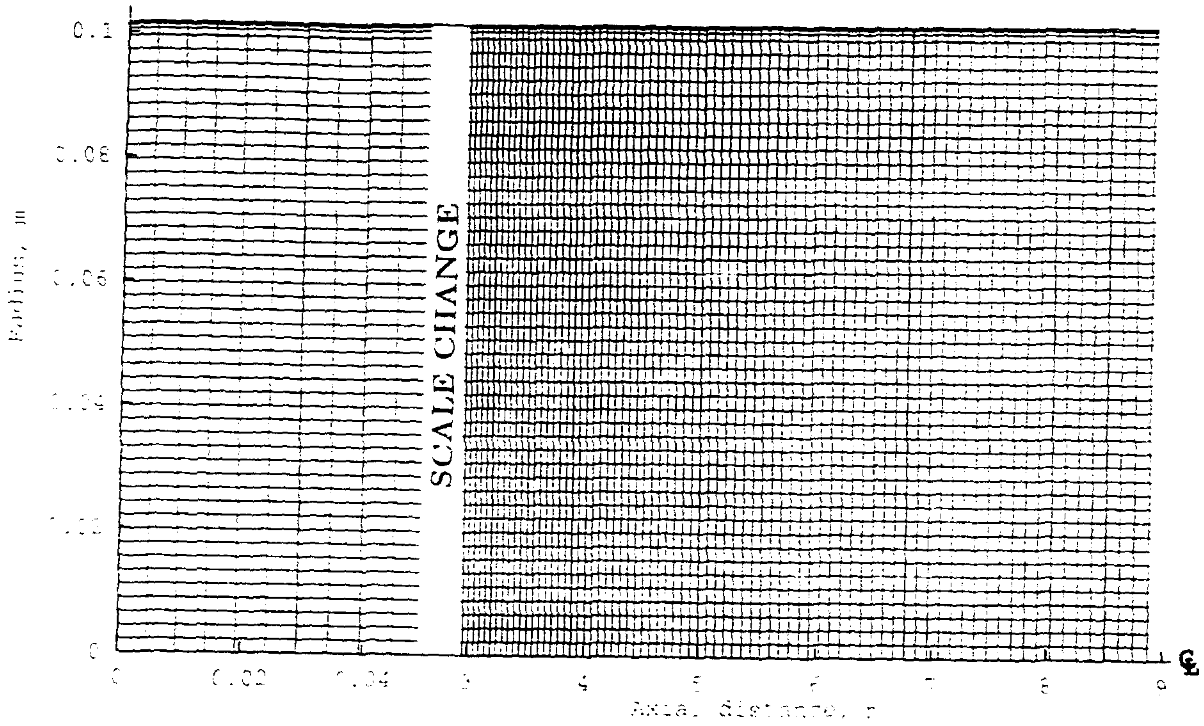
1 COPE, W. K., "UIFLOW-2D - A Computer Program for the Numerical Simulation of Turbulent Flows in Complex Geometries", M.S. Thesis, Department of Mechanical and Industrial Engineering, University of Illinois, Urbana-Champaign, 1989.

2 FAVRE, A.; "Statistical equations of turbulent gases"; in *Problems of Hydrodynamics and Continuum Mechanics*, Society of Industrial and Applied Mathematics, Pages 231-266, Philadelphia, 1969.

3 BILGER, R. W.; "Turbulent Flows with Non-premixed Reactants"; in *Turbulent Reacting Flows*, P. A. Libby and F. A. Williams, editors, Pages 65-113, Springer-Verlag, Berlin, 1980.

4 BARBIN, A. R. and JONES, J. B.; "Turbulent Flow in the Inlet Region of a Smooth Pipe", in *Journal of Basic Engineering,*, ASME, Pages 29-34, March 1963.

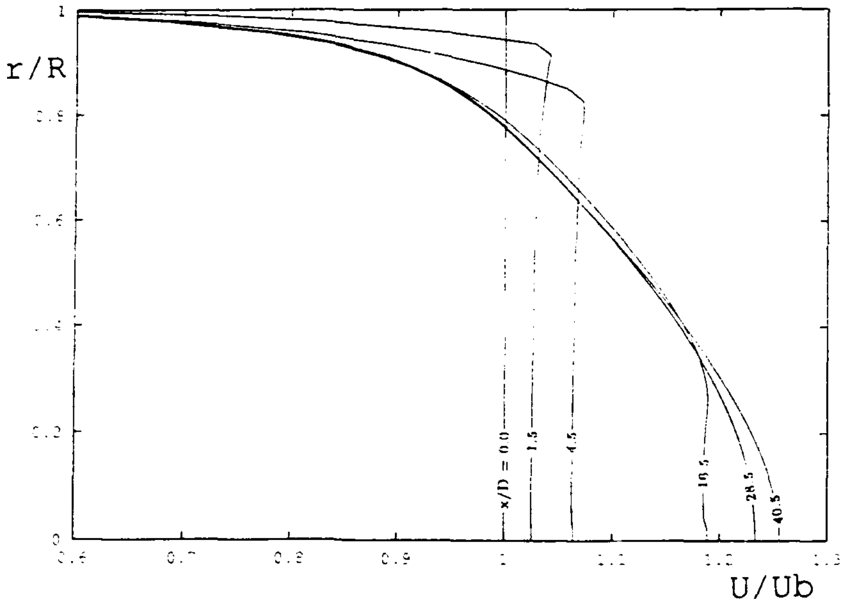
5 SPADACCINI. L. J.; OWEN, F. K., and BOWMAN, C. T.; "Influence on Aerodynamic Phenomena of Pollutant Formation in Combustion (Phase I. Gaseous Fuels)"; Report No. EPA-600/2-76-247a, United Technology Research Centre, Connecticut, September 1976.



