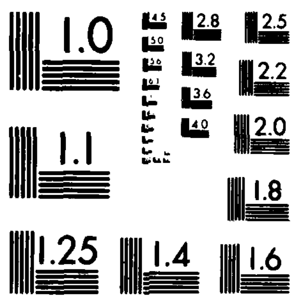
AD-A110 501 CALIFORNIA UNIV BERKELEY DEPT OF MECHANICAL ENGINEERING F/G 20/5
MEASUREMENT AND CALCULATION OF DEVELOPING TURBULENT FLOW IN A U--ETC(U)
DEC 81 J A HUMPHREY, S M CHANG N00014-80-C-0031

UNCLASSIFIED NL

1 of 1

END
DATE
FILMED
3 82
DTIC



MICROCOPY RESOLUTION TEST CHART
NATIONAL BUREAU OF STANDARDS-1963-A

MEASUREMENT AND CALCULATION OF DEVELOPING TURBULENT FLOW IN A U-BEND AND DOWNSTREAM TANGENT OF SQUARE CROSS-SECTION

Second Annual Technical Report
corresponding to period
October 1, 1980 - September 30, 1981
for project entitled

TURBULENT FLOW AND HEAT TRANSFER IN PASSAGE AROUND 180° BEND - AN EXPERIMENTAL AND NUMERICAL STUDY

by

Joseph A.C. Humphrey - Principal Investigator
Sheng M. Chang - Graduate Research Assistant
Department of Mechanical Engineering
University of California
Berkeley, CA. 94720
U.S.A.

in collaboration with

Brian E. Launder
Department of Mechanical Engineering
University of Manchester Institute of Science and Technology
Manchester, England

Prepared for
M.K. Ellingsworth, Program Monitor
The Office of Naval Research
Arlington, Virginia 22217

Under Contract No. N00014-80-C-0031, work unit NR 097-440

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

REPORT DOCUMENTATION PAGE		READ INSTRUCTIONS BEFORE COMPLETING FORM
1. REPORT NUMBER N00014-80-C-0031A	2. GOVT ACCESSION NO. AD-A110 501	3. RECIPIENT'S CATALOG NUMBER
4. TITLE (and Subtitle) Measurement and Calculation of Developing Turbulent Flow in a U-Bend and Downstream Tangent of Square Cross-Section		5. TYPE OF REPORT & PERIOD COVERED Yearly (Summary) Oct. 1, 1980-Sept. 30, 1981
		6. PERFORMING ORG. REPORT NUMBER
7. AUTHOR(s) Joseph A.C. Humphrey		8. CONTRACT OR GRANT NUMBER(s) N00014-80-C-0031
9. PERFORMING ORGANIZATION NAME AND ADDRESS Department of Mechanical Engineering University of California Berkeley, CA. 94720		10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS
11. CONTROLLING OFFICE NAME AND ADDRESS Office of Naval Research 800 N. Quincy Street Arlington, VA. 22217		12. REPORT DATE December 1981
		13. NUMBER OF PAGES 36
14. MONITORING AGENCY NAME & ADDRESS (if different from Controlling Office)		15. SECURITY CLASS. (of this report) Unclassified
		15a. DECLASSIFICATION DOWNGRADING SCHEDULE

16. DISTRIBUTION STATEMENT (of this Report)

Approved for Public Release; Distribution Unlimited

17. DISTRIBUTION STATEMENT (of the abstract entered in Block 20, if different from Report)

Same as Block No. 16

18. SUPPLEMENTARY NOTES

19. KEY WORDS (Continue on reverse side if necessary and identify by block number)

Turbulence Modeling, Curved Duct Flow, Laser Doppler Velocimetry, Numerical Computation, Finite Differencing, Heat Transfer, Algebraic Stress Modeling, Numerical Diffusion.

20. ABSTRACT (Continue on reverse side if necessary and identify by block number)

A review is provided of the second year of research pertaining to the experimental measurement and numerical prediction of curved duct flows. Experimental measurements obtained using a laser-Doppler velocimeter, recent turbulence model developments and numerical calculations are presented and discussed. The current state of the research program is revised together with a work plan for year three of research.

400436

DD FORM 1473 EDITION OF 1 NOV 65 IS OBSOLETE

SECURITY CLASSIFICATION OF THIS PAGE

Table of Contents

Foreword and Acknowledgements

1. The Problem of Interest: Motivation, Scope and Objectives

2. Summary of Research Accomplishments for Year Two

   2.1 Contribution to the Stanford Conference on Complex Turbulent Flows

   2.2 The Formulation of a More General k-ε Model of Turbulence

   2.3 Measurements and Calculations of Three-Dimensional Curved Duct Flow

3. Conclusions and Research Plans for Year Three

4. References

Appendix 1. Prediction of Case 512 for the 1981-1982 AFOSR-HTTM-Stanford Conference on Complex Turbulent Flows

FOREWORD AND ACKNOWLEDGEMENTS

This report serves two purposes. It is the second annual report to the Office of Naval Research (ONR) for Contract No. N00014-80-C-0031. It is also a final report to the National Science Foundation (NSF), pertaining to Grant No. ENG-7827007.

In several important ways the interests of these two research programs have overlapped strongly, to their mutual benefit. The overlap was entirely complementary, duplication of research efforts being strictly barred. However, it was deemed preferable by the author to await the outcome of the second year of the ONR research program prior to submitting a final report to NSF which should also serve as a second yearly report to ONR. The author wishes to express his most sincere gratitude to Messieurs Win Aung and Keith Ellingsworth, of the NSF and ONR respectively, for permitting and encouraging this research effort collaboration.

While the work presented here has been conducted entirely on the Berkeley campus of the University of California, it has been carried out in close collaboration with a 'sister' project under the direction of Professor B.E. Launder in the Mechanical Engineering Department of the University of Manchester, Institute of Science and Technology (UMIST). The experimental investigation in UMIST represents the heat transfer counterpart of the fluid mechanics activity in Berkeley. The collaborative effort is not only experimental, it also extends to the improved formulation of a theoretical model of turbulence applicable to curved duct flows.

During the course of the second year of research the following people have assisted in advancing experimental and theoretical aspects of the research project: M. Arnal, S.M. Chang, J. Flores, T. Han, G. Lewis, A. Modavi, J. Sabnis and P. Turi. The author gratefully acknowledges their very helpful support.

1. THE PROBLEM OF INTEREST: MOTIVATION, SCOPE AND OBJECTIVES

This section is intended to provide a brief reminder of the motivation, scope and objectives of the present study.

Flow in curved ducts represents a phenomenon of considerable industrial importance. In particular, it is considered here in connection with bend components in heat exchange equipment. The study is strongly motivated by the need to quantify the fluid mechanic and heat transfer characteristics of such flows in order to reach conclusions leading to the improved performance and more compact design of heat exchange equipment. That this need is very real, in both industrial and military environments, was quite clearly established at the July 1981 Navy Symposium on Heat Transfer Research, held at the U.S. Naval Academy, Annapolis, Maryland.

The scope of this study is not limited by the applied nature of the motivation. In fact, the study addresses the very fundamental issues of turbulent flow and heat transfer in complex, three-dimensional (3D) configurations. Thus, the scope of the present investigation embraces the need for improved theoretical formulations and accurate experimental data, useful for prediction and testing purposes respectively.

However, the role of the experimental data is not limited to that of a test matrix alone. Through careful analysis and evaluation the data also offers the opportunity for an increased understanding of the physics governing turbulent fluid mechanics. As discussed in the first yearly report, curved duct flow configurations are ideal for investigating the transition between two important types of cross-stream secondary motion. In a curved duct (bend) component, the imbalance between centrifugal and radial pressure gradient forces sets up a fairly intense cross-stream flow. As the flow leaves the bend to enter a downstream straight duct section the force imbalance disappears. At this point turbulence diffusion and redistribution processes force the flow to undergo a 'relaxation' stage which acts to erase all memory of the force imbalance acting on the flow in the bend. However, a very weak cross-stream secondary motion persists in the downstream straight duct section due to differences in the cross-stream gradients of the Reynolds stresses. It is clear that the flow configuration composed of a curved duct followed by a straight duct section is not only one of industrial relevance, but, due to its complexity, represents a rather severe test for models used to predict general turbulent flows.

The objectives of this investigation are two:

1) To develop, test and apply a model of turbulence embodied in a numerical calculation procedure capable of predicting complex three-dimensional flows with elliptic effects retained in the pressure field only.
2) To obtain experimental data of value for testing the numerical model and for advancing the understanding of turbulent flow in general.

The strategy for achieving the above objectives has already been outlined in the study proposal and first yearly report to ONR. The next section summarizes specific accomplishments related to the objectives, attained during year two of research.

2. SUMMARY OF RESEARCH ACCOMPLISHMENTS FOR YEAR TWO

Three major accomplishments are claimed under this heading and are summarized briefly here. More detailed expositions of the accomplishments are provided in the references given in this report.

2.1 Contribution to the Stanford Conference on Complex Turbulent Flows

In response to a "call for predictions" by the AFOSR-HTTM-Stanford Conference on Complex Turbulent Flows, calculations were made (Appendix 1) of the curved duct flow configuration measured by Humphrey [1]. This flow corresponds to Case 512 of the Stanford Conference and was also computed by various other groups using similar modeling approaches. A two-equation k-ε model of turbulence was used for the predictions plotted in Appendix 1. A semi-elliptic calculation procedure was the basis for the computational algorithm. Two finite-difference schemes for convection terms in the momentum equations were tested. These have been relatively evaluated and discussed by Han, Humphrey and Launder [2]. Calculations performed on four laminar and two turbulent flow test-case configurations yielded accurate results, indicating the validity of the numerical procedure. However, calculations of Case 512 (even when using the more accurate of the two differencing schemes) yielded poor agreement with the measurements as of a bend angle of 45°. Nevertheless, the results were better than the (coarser grid) calculations performed by Humphrey, Whitelaw and Yee [3] using a fully-elliptic calculation procedure, and, without exception, were consistently better than any of the predictions offered of Case 512 by other groups at the Stanford Conference.

While it is doubtful that a two-equation k-ε model will provide the accurately needed representation of the turbulence characteristics arising in three-dimensional curved duct flow, numerical diffusion in the calculations still obscures a precise judgement of the issue. The numerical and modeling aspects of the problem warrant continued and careful research. Both of these aspects are being pursued in Berkeley.

2.2 The Formulation of a More General k-ε Model of Turbulence

In the study presented in reference [4], Pourahmadi and Humphrey address the problem of including streamline curvature and pressure strain effects in the C_μ coefficient of $\nu_t = C_\mu k^2/\varepsilon$. The study shows that improved predictions of developing curved channel flow mean velocity, turbulent kinetic energy and friction coefficient, are given with the generalized C_μ formulation. In particular, the inclusion of wall-dampening contributions to the pressure strain term is shown to be significant. This suggests that calculations of other flow such as, for example, backward facing steps or sudden expansions, where streamline curvature and pressure-strain contributions to C_μ are important, should include the latter contribution to C_μ. It is important to note that generalization of the C_μ function along the lines of reference [4] avoids the need of ad hoc modifications (and associated constant optimization) in the k-ε model, in order to include streamline curvature and pressure strain contributions to turbulent diffusion. It is conceivable, but it would have to be checked, that the inclusion of a more general 3D formulation of the C_μ coefficient in the k-ε model of Appendix 1 would yield improved predictions of curved duct flow. However, the level of effort involved in executing this task is comparable to that of deriving a set of algebraic relations for the Reynolds stresses directly.

