

FLOW AROUND SHIPS' STERNS

BY

A M ABDELMEGUID

THESIS SUBMITTED FOR THE DEGREE OF
DOCTOR OF PHILOSOPHY
IN THE FACULTY OF ENGINEERING
UNIVERSITY OF LONDON
AND
FOR THE DIPLOMA OF MEMBERSHIP
OF IMPERIAL COLLEGE

Heat Transfer Section,
Mechanical Engineering Dept.,
Imperial College of Science & Technology,
London S.W.7

February, 1981.

In the name of Allah, The Most Gracious,
The Most Merciful .

"O Lord, advance me in knowledge"

The Qur'an,
Ta Ha, 115 .

A B S T R A C T

The governing partial-differential equations for three-dimensional turbulent flow around ship's hulls are solved by a finite-difference procedure. A novel non-orthogonal coordinate system has been developed to describe the arbitrary geometry of the hull considered. The velocity components solved for are the axial, radial and circumferential components of the cylindrical-polar system from which the non-orthogonal coordinate system is derived.

The study is limited to situations in which the flow is predominant along the axis of the ship, so that there are no regions of flow 'recirculation'.

The calculation of turbulent flows is made by a two-equation turbulence model. Two additional partial-differential equations are solved for: those for the transport of the kinetic energy of turbulence and for its volumetric rate of dissipation. The turbulent stresses are related to the mean-velocity gradients through a scalar viscosity, calculated from the above turbulence variables. Special practices are adopted for the calculation of flow regions adjacent to the walls.

The calculation method has been applied to two simple flows. The first is the flow between two-concentric

cylinders; the second is for inviscid flow past a Rankine body. The results of the prediction were then compared with the known exact solutions, and the agreement was good.

The calculation method has been applied to the computation of flow around the rear part of a ship. Comparison of the computations with experimental measurements were made, wherever possible, and the agreement is good over most of the hull; very near to the stern, the agreement was not good.

The prediction procedure can be applied easily to the prediction of other three-dimensional turbulent flows around bodies of irregular, but basically cylindrical, shape. Examples of such bodies are: submarine or torpedo hulls, or an aircraft fuselage.

TABLE OF CONTENTSPAGE NO.

ABSTRACT	1
PREFACE	7
 CHAPTER 1 - INTRODUCTION	 12
1.1 Prelude	12
1.2 The Problem Considered	13
1.3 Practical Relevance	16
1.4 Past Work	17
1.4.1 Classification of flows	17
1.4.2 Review	20
1.4.3 Elaborations	25
1.5 The Present Contribution	28
1.6 Outline of the Thesis	29
 CHAPTER 2 - THE MATHEMATICAL PROBLEM	 31
2.1 Introduction	31
2.2 The Governing Differential Equations	31
2.2.1 Choice of the coordinates	31
2.2.2 Basic equations	34
2.2.3 The non-orthogonal curvilinear coordinate system	39
2.3 Turbulence Modelling	50
2.3.1 Approach	50
2.3.2 The k- ϵ turbulence model	51
2.3.3 The near-wall treatment	55

<u>TABLE OF CONTENTS</u> (Continued)	<u>PAGE NO.</u>
2.4 Auxiliary Information	58
2.5 Closure	62
CHAPTER 3 - THE SOLUTION PROCEDURE	64
3.1 Introduction	64
3.2 The Finite-Difference Scheme	65
3.2.1 The finite-difference grid	66
3.2.2 The locations of variables	66
3.2.3 The control volumes	71
3.2.4 Location of flow variables near the boundary	71
3.2.5 Integration of the differential equations	74
3.2.6 Upwind-differencing practice	85
3.3 The Solution Method	86
3.3.1 Main features	86
3.3.2 Sequence of calculation steps	89
3.3.3 Methods of solving the difference equations	93
3.4 Incorporation of Auxiliary Information	96
3.4.1 Initial conditions	96
3.4.2 Boundary conditions	96
3.5 Remarks on the Numerical Procedure	97
3.5.1 Accuracy of the procedure	97
3.5.2 Convergence of the procedure	102

<u>TABLE OF CONTENTS (Continued)</u>	<u>PAGE NO.</u>
3.5.3 Stability	106
3.5.4 Computer time	107
3.6 Closure	107
CHAPTER 4 - THE APPLICATION OF THE PREDICTION METHOD TO TWO SIMPLE TEST CASES	
4.1 Introduction	108
4.2 Test Case (A): Flow in Pipes	109
4.2.1 Preliminaries	109
4.2.2 Inlet and boundary conditions	110
4.2.3 Computational results	112
4.2.4 Results	112
4.3 Test Case (B): Flow Over a Rankine Body	114
4.3.1 Preliminaries	114
4.3.2 Inlet and boundary conditions	117
4.3.3 Computational details	118
4.3.4 Results	119
4.4 Closure	123
CHAPTER 5 - PREDICTION OF FLOW AROUND SHIPS	
5.1 Introduction	125
5.2 General Definition of the Hull	126
5.3 The Test Case	126
5.3.1 Preliminary remarks	126
5.3.2 The test model and the measurements	129
5.3.3 Computational details	132

<u>TABLE OF CONTENTS (Continued)</u>	<u>PAGE NO.</u>
5.4 Presentation of Results	136
5.4.1 Cross-stream plots	136
5.4.2 Circumferential plots	137
5.4.3 Comparison with measurements	144
5.4.4 Turbulence characteristics	156
5.5 Discussion	156
5.5.1 Cross-stream plots	156
5.5.2 Circumferential plots	159
5.5.3 Comparison with measurements	160
5.5.4 Turbulence characteristics	162
5.5.5 Summary	163
5.6 Comparison Between the Parabolic and Partially-Parabolic Solutions	165
5.7 Closure	167
 CHAPTER 6 - CONCLUDING REMARKS	
6.1 Present Achievements	169
6.2 Suggestions for Future Work	170
 REFERENCES	172
 NOMENCLATURE	181
 APPENDIX - Published paper on "Turbulent Flow and Heat Transfer in Pipes with Buoyancy Effects"	184

P R E F A C E

This thesis presents the major part of my research activities during the period October 1975 to March 1980. These research activities centred mainly around the development and validation of finite-difference calculation procedure to predict three-dimensional flow around ships' hulls. The study was carried out under a research contract with the Science Research Council (SRC), and was supervised by Professor D Brian Spalding.