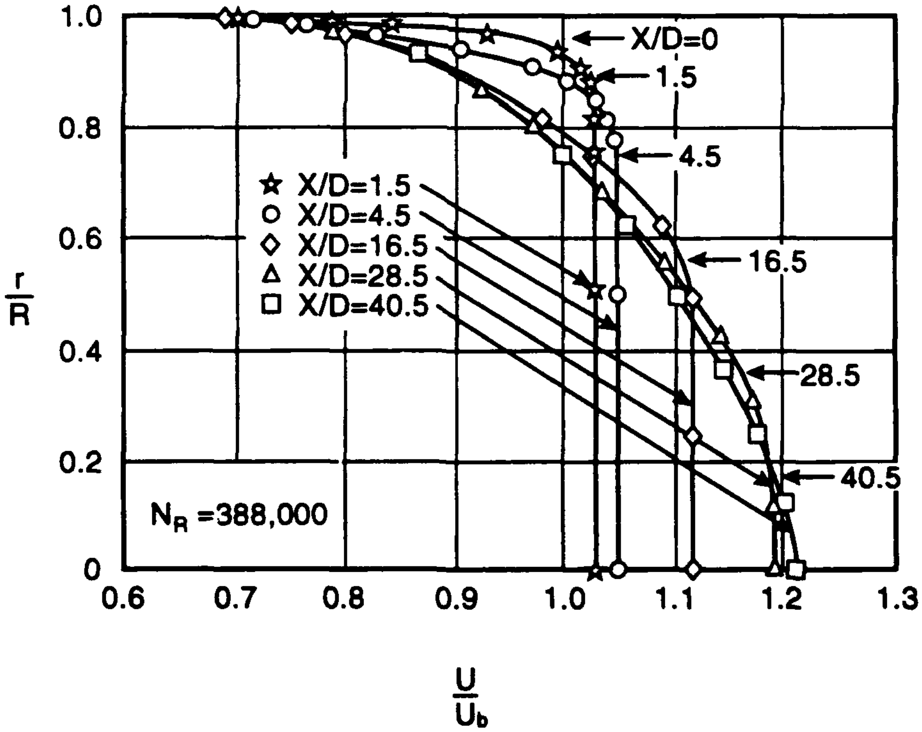
Grid used for the first case

Figure 1

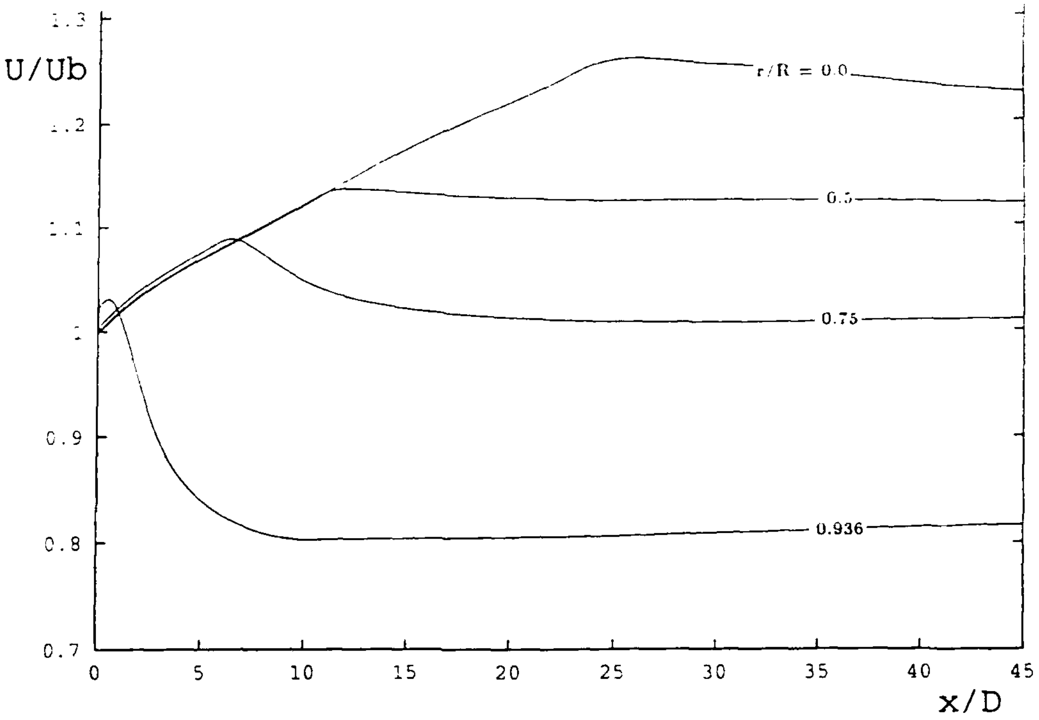
COPY AVAILABLE TO DTIC DOES NOT PERMIT FULLY LEGIBLE REPRODUCTION