Given that the availability of algebraic stress relations would preclude the need for the concept of an eddy viscosity, and hence C_μ, and given also that a model approach based on the use of algebraic stresses can address, at least to first order, the issue of anisotropy in turbulent flow, the algebraic stress approach is to be preferred. The work in Berkeley has centered on formulating an algebraic stress approach in collaboration with the group headed by Professor Launder in UMIST.

2.3 Measurement and Calculation of Three-Dimensional Curved Duct Flow

Reference [5] provides an exposition of the curved duct flow experimental work and the most recent numerical calculations, using a k-ε model of turbulence, conducted in Berkeley during year two of this contract.
The measurements were made using a laser-Doppler velocimeter (LDV) of DISA manufacture, and consisted of two velocity components and the associated Reynolds stresses. At each of nine streamwise locations several profiles were taken of the variables of interest. The data was recorded and processed on-line by means of a PDP 1134 minicomputer. A description of tre apparatus and instrumentation, plots of profiles with an interpretation of measured and calculated results, and a discussion of possible error sources affecting the precision of the measurements are provided in [5].

As a general observation, it can be stated here that the flow in a 180° bend of square cross-section differs markedly from that in a 90° bend in its mean and fluctuating characteristics. The efforts put into measuring a curved duct configuration of square cross-section have been amply rewarded in terms of the very useful and (relatively) easily obtained experimental results. The ease with which measurements were made was enhanced by the presence of flat walls in the test section. Corresponding measurements of comparable precision would be considerably more difficult to obtain in curved pipes. A major conclusion derived from the experimental study is the need for additional data in the straight duct section downstream from the bend where the turbulent flow relaxes. Similarly, further information between bend angles of 45° and 180°, where the flow changes dramatically in its characteristics, would benefit theoretical and turbulence modeling advancements.

Turbulent (and laminar) flow calculations+ were performed for the test section configuration of the experiment and are discussed in [5]. To summarize: although the turbulent flow results show good qualitative agreement with the measurements, they display discrepancies as large as those presented for the 90° bend flow of Appendix 1; as in the Appendix, the discrepancies are attributed to failings in the ability of the k-ε model to represent faithfully the turbulent features of the flow, and to the presence of numerical diffusion in the calculations. There is a pressing need to separate and establish properly the respective contributions of these two effects on overall numerical inaccuracy.

With respect to the paragraph above, it is rather important to note that advances of theoretical significance, and therefore of consequence to turbulence modeling practice, are much more likely to progress quickly in relation to ducts of square cross-section, where accurate experimentation is relatively easy to perform, than in ducts of circular cross-section. Since the theory for predicting turbulent flows in ducts of square cross-section includes as a subset the class of flows in ducts of circular cross-section, it seems appropriate to emphasize theoretical and experimental research in the square duct flow configuration. This is especially true if, as is the case here and for the UMIST project, measurements and calculations involving heat transfer are a major consideration.

Laminar regime calculations of the 180° bend experimental configuration reveal flow patterns very distinct from those arising in turbulent regime. Unfortunately, the lack of experimental data for the moment precludes quantifying exactly the accuracy of the predictions. Grid refinement tests and

+Corresponding laminar flow measurements have not been made.

calculations of the 90° bend laminar flow in [1] suggest that predictions of the streamwise component of velocity are accurate to within about ± 10%. It is a major conclusion of this report that detailed laminar flow measurements are required for conducting a careful examination of the extent to which false diffusion can affect the accuracy of numerical calculations in 3D laminar flows with elliptic effects. Such knowledge would assist in establishing more clearly the role of false diffusion in the turbulent flow regime.

3. CONCLUSIONS AND RESEARCH PLANS FOR YEAR THREE

The following are the major conclusions derived from the activities conducted in relation to the second year of this research.

1. The July 1981 Navy Symposium on Heat Transfer Research clearly established the need for an improved theoretical formulation of three-dimensional turbulent flows with elliptic effects, due to the strong financial incentives of dealing with such flows directly through numerical computation.

2. To assist in turbulence model development, and to contribute to the pool of information required in order to advance the current level of understanding of turbulent fluid mechanics, fundamental experimental measurements have been made in a curved duct configuration of both industrial and academic significance.

3. Numerical calculations using a two equation k-ε model of turbulence of: a) Case 512 of the Stanford Conference on Complex Turbulent Flows; and, b) the present experimental configuration, reveal the need to separate inaccuracies arising from numerical diffusion from those arising due to turbulence model defficiences. Presently, numerical efforts in Berkeley are being directed towards the more effective use of higher order finite difference schemes for convection terms in the transport equations. In parallel, turbulence modeling efforts are being focussed on an algebraic stress formulation; obviating the need for an eddy viscosity concept, and dealing with flow anisotropy directly. This formulation retains streamline curvature and pressure-strain effects, including wall-induced redistribution of the energy among the normal stress components.

4. The measurements obtained to date have been made in a duct (curved section and downstream tangent) of square cross-section. The flat walls composing the square shape allow easy optical access to the velocimeter laser beams, and yield Doppler bursts of high signal to noise ratio. Similar quality data, and as extensive, is not readily measured in ducts of circular cross-section. Because the turbulence modeling concepts required for predicting curved duct flows of square cross-section encompass those necessary for predicting similar flows in ducts of circular cross-section, the experimental limitations affecting the latter configuration are not particularly worrisome.

   It should be clear that a turbulence model validated primarily with respect to measurements obtained in a duct of square cross-section, and which predicts this flow accurately, must, by necessity, model the simpler flow in a curved pipe. The use of the term "simpler" in connection with curved pipe flow implies the absence of corner effects inthe duct cross-section.

5. Although an algebraic stress model, similar to that of Sindir [6], has already been formulated in Berkeley in collaboration with UMIST, it has not yet been applied successfully to the curved duct flow configuration of interest to this study. Presently it is being tested with respect to developing flow in a straight duct of square cross-section. When this case is satisfactorily predicted the algebraic stress model will be applied to the curved duct configurations of this study and of Humphrey, Whitelaw and Yee [3].

Listed below are the tasks to be accomplished during year three of research, together with an estimate of the time required for completion.

1. A review of the data collected to date indicates a need for additional measurements in the curved duct and downstream tangent of square cross-section in order to complete the matrix of experimental results necessary for a rigorous model development and testing. This "second experimental pass" is intended to provide more detailed information of the flow characteristics entering the bend, near the curved and side walls in the bend and downstream tangents, and at several additional planes in the bend and downstream tangent. The collection and processing of this new data will take between 4-6 months+.

2. Upon completion of task 1 the test section of square cross-section will be replaced by one of circular cross-section. In the circular configuration, measurements of the flow mean velocity and turbulence characteristics will be confined to optically accessible regions of the flow with high signal to noise ratio. Due to the extensive and high quality data presently being obtained in the more complex square duct configuration, the above limitation is not serious. In combination, the body of experimental results from this study should suffice for extending the numerical model to curved pipe flows. Collection and processing of this data will taken between 6-8 months+.

3. The algebraic stress turbulence model developed for this work will be applied to the prediction of curved duct flow upon completion of its preliminary testing. Predictions will be made of the square duct

+The estimates for completion of tasks 1 and 2 include time periods during which the laser-Doppler velocimeter and minicomputer are shared with two other research groups.

flow configuration of [3] and of this study using the semi-elliptic procedure and higher order convective differencing scheme described in reference [5]. It is estimated that the accomplishment of this task will take between 6-9 months.

4. Upon completion of task 3 the validated numerical procedure will be used to compute a variety of flows in which the following parameters are varied over a range of practical interest: Reynolds number, duct aspect ratio, mean radius of curvature, downstream tangent length. This task will take approximately 3 months.

Since there is support in Berkeley for only one graduate student the above tasks are expected to run into a fourth year of research. This will certainly be the case in order to accomplish task 5 below.

5. Modifications will be made to the numerical model calculation procedure to allow for the transport of heat. This will require an extension of the theoretical formulation, and will be conducted in collaboration with Professor Launder's research group in UMIST. The time for completing this task in Berkeley will be approximately 6 months.

Table 1 presents a tentative research schedule for year three of research. Also indicated in the table are the activities which are anticipated will run into a fourth year of research.

Table 1: Tentative Research Schedule for Year Three (and Four) of Research

Research Task	Estimated Date of Completion	Comments
Completion of velocimeter measurements in duct of square cross-section.	April 1982	This data complements earlier measurements and is crucial for a complete documentation of the flow configuration.
Completion of velocimeter measurements in ducts of circular cross-section.	December 1982	It is likely that this activity will run into a fourth year of research. The data complements results obtained in a square cross-section configuration and is very desirable for model testing.
Completion and testing of algebraic stress model closure. Prediction of test flow configuration and other data.	June 1982	
Application of the numerical procedure to a range of flow conditions of practical interest.	December 1982	This activity is likely to run into a fourth year of research but will include predictions using a k-ϵ model of turbulence.
Extension of the calculation model to include heat transfer	June 1983	Although originally conceived to be part of the three year research program, these two activities will run into a fourth year of research.
Application of the extended procedure to a range of heat transfer flow configurations.	August 1983	

4. REFERENCES

1. J.A.C. Humphrey, "Flow in Ducts with Curvature and Roughness," Ph.D. Thesis, University of London, 1977.

2. T. Han, J.A.C. Humphrey and B.E. Launder, "A Comparision of Hybrid and Quadratic-Upstream Differencing in High Reynolds Number Elliptic Flows," Computer Methods in Applied Mechanics and Engineering, 29, 81, 1981.

3. J.A.C. Humphrey, J.H. Whitelaw and G. Yee, "Turbulent Flow in a Square Duct with Strong Curvature," Journal of Fluid Mechanics, Vol. 103, pp. 443-463, 1981.

4. F. Pourahmadi and Joseph A.C. Humphrey, "Prediction of Curved Channel Flow with an Extended k-ε Model of Turbulence," Mechanical Engineering Department Report No. FM-81-5, University of California, Berkeley, 1981.

5. J.A.C. Humphrey, A Modavi and S.M. Chang, "Developing Turbulent Flow in a 180° Bend and Downstream Tangent of Square Cross-Sections," Mechanical Engineering Department Report No. FM-82-1, University of California, Berkeley, 1981.

6. M.M. Sindir, "User's Guide to VAST-STEP; A general Purpose Elliptic Program for Two-Dimensional Steady Turbulent Flow Predictions," Department of Mechanical Engincering, University of California, Davis, 1980.

PREDICTION OF CASE 512 FOR THE 1981-1982

AFOSR-HTTM-STANFORD CONFERENCE ON COMPLEX TURBULENT FLOWS

by

S.M. CHANG [1]

T. HAN [2]

J.A.C.HUMPHREY [3]

in collaboration with

B.E. LAUNDER [4]

UNIVERSITY OF CALIFORNIA
DEPARTMENT OF MECHANICAL ENGINEERING
BERKELEY, CALIFORNIA 94720

REPORT NUMBER

FM-81-4

SEPTEMBER 1981

1 Research Graduate
2 Research Associate
3 Assistant Professor
4 Professor; University of Manchester
Institute of Science and Technology

Prediction of Case 512 for the 1981-1982
AFOSR-HTTM-Stanford Conference on Complex Turbulent Flows

by

S.M. Chang, T. Han and J.A.C. Humphrey
University of California
Berkeley, California 94720

in collaboration with

B.E. Launder
University of Manchester
Institute of Science and Technology
Manchester, England

1. Introduction

The following pages contain a summary of the computational methodology and experience gained at the University of California, Berkeley, in relation to the prediction of Case 512 [1] for the 1981-1982 AFOSR-HTTM-Stanford Conference on Complex Turbulent Flows. The present contribution is part of a more extensive and coordinated effort aimed at documenting the performance of various (similar or related) turbulence models embodied in a class of numerical procedures familiar to the various collaborating groups. The institutions participating in this collective effort are listed in the computational summary presented elsewhere in this conference volume by Launder, Leschziner and Sindir. The results appear in the conference volume under the code heading "LHHGM".