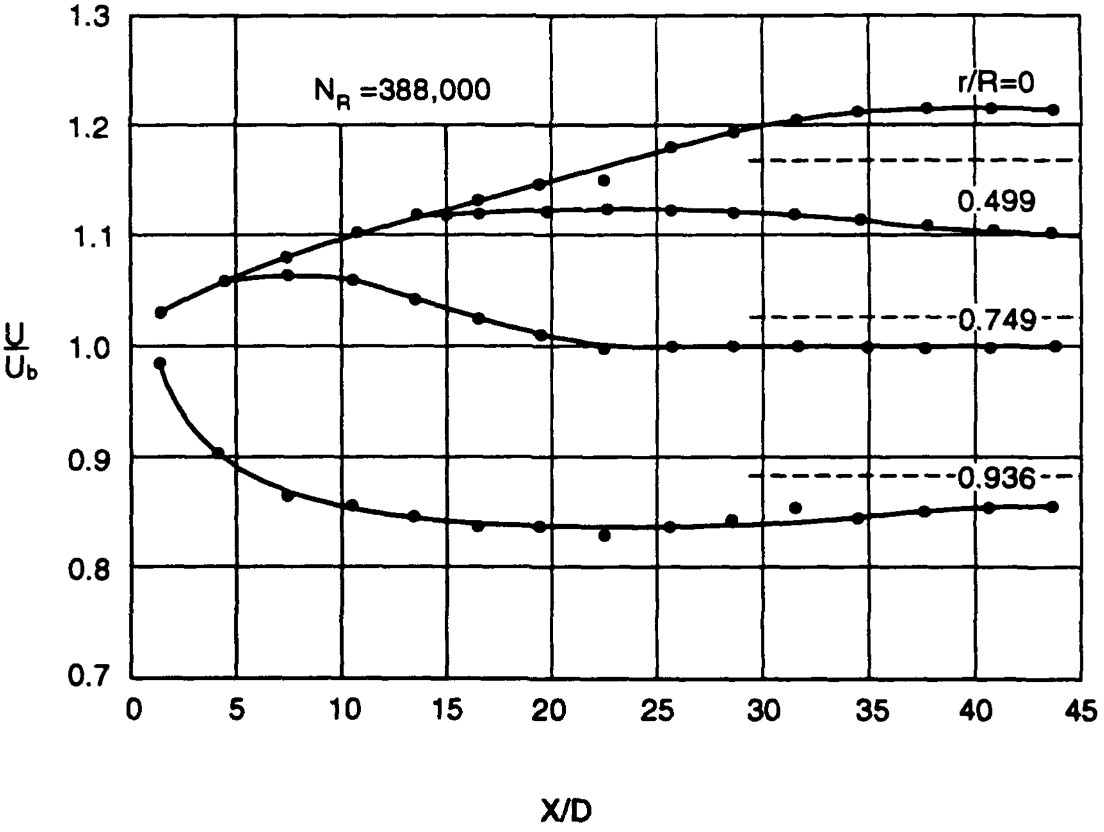
Velocity profiles in inlet region
Numerical results
Figure 2



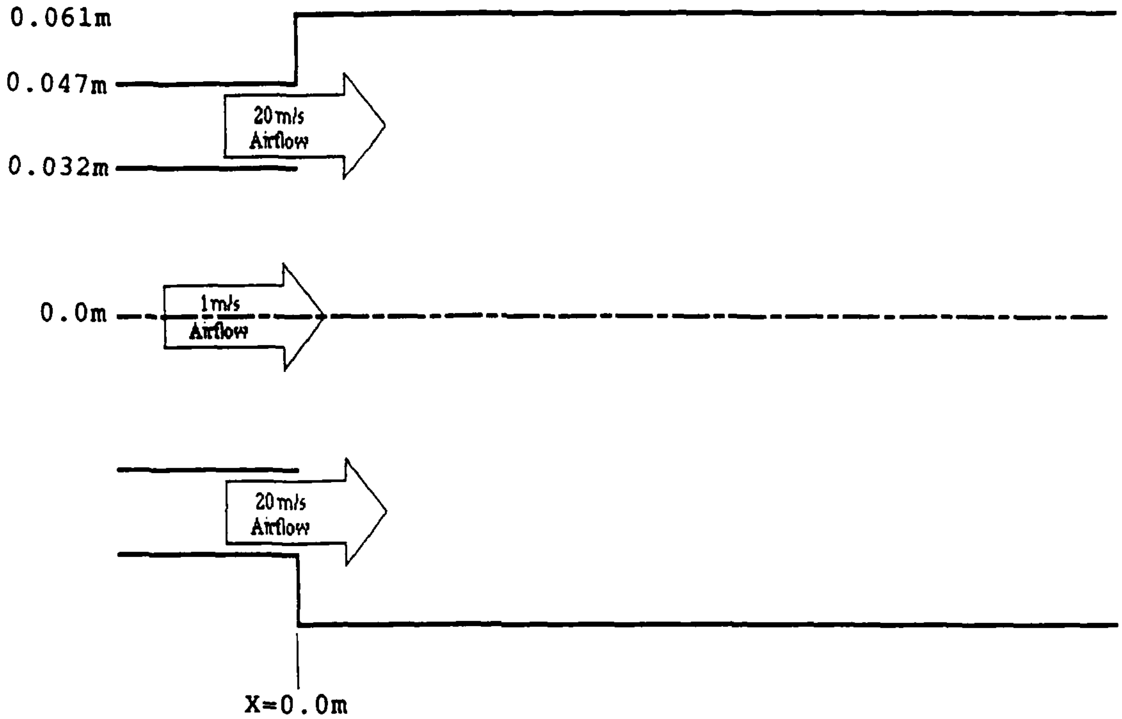
Velocity profiles in inlet region
Experimental results
Figure 3



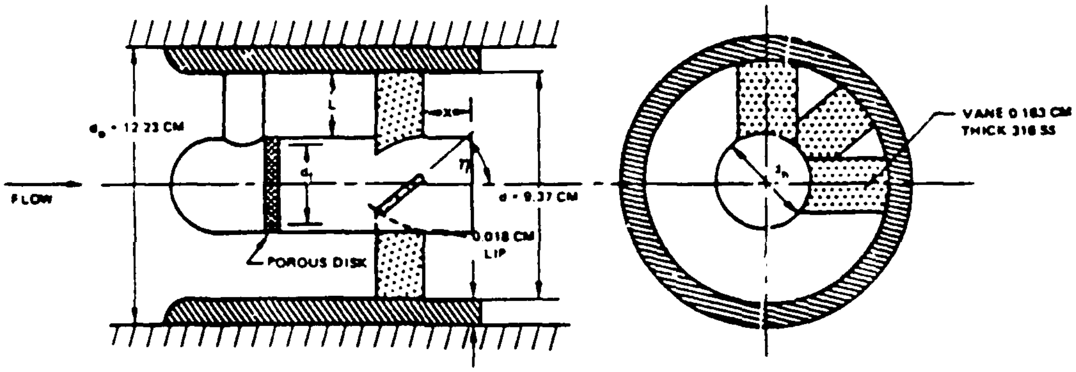
Variation of velocity along pipe
Numerical results
Figure 4



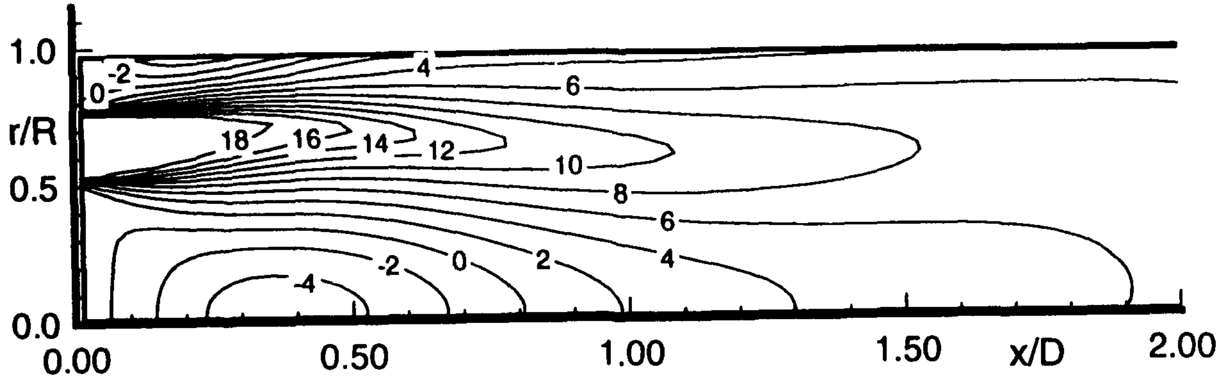
Variation of velocity along pipe
Experimental results
Figure 5



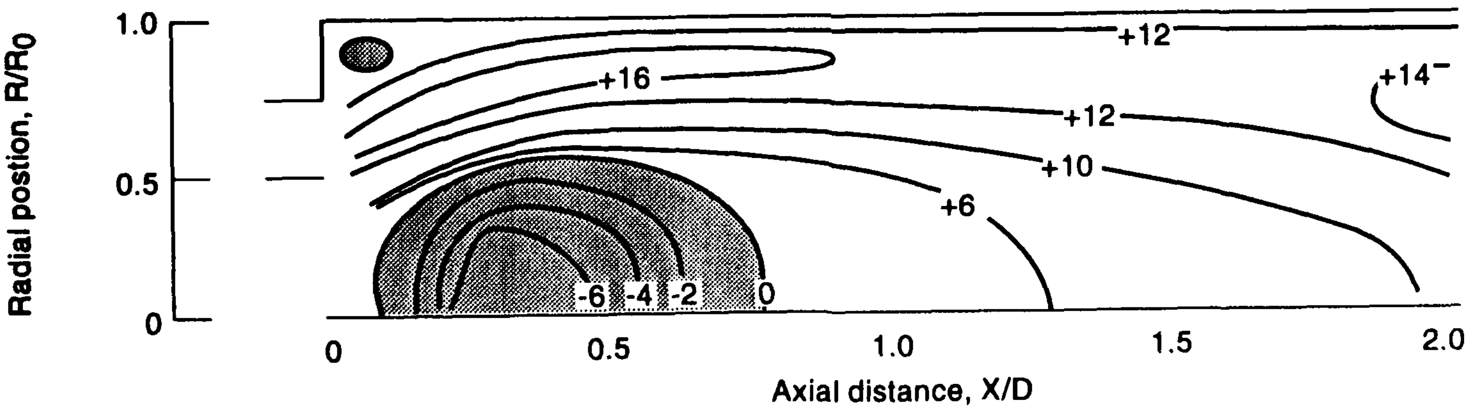
Simplified injector geometry used in simulation
Figure 6