The contribution summarized here is based on the use of a two-equation (k-ε) model of turbulence as presented in, for example, [2]. The numerical algorithm solving finite difference forms of the transport equations is the Imperial College TEACH-2E code generalized to three-dimensional (3-D) flows [1,3] and subsequently extended as described in [4] to encompass turbulent flows. A "semi-elliptic" version of the numerical procedure, developed along the lines of the work in [5], was recently completed [6] and includes the use of the QUICK scheme for convective differentiation proposed in [7] and tested in [8]. The principal results prepared for the conference volume were calculated using the semi-elliptic version of the 3-D code using the higher order QUICK scheme for convective differentiation in the cross-stream plane of the flow. Additional predictions of Case 512 using the HYBRID differencing scheme of the standard TEACH codes have also been made but these are less accurate.

While qualitative features of the 90 degree curved duct flow of Case 512 are well represented by the numerical calculations these yield poor quantitative agreement with the measurements. The discrepancies are attributed principally to the failure of the two-equation model to account for large-scale anisotropy in the flow.

2.1 Equations, Turbulence Model and Boundary Conditions

Time-averaged continuity and momentum equations governing steady, developing, incompressible, isothermal, turbulent flow in cylindrical coordinates are given by [4]:

Continuity

$$\frac{\partial U_r}{\partial r}+\frac{1}{r}\frac{\partial U_\theta}{\partial \theta}+\frac{\partial U_z}{\partial z}+\frac{U_r}{r}=0. \qquad (1)$$

Momentum

$$\rho\left[U_r\frac{\partial U_r}{\partial r}+\frac{U_\theta}{r}\frac{\partial U_r}{\partial \theta}+U_z\frac{\partial U_r}{\partial z}-\frac{U^2_\theta}{r}\right]=-\frac{\partial P}{\partial r}+\frac{1}{r}\frac{\partial}{\partial r}\left(\mu_{eff}\, r\frac{\partial U_r}{\partial r}\right)$$
$$\boxed{+\frac{1}{r}\frac{\partial}{\partial \theta}\left(\mu_{eff}\frac{1}{r}\frac{\partial U_r}{\partial \theta}\right)}+\frac{\partial}{\partial z}\left(\mu_{eff}\frac{\partial U_r}{\partial z}\right)-\mu_{eff}\frac{2U_r}{r^2}-\frac{2}{r^2}\mu_{eff}\frac{\partial U_\theta}{\partial \theta}+S_r; \qquad (2)$$

$$\rho\left[U_r\frac{\partial U_\theta}{\partial r}+\frac{U_\theta}{r}\frac{\partial U_\theta}{\partial \theta}+U_z\frac{\partial U_\theta}{\partial z}+\frac{U_rU_\theta}{r}\right]=-\frac{1}{r}\frac{\partial P}{\partial \theta}+\frac{1}{r}\frac{\partial}{\partial r}\left(\mu_{eff}\, r\frac{\partial U_\theta}{\partial r}\right)$$
$$\boxed{+\frac{1}{r}\frac{\partial}{\partial \theta}\left(\frac{\mu_{eff}}{r}\frac{\partial U_\theta}{\partial \theta}\right)}+\frac{\partial}{\partial z}\left(\mu_{eff}\frac{\partial U_\theta}{\partial z}\right)-\mu_{eff}\frac{U_\theta}{r^2}+\frac{2}{r^2}\mu_{eff}\frac{\partial U_r}{\partial \theta}+S_\theta; \qquad (3)$$

$$\rho\left[U_r\frac{\partial U_z}{\partial r}+\frac{U_\theta}{r}\frac{\partial U_z}{\partial \theta}+U_z\frac{\partial U_z}{\partial z}\right]=-\frac{\partial P}{\partial z}+\frac{1}{r}\frac{\partial}{\partial r}\left(\mu_{eff}\, r\frac{\partial U_z}{\partial r}\right)$$
$$\boxed{+\frac{1}{r}\frac{\partial}{\partial \theta}\left(\mu_{eff}\frac{1}{r}\frac{\partial U_z}{\partial \theta}\right)}+\frac{\partial}{\partial z}\left(\mu_{eff}\frac{\partial U_z}{\partial z}\right)+S_z; \qquad (4)$$

where

$$S_r=\frac{1}{r}\frac{\partial}{\partial \theta}\left(\mu_t\, r\frac{\partial}{\partial r}\left(\frac{U_\theta}{r}\right)\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(\mu_t\, r\frac{\partial U_r}{\partial r}\right)+\frac{\partial}{\partial z}\left(\mu_t\frac{\partial U_z}{\partial r}\right)-\mu_t\frac{U_r}{r^2};$$

$$S_\theta=\frac{1}{r}\frac{\partial}{\partial \theta}\left(\mu_t\left(2\frac{U_r}{r}\boxed{+\frac{1}{r}\frac{\partial U_\theta}{\partial \theta}}\right)\right)+\frac{\partial}{\partial r}\left(\frac{\mu_t}{r}\left(\frac{\partial U_r}{\partial \theta}-U_\theta\right)\right)+\frac{\partial}{\partial z}\left(\frac{\mu_t}{r}\frac{\partial U_z}{\partial \theta}\right)+\frac{\mu_t}{r}\left(\frac{\partial U_\theta}{\partial r}-\frac{U_\theta}{r}\right);$$

$$S_z=\frac{1}{r}\frac{\partial}{\partial \theta}\left(\mu_t\frac{\partial U_\theta}{\partial z}\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(\mu_t\, r\frac{\partial U_r}{\partial z}\right)+\frac{\partial}{\partial z}\left(\mu_t\frac{\partial U_z}{\partial z}\right)$$

and

$$\mu_{eff}=\mu+\mu_t\simeq\mu_t$$

The turbulent viscosity, μ_t, is assumed to be determined uniquely by the local values of density ρ, turbulent kinetic energy k, and a turbulent length scale ℓ. At high Reynolds numbers ℓ is proportional to $k^{3/2}/\varepsilon$, where ε is the rate of dissipation of turbulent kinetic energy and thus [2]:

$$\mu_t=C_\mu\rho k^2/\varepsilon, \qquad (5)$$

where C_μ has the constant value given below. The spatial variation of μ_t is determined by solving transport equations for k and ε in cylindrical coordinates, readily derived from the general tensor equtions given in [9], i.e.:

$$\rho\left[U_r\frac{\partial k}{\partial r}+\frac{U_\theta}{r}\frac{\partial k}{\partial \theta}+U_z\frac{\partial k}{\partial z}\right]=\frac{1}{r}\frac{\partial}{\partial r}\left(\frac{\mu_{eff}}{\sigma_k}r\frac{\partial k}{\partial r}\right)\boxed{+\frac{1}{r^2}\frac{\partial}{\partial \theta}\left(\frac{\mu_{eff}}{\sigma_k}\frac{\partial k}{\partial \theta}\right)}+\frac{\partial}{\partial z}\left(\frac{\mu_{eff}}{\sigma_k}\frac{\partial k}{\partial z}\right)+G-\rho\epsilon, \tag{6}$$

and

$$\rho\left[U_r\frac{\partial \epsilon}{\partial r}+\frac{U_\theta}{r}\frac{\partial \epsilon}{\partial \theta}+U_z\frac{\partial \epsilon}{\partial z}\right]=\frac{1}{r}\frac{\partial}{\partial r}\left(\frac{\mu_{eff}}{\sigma_\epsilon}r\frac{\partial \epsilon}{\partial r}\right)\boxed{+\frac{1}{r^2}\frac{\partial}{\partial \theta}\left(\frac{\mu_{eff}}{\sigma_\epsilon}\frac{\partial \epsilon}{\partial \theta}\right)}+\frac{\partial}{\partial z}\left(\frac{\mu_{eff}}{\sigma_\epsilon}\frac{\partial \epsilon}{\partial z}\right)+C_{\epsilon 1}\frac{\epsilon}{k}G-C_{\epsilon 2}\rho\frac{\epsilon^2}{k}, \tag{7}$$

with

$$\begin{aligned}G=\mu_t\Bigg\{2\Bigg[&\left(\frac{\partial U_r}{\partial r}\right)^2+\left(\frac{1}{r}\frac{\partial U_\theta}{\partial \theta}\right)^2+\left(\frac{\partial U_z}{\partial z}\right)^2-\frac{U_\theta}{r}\left(\frac{1}{r}\frac{\partial U_r}{\partial \theta}+\frac{\partial U_\theta}{\partial r}\right)\\&+\frac{U_r}{r}\left(\frac{U_r}{r}+\frac{2}{r}\frac{\partial U_\theta}{\partial \theta}\right)+\frac{1}{r}\left(\frac{\partial U_r}{\partial \theta}\frac{\partial U_\theta}{\partial r}+\frac{\partial U_z}{\partial \theta}\frac{\partial U_\theta}{\partial z}\right)+\frac{\partial U_r}{\partial z}\frac{\partial U_z}{\partial r}\Bigg]\\&+\left(\frac{U_\theta}{r}\right)^2+\left(\frac{\partial U_\theta}{\partial r}\right)^2+\left(\frac{\partial U_\theta}{\partial z}\right)^2+\left(\frac{1}{r}\frac{\partial U_r}{\partial \theta}\right)^2+\left(\frac{\partial U_r}{\partial z}\right)^2+\left(\frac{\partial U_z}{\partial r}\right)^2+\left(\frac{1}{r}\frac{\partial U_z}{\partial \theta}\right)^2\Bigg\}.\end{aligned} \tag{8}$$

The constants in these equations were taken as $C_\mu = 0.09$, $C_{\varepsilon 1} = 1.47$, $C_{\varepsilon 2} = 1.92$, $\sigma_k = 1.0$ and $\sigma_\varepsilon = 1.3$, in accordance with the recommendations in [10].

In all the above equations capital letters denote mean quantities. Components of the Reynolds stress tensor in the momentum equations have been modeled according to the Boussinesq approximation, relating stresses to mean flow gradients through the turbulent viscosity μ_t. Terms enclosed in boxes were not included in the semi-elliptic numerical procedure.

Equations 1-7 were solving using the boundary conditions summarized in Table 1.

2.2 Numerical Procedure

Finite difference forms of the transport equations were obtained by volume integration over cells discretizing the flow domain as explained in [5] to generate a semi-elliptic calculation scheme. In this scheme the neglect of streamwise diffusion in the momentum equations allows a "parabolic" treatment of velocity, requiring two-dimensional storage of velocity components at only two streamwise locations. Elliptic effects are retained in the numerical procedure through three-dimensional storage of pressure. Of course, the use of this scheme precludes the calculation of streamwise flow recirculation.

Further discussion regarding the development and application of the semi-elliptic calculation scheme is available in [5,6].