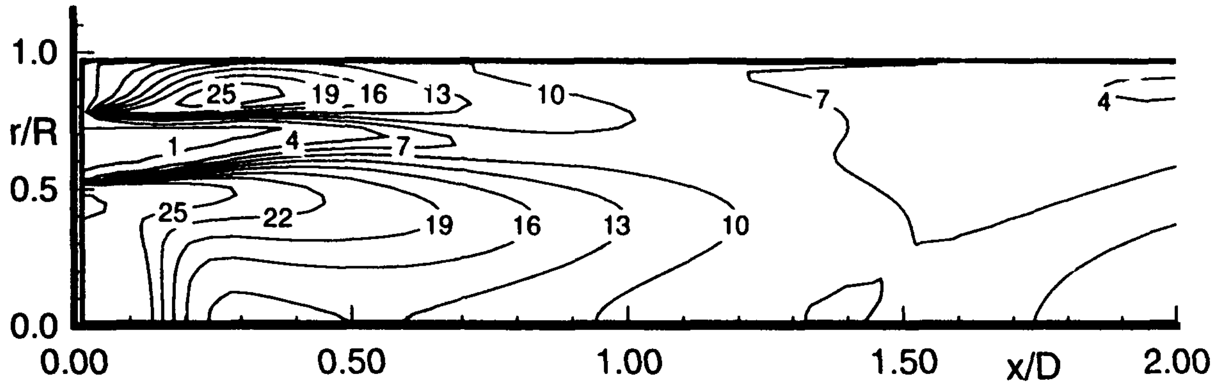
Injector apparatus used in experiment
Figure 7



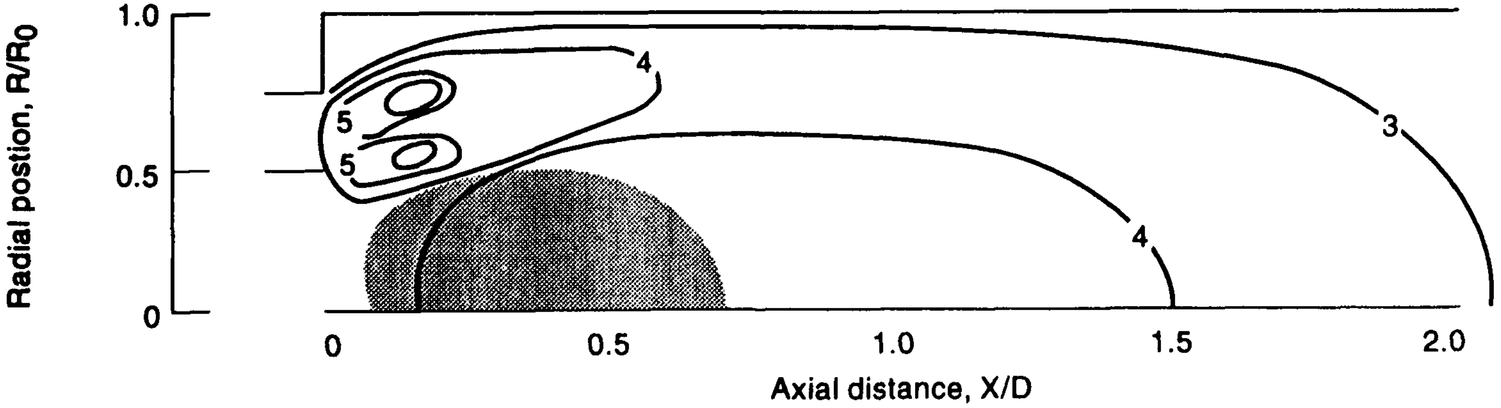
Axial velocity contours (no swirl)
Numerical results
Figure 8



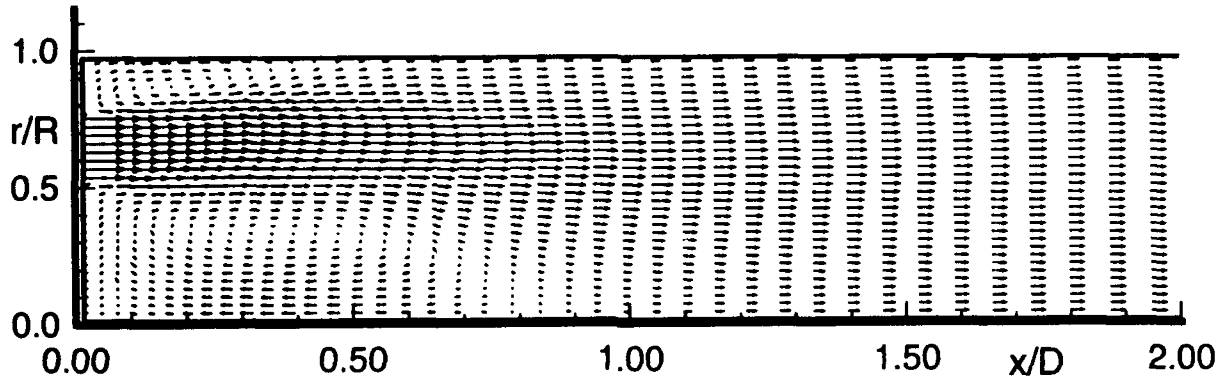
Axial velocity contours (no swirl)
Experimental results
Figure 9



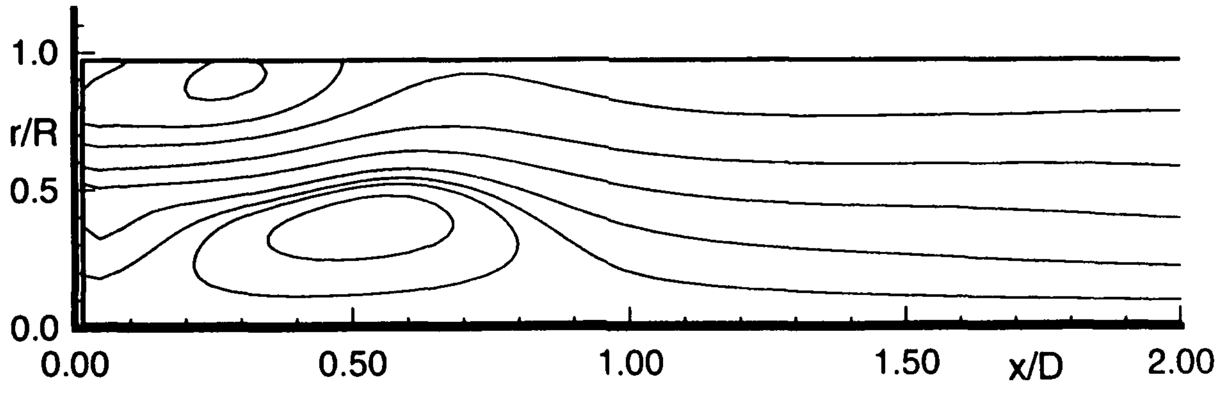
Turbulent kinetic energy contours (no swirl)
Numerical results
Figure 10



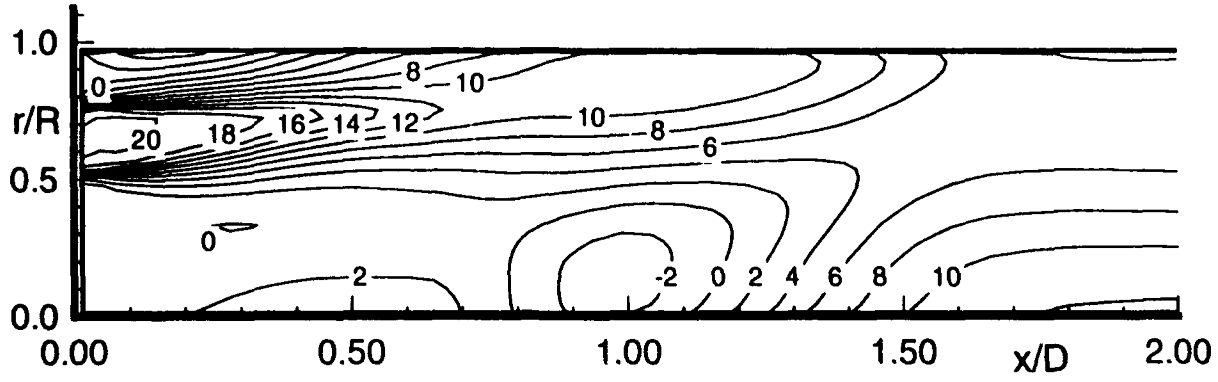
Axial RMS velocity contours (no swirl)
Experimental results
Figure 11



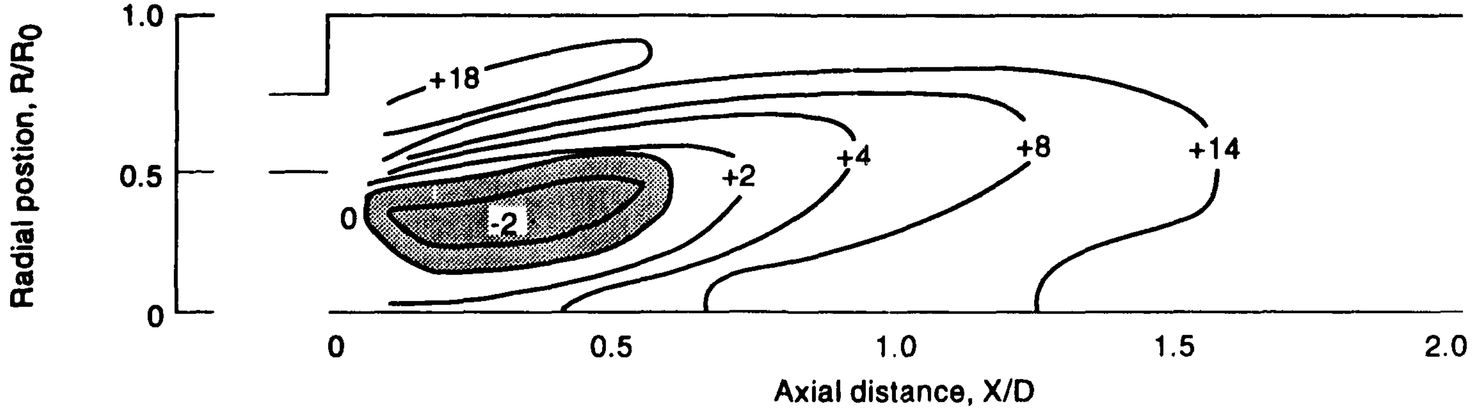
Velocity vectors (no swirl)
Numerical results
Figure 12



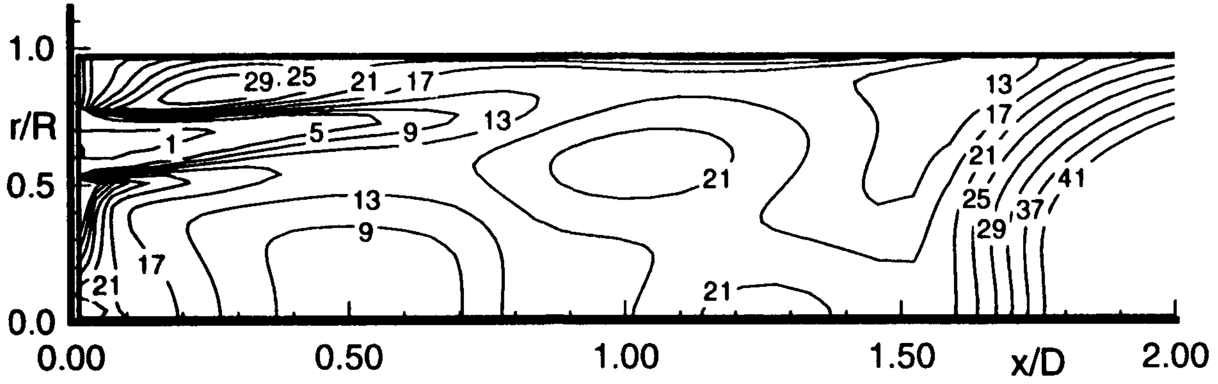
Streamlines (no swirl)
Numerical results
Figure 13