2.3 Test Cases

The cases listed in Table 2 were predicted to test the worthiness and accuracy of the numerical procedure. In addition to the laminar flow tests two turbulent flow calculations were conducted to verify the two-equation turbulence model for conditions where it is known to yield fairly accurate results. The two-dimensional flow cases were predicted by imposing two (streamwise) symmetry plane conditions in the 3-D semi-elliptic calculation scheme.

3. Remarks on the Prediction of Case 512

A comparison between measurements and predictions of Case 512 obtained by us shows that although qualitative agreement has been established, quantitative agreement is rather poor. The calculations were performed on an equally spaced grid of refinement $(r = 14) \times (z = 10) \times (\theta = 36)$ in the curved duct. The grids in the upstream and downstream tangents were $(14 \times 10 \times 37)$ and $(14 \times 10 \times 17)$ respectively. Computation costs prohibited optimizing the grid distribution. A typical converged run time for these grids was 3.6×10^{-5} CPUs per node visitation and required 135 k_8 words of storage. The criterion for convergence was that the maximum normalized residual summation should be less than 10^{-3}. A comparison between QUICK-generated and HYBRID-generated+ calculations for both laminar and turbulent curved duct flow showed clearly the superior performance of the former scheme for the same number of equivalently distributed grid nodes.

The use of the QUICK scheme in the cross-stream plane of the flow and the streamwise refinement allowed by the semi-elliptic scheme suggest that it is turbulence model defficiency rather than numerical diffusion which produces the discrepancies observed.

The use of a C_μ function (as opposed to a constant value of 0.09) along the lines of [11] did not appear to improve the calculated results.

Initially, calculations were performed using the straight duct developed flow data provided to the Stanford Conference organizers by A. Melling. Calculations using this data revealed an extra pair of small counter-rotating vortices at the outer-radius wall of the curved duct. Calculations using the mass-adjusted data* provided by Melling in Figure A5·10 of reference [15] or in which the upstream tangent cross-stream motion was suppressed did not reveal the second pair of outer-radius wall vortices. Since the measurements corresponding to Case 512 [1,4] do not show nor suggest the presence of a second pair of vortices it is believed that the predictions based on the mass adjusted

+The HYBRID schem employs central differencing when the cell Peclet number is $|Pe| \leq 2$ and upwind differencing when it is $|Pe| > 2$.

*Mass-adjusting had the effect of removing some of the asymmetry in the cross-stream velocity profiles.

data are the more accurate of the two. It is also worth noting that differences between the sets of calculations with (mass adjusted) and with upstream tangent cross-stream flow suppressed were not significantly different. This is attributed to the pressure-dominated nature of the flow in the curved duct. To some extent, such a condition relieves the need for a very accurate specification of the cross-stream flow magnitude at the entrance plane.

In our opinion accurate numerical calculations of this case study and similar curved duct flows [12] could probably be started with the entrance plane located nearer to the 0° plane of the curved duct and with only a specification of the main flow component. Measurements at $x = -2.5$ hydraulic diameters in [1] support this contention and continued research at Berkeley should help quantify this point.

Acknowledgement

The present numerical study was made possible through funding by the Office of Naval Research, Contract No. N00014-80-C-0031. We are particularly grateful to Mr. Keith Ellingsworth for his assistance in obtaining this funding.

Table 1: Boundary Conditions for Prediction of Case 512

Streamwise Location[+]	Variable
Entrance plane ($x = -7.84$; in upstream tangent)	U_θ taken from A. Melling's data provided by Stanford Conference organizers U_r, U_z taken from Figure A5.10 in A. Melling's Ph.D. Thesis $k \equiv 1/2\,(\overline{u_1^2} + \overline{u_2^2} + \overline{u_3^2})$; taken from A. Melling's data provided by Stanford conference organizers. $\varepsilon = k^{3/2}/(0.01)$
Exit plane ($x = 2.24$; in downstream tangent)	Pressure at the exit plane is updated every iteration by adding to each node of the preceding (streamwise) plane the calculated average pressure drop which ensures overall mass-flow continuity at the exit plane. This results in values of $\partial P/\partial r$ and $\partial P/\partial z$ at the exit plane being fixed to the values of the preceding plane.
Side Walls ("P" denotes node nearest wall)	U_θ, U_r and U_z boundary conditions specified by imposing wall shear stress through law of the wall: $$\tau_w \simeq \tau_p = \frac{\rho C_\mu^{1/4} k_p^{1/2} U_p}{A \ln\{y_p C_\mu^{1/4} k_p^{1/2}/\nu\} + B}$$ $A = 2.39$; $B = 5.45$ k_P found from transport equation with diffusion neglected and generation according with wall shear stress. ε_P determined by requiring that the turbulence length scale vary linearly with distance from wall and assuming local equilibrium: $$\epsilon_p = A\frac{C_\mu^{3/4} k_p^{3/2}}{y_p}.$$

[+]Flow geometry and notation are defined in 1980-1981 AFOSR-HTTM-Stanford Conference on Complex Turbulent Flow; Summary, Flow 512, pp. 361-379 and 393-398.

Table 2: Predicted Test Cases

Case	Agreement with experimental, numerical or theoretical data	Reference for Comparison	Comments
Curved channel (2-D); developing laminar flow	Excellent	[12]	HYBRID scheme used for calculations on a $(r = 15) \times (\theta = 36)$ grid.
Curved channel (2-D); developing turbulent flow	Very good	[12]	HYBRID scheme used for calculations on a $(r = 15) \times (\theta = 36)$ grid.
Straight duct (3-D); developing laminar flow	Excellent	[13]	HYBRID scheme used for calculations on a $31 \times 31 \times 52$ grid. Calculations performed in one quadrant. Results for a $13 \times 13 \times 52$ grid were within 4% of experimental data.
Straight duct (3-D); developing turbulent flow	Very good	[14]	HYBRID scheme used for calculations on a $10 \times 10 \times 240$ grid. Discrepancies attributed to inability to predict cross-stream flow. Comparison restricted to maximum centerline velocity and pressure drop coefficient. Calculations performed in one quadrant.
Straight duct (3-D); developing laminar flow; one wall sliding at right angles to main flow	Excellent	[8]	HYBRID and QUICK scheme used for calculations on a $15 \times 15 \times 80$ grid. QUICK calculations were considerably more accurate than corresponding HYBRID results.
90 degree curved duct (3-D); developing laminar flow	Very good at bend angles of 0°, 30° and 90° but discrepancies of order 30% found at 60°.	[1]	HYBRID $(z = 15 \times r = 25 \times \theta = 36)$ and QUICK $(z = 11 \times r = 17 \times \theta = 36)$ schemes used for calculations. Upstream and downstream tangents attached of length -2.6 and 5.4 hydraulic diameters respectively. Coarse grid QUICK scheme results are as accurate as refined grid HYBRID scheme calculations.

References

1. Humphrey, J.A.C. 1977, Flow in ducts with curvature and roughness Ph.D. Thesis, University of London.

2. Launder, B.E. and Spalding, D.B. 1974, The numerical computation of turbulent flows. Comp. Meths. Appl. Mech. Engr. 3, 269.

3. Humphrey, J.A.C. 1978, Numerical calculation of developing laminar flow in pipes of arbitrary curvature radius. Can. J. Chem. Eng. 56, 151.

4. Humphrey, J.A.C., Whitelaw, J.H. and Yee, G. 1981, Turbulent flow in a square duct with strong curvature. J. Fluid Mech. 103, 443.

5. Pratap, V.S. 1975, Flow and heat transfer in curved ducts. Ph.D. Thesis, University of London.

6. Humphrey, J.A.C. 1981, Measurement and calculation of incompressible turbulent flow in a U-bend and downstream tangent of square cross-section. 2nd yearly final report to the Office of Naval Research - Contract No. N00014-80-C-0031 (to appear). Report prepared in collaboration with S.M. Chang, T. Han, B.E. Launder and A. Modavi.

7. Leonard, B.P. 1979, A stable and accurate convective modelling procedure based on quadratic upstream interpolation. Comp. Meths. Appl. Mech. Eng. 19, 59.

8. Han, T., Humphrey, J.A.C. and Launder, B.E. 1981, A Comparison of hybrid and quadratic-upstream differencing in high Reynolds number elliptic flows. Comp. Meths. Appl. Mech. Eng. (to appear).

9. Bryant, D. and Humphrey, J.A.C. 1976, Conservation equations for laminar and turbulent flows in general three-dimensional curvilinear coordinates. Imperial College, Mech. Engrg. Rep. No. CHT/76/6.

10. Patankar, S.V., Pratap, V.S. and Spalding, D.B. 1975, Prediction of turbulent flow in curved pipes. J. Fluid Mech. 67, 583.

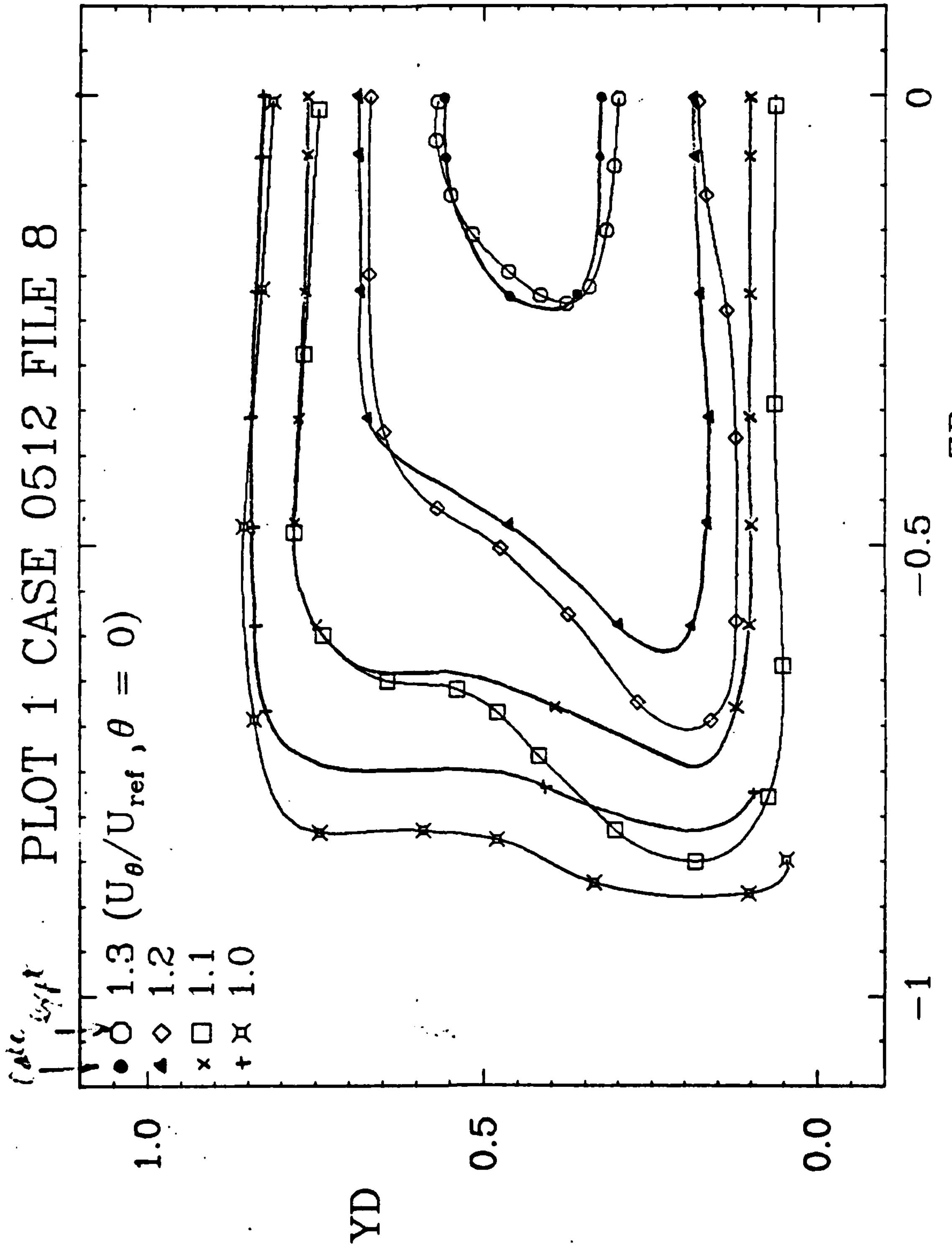
11. Humphrey, J.A.C. and Pourahmadi, F. 1981, A generalized algebraic relation for predicting developing curved channel flow with a k-ϵ model of turbulence. Paper presented at the Third Symposium on Turbulent Shear Flow, University of California, Davis, Sept. 9-11.