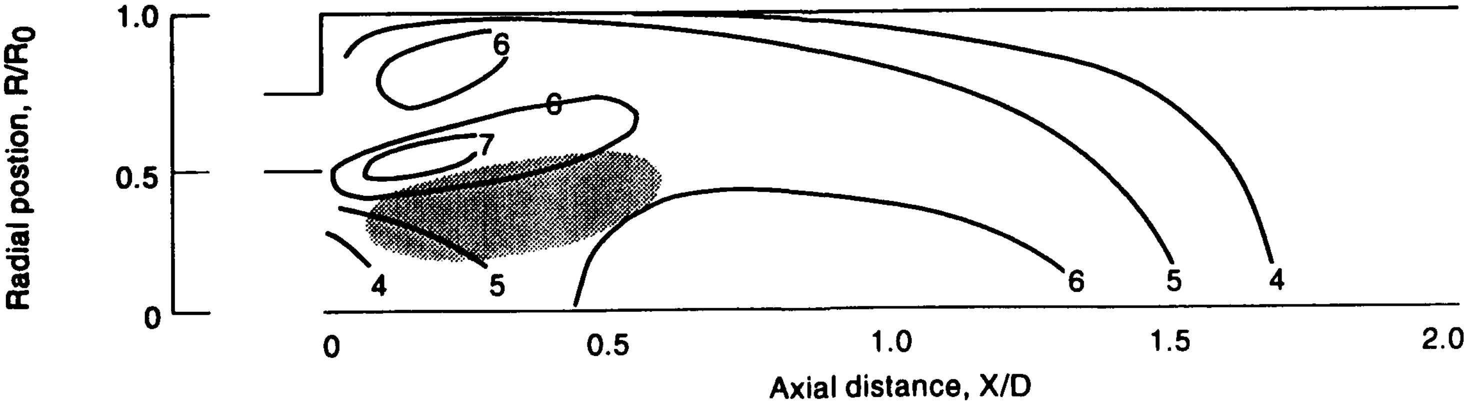
Axial velocity contours (with swirl)
Numerical results
Figure 14



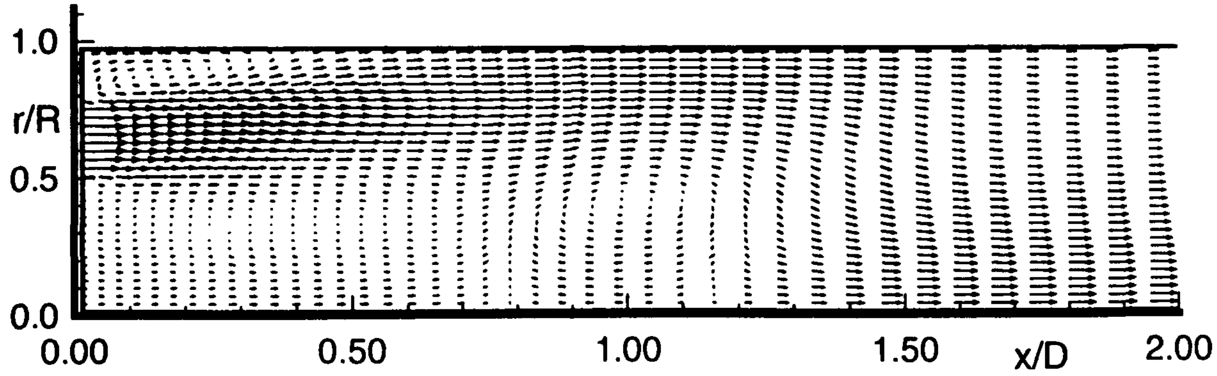
Axial velocity contours (with swirl)
Experimental results
Figure 15



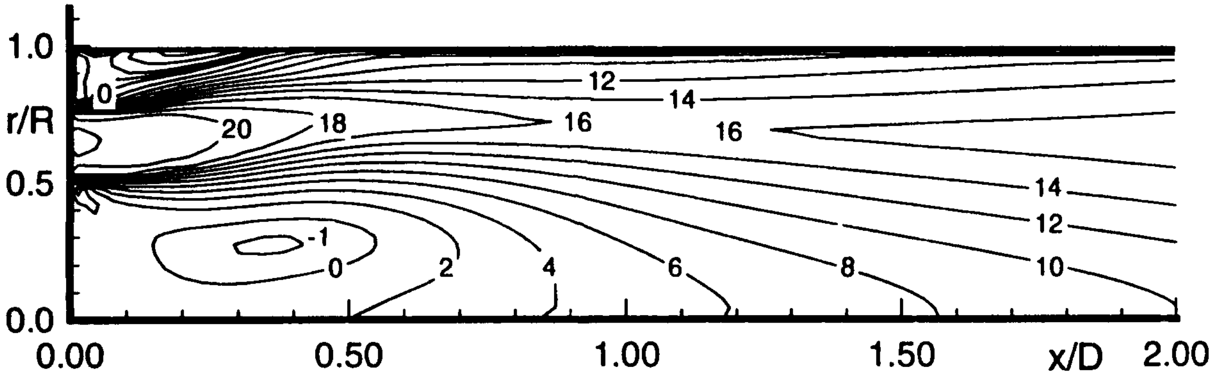
Turbulent kinetic energy contours (with swirl)
Numerical results
Figure 16



Axial RMS velocity contours (with swirl)
Experimental results
Figure 17



Velocity vectors (with swirl)
Numerical results
Figure 18



Axial velocity contours (with swirl)
Methane flame - Numerical results
Figure 19

DISTRIBUTION

AUSTRALIA

Department of Defence

Defence Central

- Chief Defence Scientist
- AS, Science Corporate Management } shared copy
- FAS Science Policy
- Counsellor, Defence Science, London (Doc Data sheet only)
- Counsellor, Defence Science, Washington (Doc Data sheet only)
- Scientific Adviser, Defence Central
- OIC TRS, Defence Central Library
- Document Exchange Centre, DSTIC (8 copies)
- Defence Intelligence Organisation
- Librarian, Defence Signals Directorate, (Doc Data sheet only)

Aeronautical Research Laboratory

- Director
- Library
- Chief Airframes and Engines Division
- Author: E. Brizuela

Defence Science & Technology Organisation - Salisbury

- Library

Navy Office

- Navy Scientific Adviser (3 copies Doc Data sheet only)
- Naval Support Command
  - Superintendent, Naval Aircraft Logistics
- Directorate of Aviation Projects - Navy

Army Office

- Scientific Adviser - Army (Doc Data sheet only)
- Engineering Development Establishment Library

Air Force Office

- Air Force Scientific Adviser (Doc Data sheet only)
- Aircraft Research and Development Unit
  - Library
- PDR AF
- DENGPP-AF
- DGELS AIRREG4 HQLC
- OIC ATF, ATS, RAAFSTT, WAGGA (2 copies)

Other Organisations
- NASA (Canberra)
- AGPS

Universities and Colleges
- Sydney
  - Department of Mechanical Engineering
    - Author: D. Davis

SPARES (4 COPIES)

TOTAL (33 COPIES)

AL 149 DEPARTMENT OF DEFENCE

DOCUMENT CONTROL DATA

PAGE CLASSIFICATION
UNCLASSIFIED

PRIVACY MARKING

1a. AR NUMBER AR-008-338	1b. ESTABLISHMENT NUMBER ARL-TN-38	2. DOCUMENT DATE JUNE 1993	3. TASK NUMBER DST 92/128
4. TITLE APPLICATIONS OF THE PROGRAM UIFLOW-2D TO THE NUMERICAL SIMULATION OF AXISYMMETRIC FLOWS		5. SECURITY CLASSIFICATION (PLACE APPROPRIATE CLASSIFICATION IN BOX(S) IE. SECRET (S), CONF. (C) RESTRICTED (R), LIMITED (L), UNCLASSIFIED (U)). U DOCUMENT U TITLE U ABSTRACT	6. NO. PAGES 22 7. NO. REFS. 5

8. AUTHOR(S)
D. DAVIS
E. BRIZUELA

9. DOWNGRADING/DELIMITING INSTRUCTIONS

Not applicable

10. CORPORATE AUTHOR AND ADDRESS

AERONAUTICAL RESEARCH LABORATORY

AIRFRAMES AND ENGINES DIVISION

506 LORIMER STREET

FISHERMENS BEND VIC 3207

11. OFFICE/POSITION RESPONSIBLE FOR:

SPONSOR DSTO

SECURITY -

DOWNGRADING -

APPROVAL CAED

12. SECONDARY DISTRIBUTION (OF THIS DOCUMENT)

Approved for public release.

OVERSEAS ENQUIRIES OUTSIDE STATED LIMITATIONS SHOULD BE REFERRED THROUGH DSTIC, ADMINISTRATIVE SERVICES BRANCH, DEPARTMENT OF DEFENCE, ANZAC PARK WEST OFFICES, ACT 2601

13a. THIS DOCUMENT MAY BE ANNOUNCED IN CATALOGUES AND AWARENESS SERVICES AVAILABLE TO ....

No limitations.

13b. CITATION FOR OTHER PURPOSES (IE. CASUAL ANNOUNCEMENT) MAY BE ☒ UNRESTRICTED OR ☐ AS FOR 13a.

14. DESCRIPTORS
Turbulent flow
Axisymmetric flow
Numerical simulation
Uiflow-2D (computer program)
Combustion chambers

15. DISCAT SUBJECT CATEGORIES
2004
2102

16. ABSTRACT

The computer program UIFLOW-2D has been applied to the numerical simulation of turbulent axisymmetric flows.

In a preliminary test of the performance of the program it has been used to simulate four different types of flows, isothermal and reactive, with and without swirl. Numerical results are discussed and compared with published experimental data. Recommendations are made for future development and application to gas turbine combustors.

PAGE CLASSIFICATION
UNCLASSIFIED

PRIVACY MARKING

THIS PAGE IS TO BE USED TO RECORD INFORMATION WHICH IS REQUIRED BY THE ESTABLISHMENT FOR ITS OWN USE BUT WHICH WILL NOT BE ADDED TO THE DISTIS DATA UNLESS SPECIFICALLY REQUESTED.

16. ABSTRACT (CONT).		
17. IMPRINT **AERONAUTICAL RESEARCH LABORATORY, MELBOURNE**		
18. DOCUMENT SERIES AND NUMBER Technical Note 38	19. WA NUMBER 42 434F	20. TYPE OF REPORT AND PERIOD COVERED
21. COMPUTER PROGRAMS USED		
22. ESTABLISHMENT FILE REF.(S)		
23. ADDITIONAL INFORMATION (AS REQUIRED)		