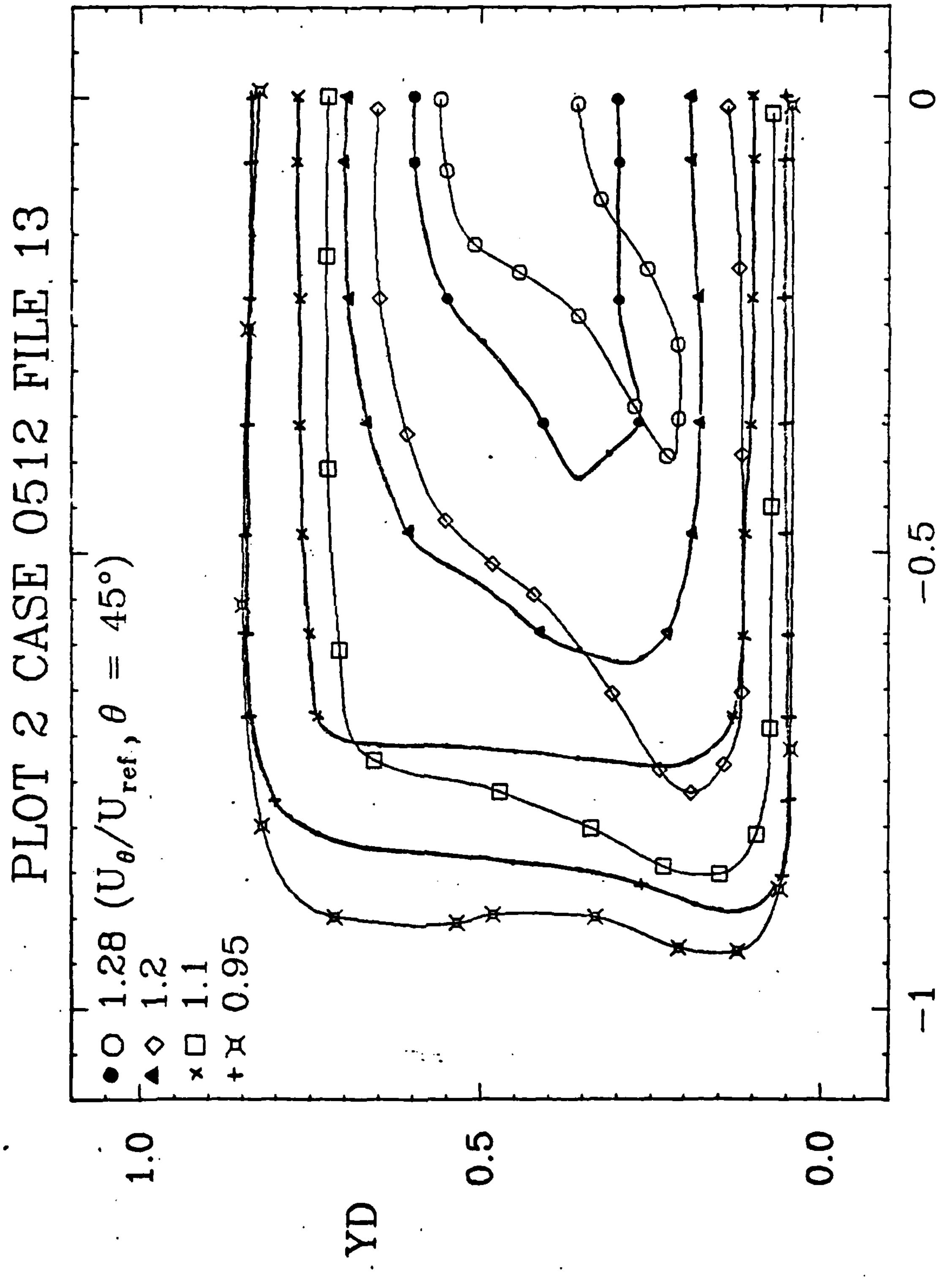
12. Buggeln, R.C. Briley, W.R. and McDonald, H. 1980, Computation of laminar and turbulent flow in curved ducts, channels, and pipes using the Navier-Stokes equations. Final report No. R80-92006-F to the Office of Naval Research.

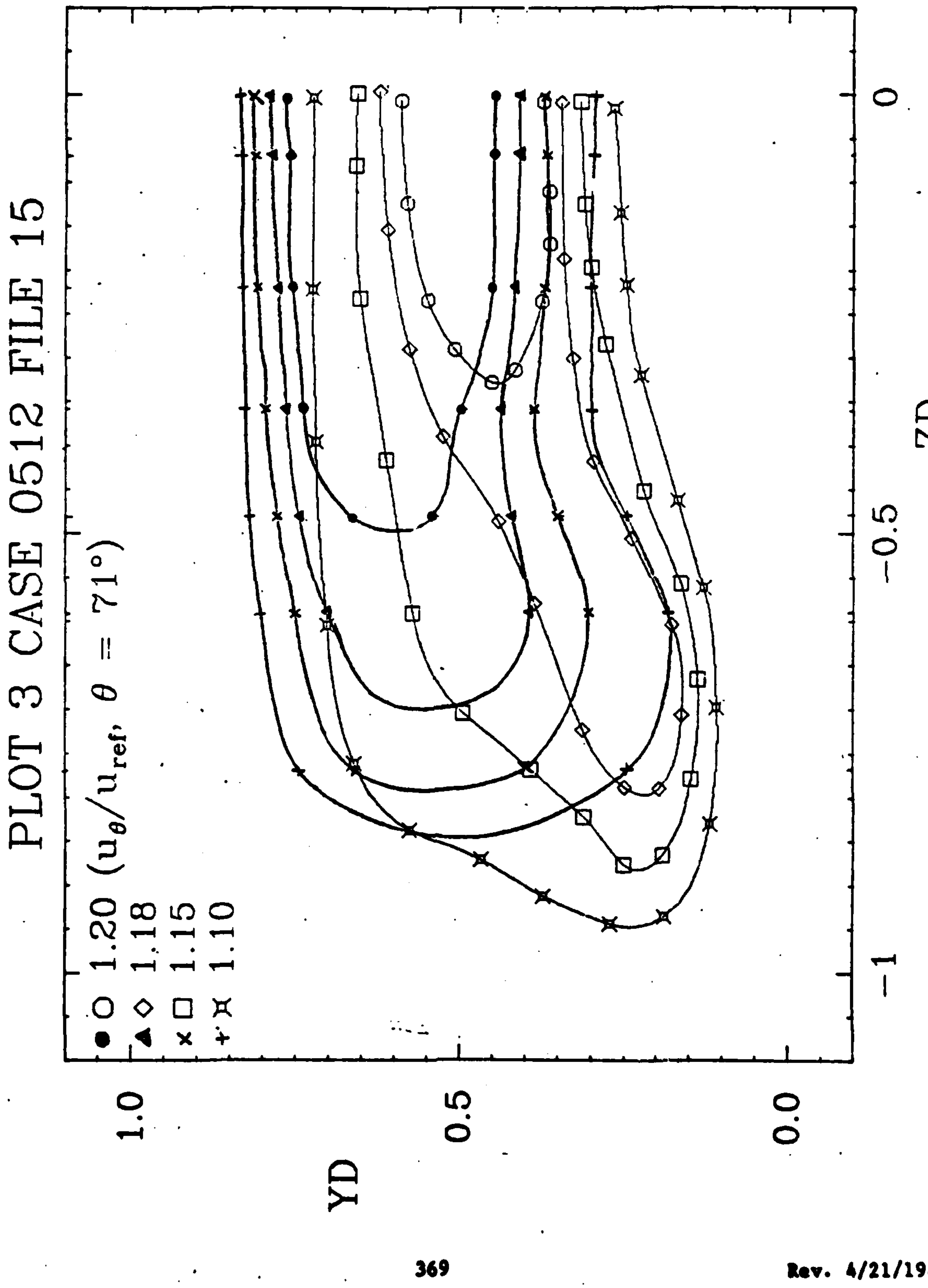
13. Patankar, S.V. and Spalding, D.B. 1972, A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. Int. J. Heat Mass Transfer 15, 1787.

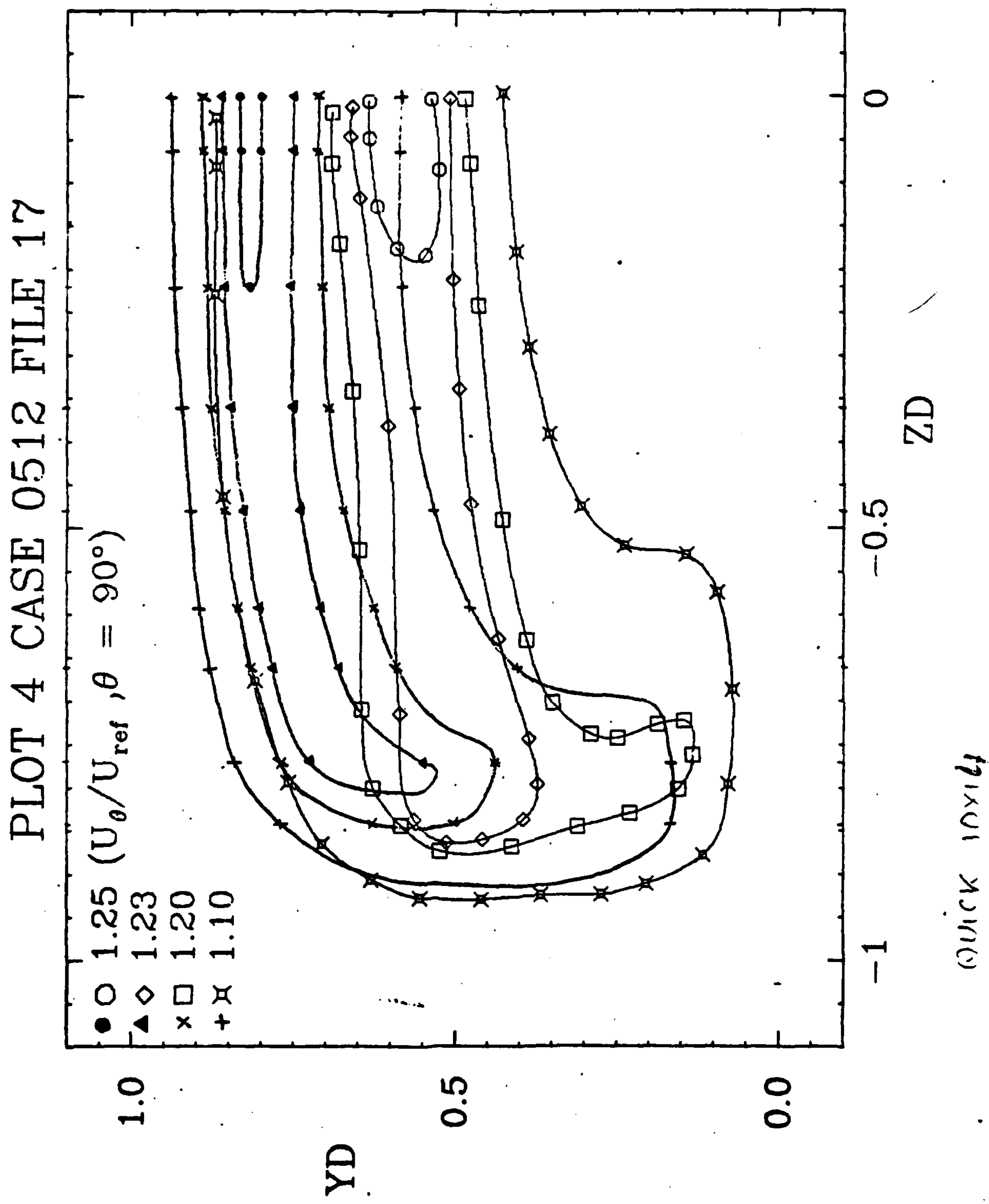
14. Gessner, F.B., Po, J.K. and Emery, A.F. 1979, Measurements of developing turbulent flow in a straight duct. Turbulent Shear Flows I, Springer-Verlag, Berlin. Editors: F. Durst, B.E. Launder, F.W. Schmidt and J.H. Whitelaw.

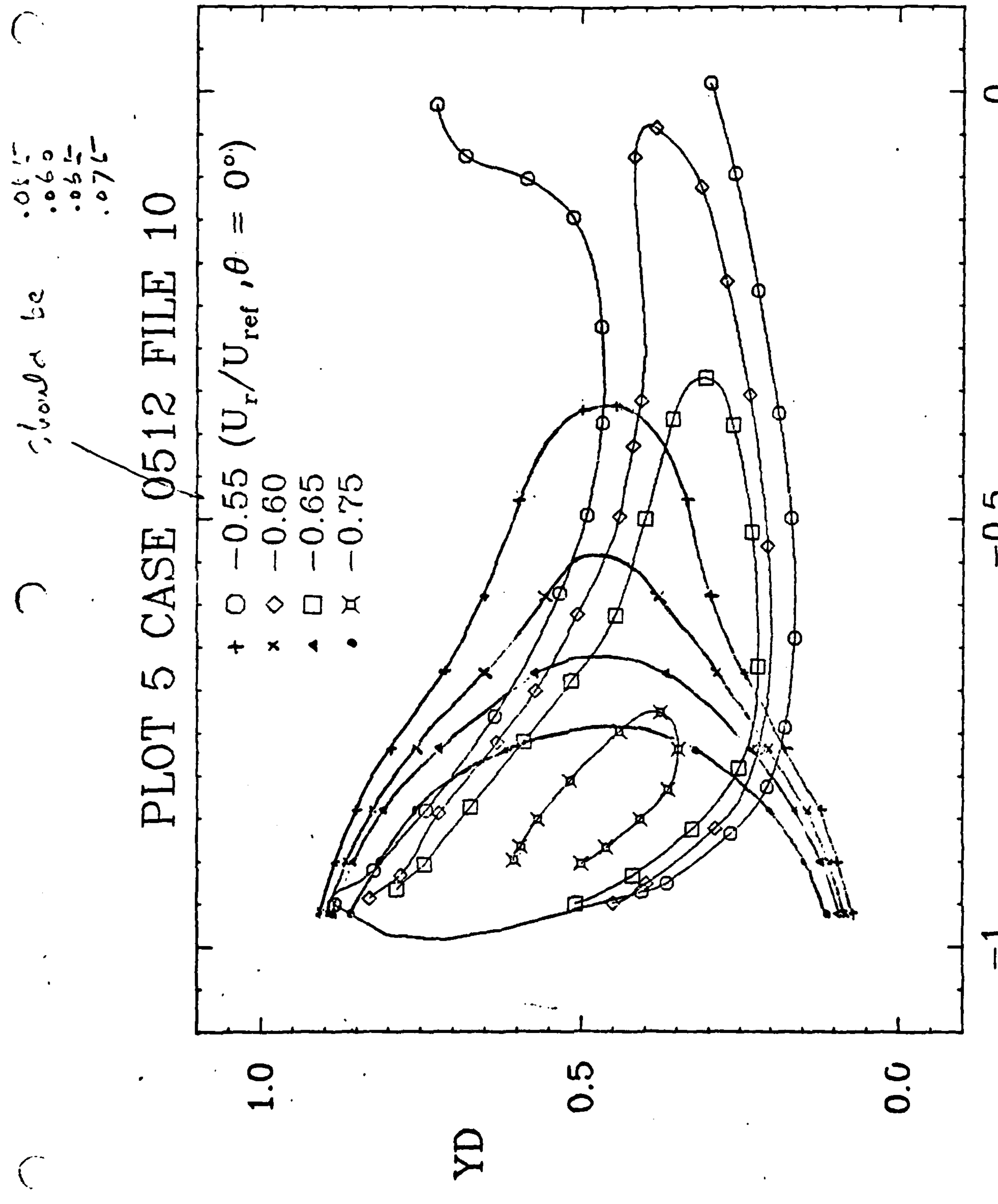
15. Melling, A. 1975, Investigation of flow in non-circular ducts and other configurations by laser Doppler anemometry. Ph.D. Thesis, University of London.

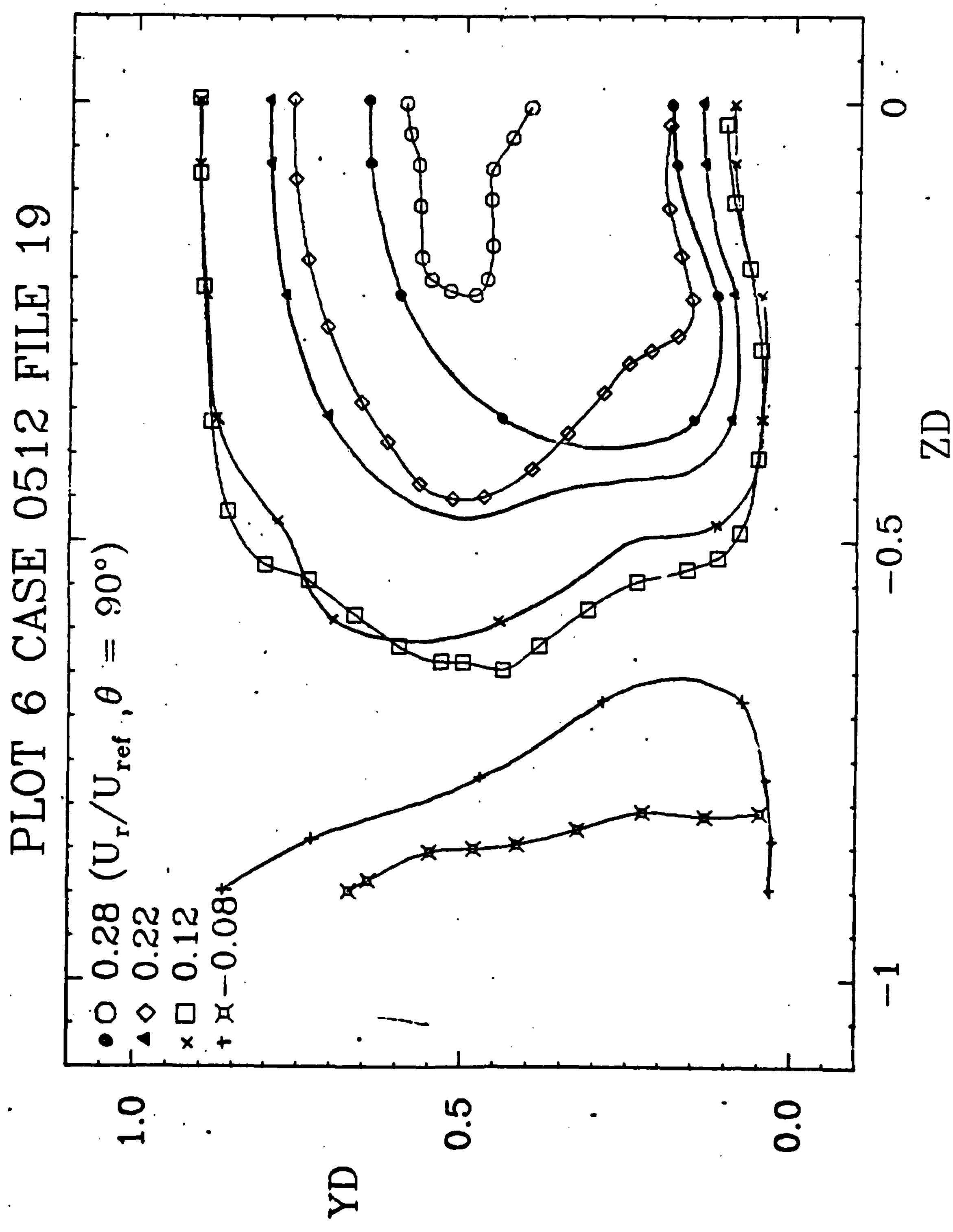

PLOT 1 CASE 0512 FILE 8
calc expt
1.3 (U_θ/U_{ref} , θ = 0)
1.2
1.1
1.0
0
−0.5
−1
ZD
1.0
0.5
0.0
YD


PLOT 2 CASE 0512 FILE 13
(U_θ/U_ref, θ = 45°)
● ○ 1.28
▲ ◇ 1.2
× □ 1.1
+ ¤ 0.95
YD
1.0
0.5
0.0
ZD
0
-0.5
-1


PLOT 3 CASE 0512 FILE 15
1.20 (u$_\theta$/u$_{ref}$, θ = 71°)
1.18
1.15
1.10
YD
1.0
0.5
0.0
ZD
0
−0.5
−1


PLOT 4 CASE 0512 FILE 17
(U_θ/U_{ref}, θ = 90°)
● ○ 1.25
▲ ◇ 1.23
× □ 1.20
+ ¤ 1.10
YD
1.0
0.5
0.0
ZD
0
−0.5
−1


PLOT 5 CASE 0512 FILE 10
+ ○ −0.55 (U_r/U_{ref} ,θ = 0°)
× ◇ −0.60
▲ □ −0.65
● ⌘ −0.75
ZD
0
−0.5
−1
YD
1.0
0.5
0.0


PLOT 6 CASE 0512 FILE 19
(U_r/U_{ref}, θ = 90°)
● ○ 0.28
▲ ◇ 0.22
× □ 0.12
+ ⌑ −0.08
YD
1.0
0.5
0.0
ZD
0
−0.5
−1

UNIVERSITY OF CALIFORNIA, BERKELEY

BERKELEY • DAVIS • IRVINE • LOS ANGELES • RIVERSIDE • SAN DIEGO • SAN FRANCISCO SANTA BARBARA • SANTA CRUZ

COLLEGE OF ENGINEERING
MECHANICAL ENGINEERING

BERKELEY, CALIFORNIA 94720

August 24, 1981

Professor S. Kline
Department of Mechanical Engineering
Stanford University
Stanford, CA 94305

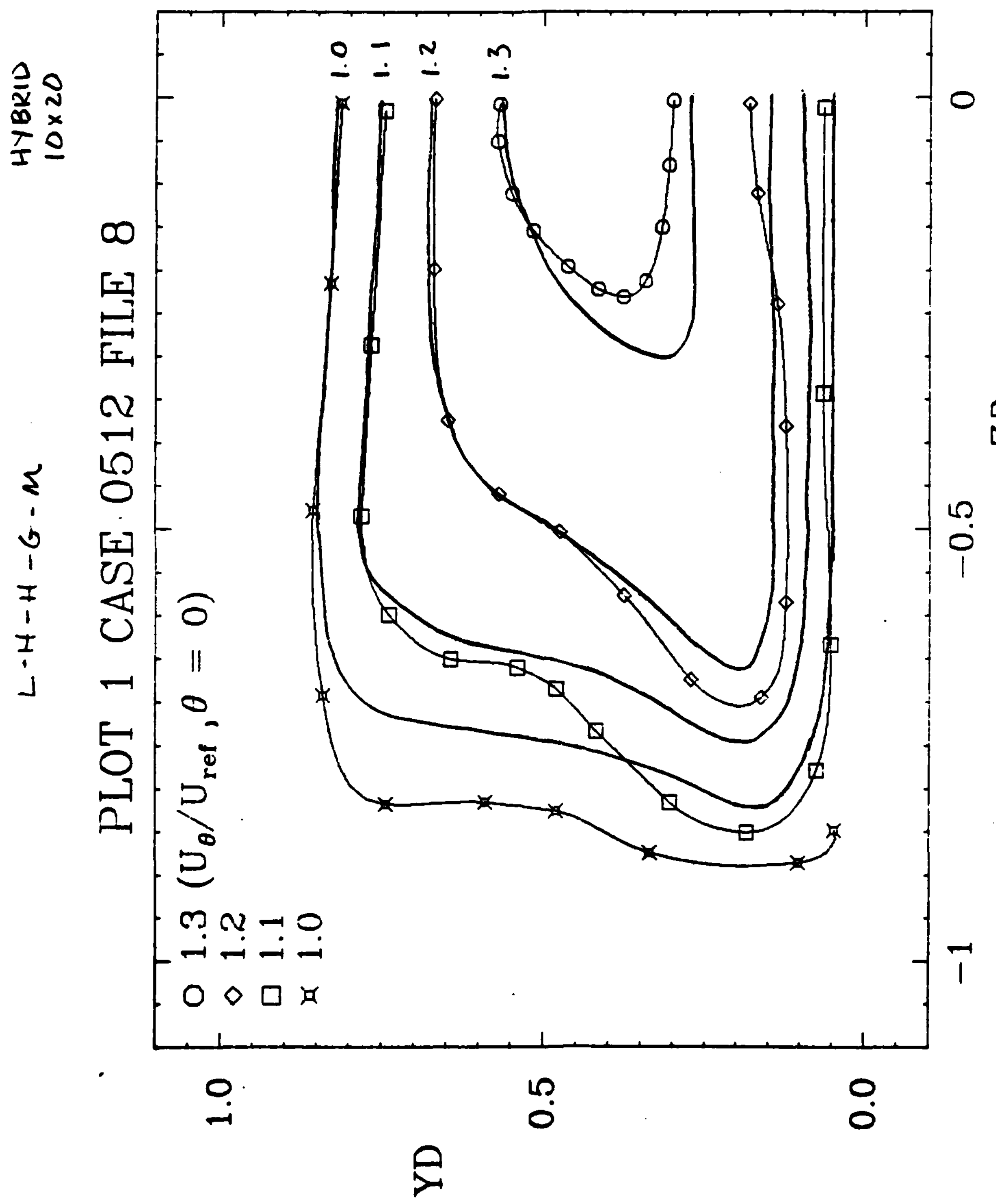
Dear Steve:

Brian Launder asked me to forward the calculations enclosed for reporting at the Stanford Conference. They pertain to Case 512 but, unlike the earlier set you already received from us, the enclosed results were performed using the standard HYBRID differencing scheme contained in the Imperial College TEACH Codes. The earlier calculations already submitted were obtained using the QUICK scheme in the cross-stream plane, and are more accurate. The summary of the Berkeley predictions (to follow soon) will clarify the differences between the two approaches and the consequences of using either.

Sincerely,

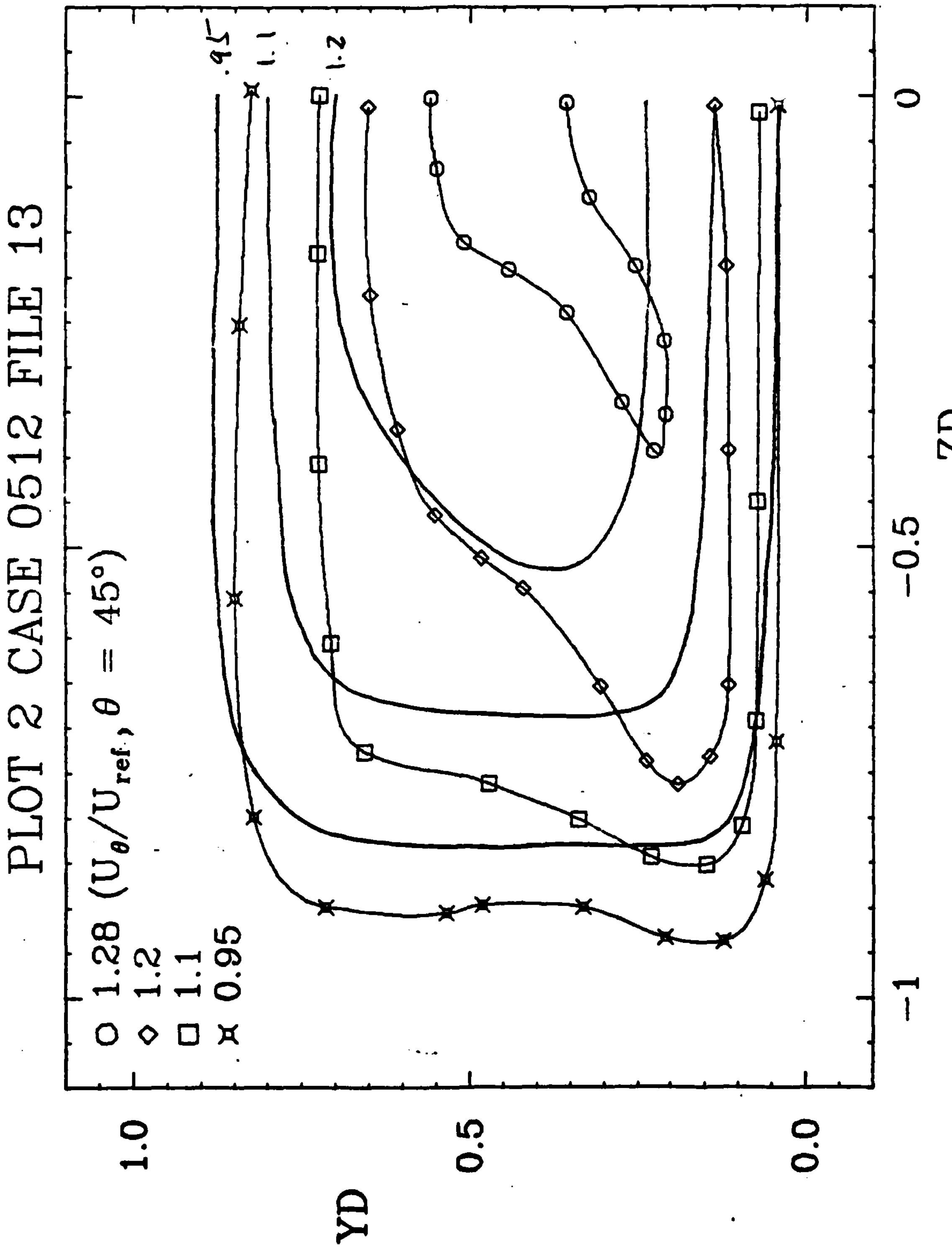
Joseph A.C. Humphrey
Assistant Professor

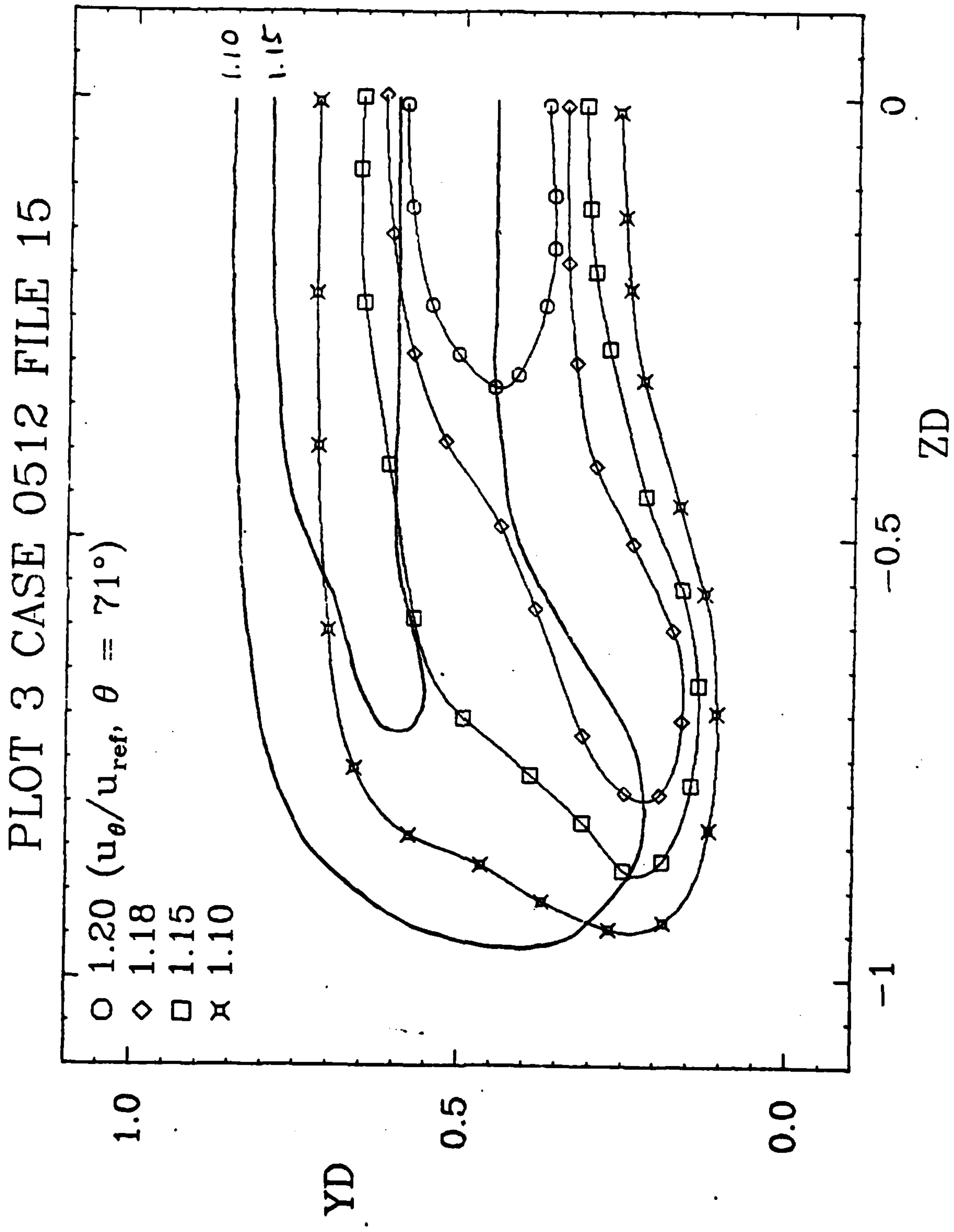
JACH:LCHD
Encl.


HYBRID
10x20
L-H-H-G-M
PLOT 1 CASE 0512 FILE 8
○ 1.3 (U_θ/U_ref, θ = 0)
◇ 1.2
□ 1.1
⌑ 1.0
1.0
1.1
1.2
1.3
0
−0.5
−1
ZD
1.0
0.5
0.0
YD

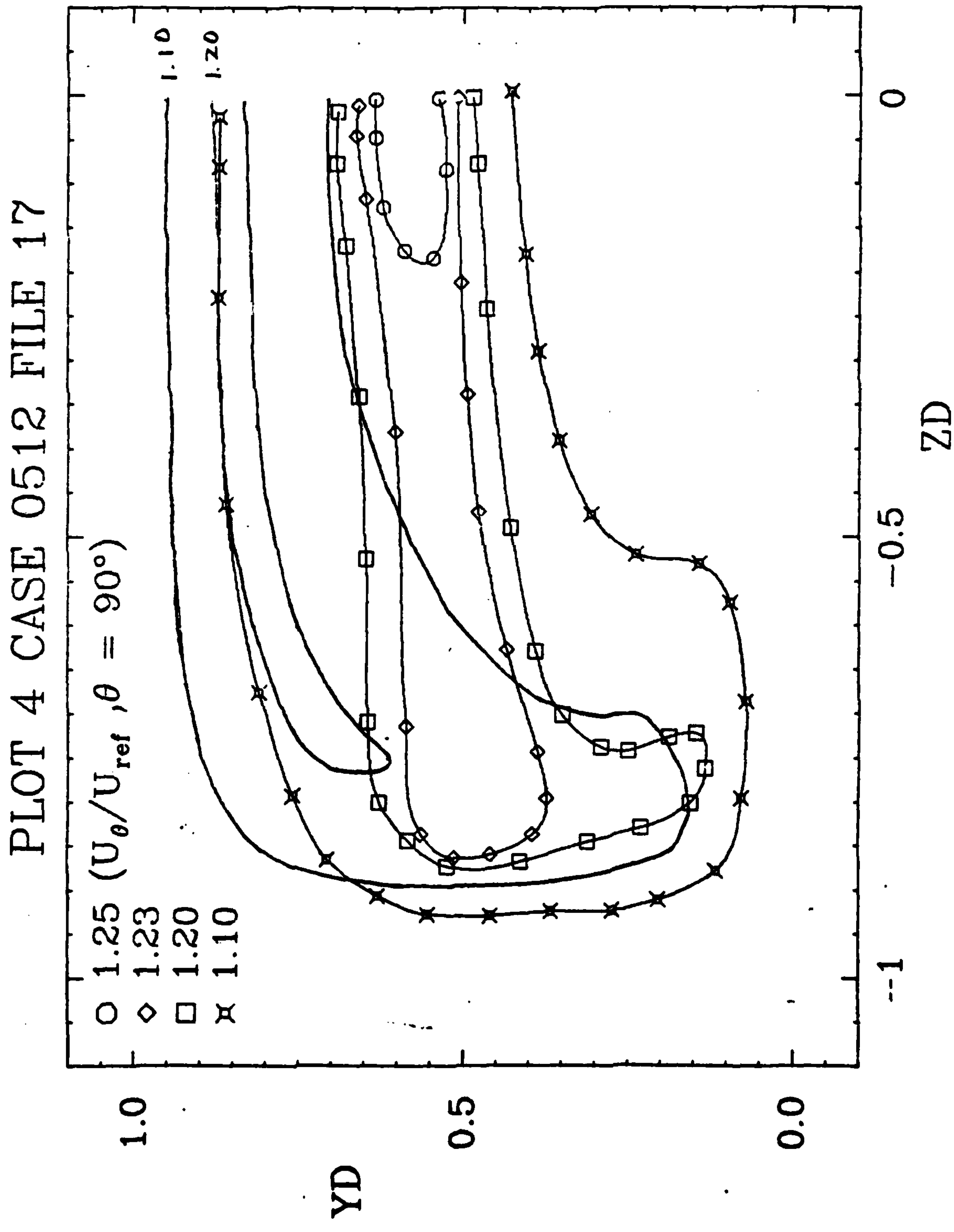
10x20
HYBRID

L-H-H-ε-M




PLOT 3 CASE 0512 FILE 15
(u$_\theta$/u$_{ref}$, θ = 71°)
○ 1.20
◇ 1.18
□ 1.15
⌘ 1.10
1.10
1.15
YD
1.0
0.5
0.0
ZD
0
−0.5
−1

HYBRID


PLOT 4 CASE 0512 FILE 17
(U_θ/U_ref , θ = 90°)
1.25
1.23
1.20
1.10
1.10
1.20
0
−0.5
−1
ZD
1.0
0.5
0.0
YD

DISTRIBUTION LIST

HEAT TRANSFER

	One copy except as noted
Mr. M. Keith Ellingsworth Power Program Office of Naval Research 800 N. Quincy Street Arlington, VA 22203	5
Defense Documentation Center Building 5, Cameron Station Alexandria, VA 22314	12
Technical Information Division Naval Research Laboratory 4555 Overlook Avenue SW Washington, DC 20375	6
Professor Paul Marto Department of Mechanical Engineering US Naval Post Graduate School Monterey, CA 93940	
Professor Bruce Rankin Naval Systems Engineering US Naval Academy Annapolis, MD 21402	
Office of Naval Research Eastern/ Central Regional Office Bldg 114, Section D 666 Summer Street Boston, Massachusetts 02210	
Office of Naval Research Branch Office 536 South Clark Street Chicago, Ill. 60605	
Office of Naval Research Western Regional Office 1030 East Green Street Pasadena, CA 91106	
Mr. Charles Miller, Code 05R13 Crystal Plaza #6 Naval Sea Systems Command Washinton, DC 20362	

Heat Exchanger Branch, Code 5223
National Center #3
Naval Sea Systems Command
Washington, DC 20362

Mr. Ed Ruggiero, NAVSEA 08
National Center #2
Washington, DC 20362

Dr. Earl Quandt Jr., Code 272
David Taylor Ship R&D Center
Annapolis, MD 21402

Mr. Wayne Adamson, Code 2722
David Taylor Ship R&D Center
Annapolis, MD 21402

Dr. Win Aung
Heat Transfer Program
National Science Foundation
Washington, DC 20550

Mr. Michael Perlsweig
Department of Energy
Mail Station E-178
Washington, DC 20545

Dr. W.H. Theilbahr
Chief, Energy Conservation Branch
Dept. of Energy, Idaho Operations Office
550 Second Street
Idaho Falls, Idaho 83401

Professor Ephriam M. Sparrow
Department of Mechanical Engineering
University of Minnesota
Minneapolis, Minnesota 55455

Professor J.A.C. Humphrey
Department of Mechanical Engineering
University of California, Berkeley
Berkeley, California 94720

Professor Brian Launder
Thermodynamics and Fluid Mechanics Division
University of Manchester
Institute of Science & Technology
PO88 Sackville Street
Manchester M601QD England

Professor Shi-Chune Yao
Department of Mechanical Engineering
Carnegie-Mellon University
Pittsburgh, PA 15213

Professor Charles B. Watkins
Chairman, Mechanical Engineering Department
Howard University
Washington, DC 20059

Professor Adrian Bejan
Department of Mechanical Engineering
University of Colorado
Boulder, Colorado 80309

Professor Donald M. McEligot
Department of Aerospace and Mechanical Engineering
Engineering Experiment Station
University of Arizona 85721

Professor Paul A. Libby
Department of Applied Mechanics and Engineering Sciences
University of California San Diego
Post Office Box 109
La Jolla, CA 92037

Professor C. Forbes Dewey Jr.
Fluid Mechanics Laboratory
Massachusetts Institute of Technology
Cambridge, Massachusetts 02139

Professor William G. Characklis
Dept. of Civil Engineering and Engineering Mechanics
Montana State University
Bozeman, Montana 59717

Professor Ralph Webb
Department of Mechanical Engineering
Pennsylvania State University
208 Mechanical Engineering Bldg.
University Park, PA 16802

Professor Warren Rohsenow
Mechanical Engineering Department
Massachusetts Institute of Technology
77 Massachusetts Avenue
Cambridge, Massachusetts 02139

Professor A. Louis London
Mechanical Engineering Department
Bldg. 500, Room 501B
Stanford University
Stanford, CA 94305

Professor James G. Knudsen
Associate Dean, School of Engineering
Oregon State University
219 Covell Hall
Corvallis, Oregon 97331

Professor Arthur E. Bergles
Mechanical Engineering Department
Iowa State University
Ames, Iowa 50011

Professor Kenneth J. Bell
School of Chemical Engineering
Oklahoma State University
Stillwater, Oklahoma 74074

Dr. James Lorenz
Component Technology Division
Argonne National Laboratory
9700 South Cass Avenue
Argonne, Illinois 60439

Dr. David M. Eissenberg
Oak Ridge National Laboratory
P.O. Box Y, Bldg. 9204-1, MS-0
Oak Ridge, Tennessee 37830

Dr. Jerry Taborak
Technical Director
Heat Transfer Research Institute
1000 South Fremont Avenue
Alhambra, CA 91802

Dr. Simion Kuo
Chief, Energy Systems
Energy Research Laboratory
United Technology Research Center
East Hartford, Connecticut 06108

Mr. Jack Yampolsky
General Atomic Company
P.O. Box 81608
San Diego, CA 92138

Mr. Ted Carnavos
Noranda Metal Industries, Inc.
Prospect Drive
Newtown, Connecticut 06470

Dr. Ramesh K. Shah
Harrison Radiator Division
General Motors Corporation
Lockport, New York 14094

Dr. Ravi K. Sakhuja
Manager, Advanced Programs
Thermo Electron Corporation
101 First Avenue
Waltham, Massachusetts 02154

Mr. Robert W. Perkins
Turbotec Products, Inc.
533 Downey Drive
New Britain, Connecticut 06051

Dr. Keith E. Starner
York Division, Borg-Warner Corp.
P.O. Box 1592
York, PA 17405

Mr. Peter Wishart
C-E Power Systems
Combustion Engineering, Inc.
Windsor, Connecticut 06095

Mr. Henry W. Braum
Manager, Condenser Engineering Department
Delaval
Front Street
Florence, New Jersey 08518

Dr. Thomas Rabas
Steam Turbine-Generator Technical Operations Division
Westinghouse Electric Corporation
Lester Branch
P.O. Box 9175 N2
Philadelphia, PA 19113

Professor Daryl Metzger
Chairman, Mechanical and Energy
Systems Engineering
Arizona State University
Tempe, Arizona 85281

END

DATE FILMED

3-82

DTIC