Upon my arrival at the Mechanical Engineering Department of Imperial College, I spent several months acquainting myself with the available numerical methods in the department. It was felt at that time, that such an acquaintance would be the logical first step in the process of developing the current prediction procedure. Therefore, I have started simultaneously on two computational programmes; "GENMIX", a two-dimensional streamline parabolic programme; and "STABLE", a three-dimensional parabolic and partially-parabolic programme. Further acquaintance was gained through the post-graduate courses which I attended in the department. During this early period of my research activity, I was supported financially by CHAM Limited; thereafter, the research funds were continuously provided by SRC.

After six months, it was natural to proceed further towards three-dimensional programmes. The task was to develop a faster and more economical programme than "STABLE". This task was carried out by a team of three, Dr G Mallinson, Ian Jennions and myself. The updating of "STABLE" took more than six months, and resulted in "STABLER", which was the base for my prediction procedure. The additional task of testing and validating "STABLER" was mine, where the programme was examined for many three-dimensional parabolic flow and heat transfer situations. Some of the interesting results of this early work have been presented in a published paper in the Journal of Fluid Mechanics (Abdelmeguid and Spalding, 1979b).

At this stage, the progress in the original project, flow around ship's hulls, was therefore, going very slowly. The research was then speeded up by cooperation between Imperial College and CHAM Limited, and was coordinated by Professor Spalding.

Thereafter, the author worked upon updating "STABLER" to "FLASH", which was tailored specifically for the problem of flow around ships. In early 1978, it was thought that "FLASH" was becoming uneconomical to run, and further economisation was made.

Research was then speeded up on the original project and resulted in the publication of the procedure (Abdelmeguid et al 1978b) and (Abdelmeguid et al 1979a). The procedure had then its limitations and pitfalls, and further improvements were carried out to overcome such disadvantages.

To acquaint myself further with ship flows, I spent three months, in early 1978, at the National Maritime Institute, Teddington, in close cooperation with other researchers working in the same field. Most of the time spent there, was on performing experiments on the waves generated from the motion of a ship. The experiments were conducted at Towing Tank number 1.

In due course of the research activities, I experienced the need to represent the flow patterns; therefore, I developed a graphical package for flow around ships. The results presented in Chapter 5 are plotted with the aid of this graphical package.

Most of the time I spent at Imperial College was directed towards improving the economy of the calculations, and seeking methods to speed up the calculations.

The research activities took a long time, and at the end no more funds were available, therefore, under the directions of Professor Spalding, I began to write up.

The author records that some of his research is not reported in this thesis due to its incompleteness, and feels that with more funds, more results could have been achieved and presented in this thesis.

I now wish to thank all those who provided me with help and encouragement during the present study. First and foremost, I thank Professor Spalding for providing guidance, inspiration and encouragement to maintain a good pace of research. To have worked in association with him, is a privilege.

Thanks are due to Miss Colleen King, Miss Sue Farmiloe and Mrs Maggie Dean, who have been eager to help in personal and administrative matters. The large amount of computations made during my research could not have been made without the smooth running and coordination of the sectional computing services administrated by Peter Dale, to whom I am grateful for providing me with enough computer resources.

Thanks are also due to Dr N Markatos, and other researchers in CHAM Limited, for their assistance during my research.

I am grateful to Ishikawajima-Harima Heavy Industries (IHI) of Japan for providing the measurements used in this thesis. Thanks in particular are due to Mr K Muraoka for several assistances on behalf of IHI which included the provision of potential flow solution at the free boundary.

I acknowledge with deep respect the encouragement provided by my parents, my mother and my beloved father, during my stay abroad, and in general, throughout my educational career. Thanks are also due to my wife for helping me in finding what I am looking for.

Finally, I am grateful to Alana Anthony for typing this thesis.

A M Abdelmeguid
February 1981*

* This thesis is a revised form of a thesis presented in May 1980. The revision is made to fulfil the requirements of the examiners. The author's grant to do this work was finished in April 1980, thus certain financial difficulties delayed this revised form.

CHAPTER 1 - INTRODUCTION

1.1 Prelude

There are many flow situations of practical importance which are characterised by the immersion of a body in a steady stream of wide extent, and with the possibility of the presence of small regions of recirculation (i.e. backward flow) resulting from the incompleteness of streamlining. Examples of such a flow type are the flow around ships, natural-draught cooling towers and other chimney-like structures, bridge piers in rivers, road vehicles, trains, and aeroplanes and rockets. The knowledge of the flow field around such bodies is of great importance in various design processes.

It is only since the development of high-speed computers that knowledge of complicated flow phenomena, such as the one described above, has become possible in a much faster and more economical fashion than by the way of the existing experimental methods.

The present study has been directed mainly towards the study of flow around ships' hulls; however, extension of the method to other shapes is not a difficult task.

1.2 The Problem Considered

Flow around ship's hulls is complicated in its nature; near to the hull, the flow is viscous; away from it, the flow is irrotational and its nature appears as a set of linkage between pressures.

A schematic representation of the flow around a ship is shown in Figure 1.2.1. In the forward region (from a to b), the boundary layer is thin on the main hull, and pressure variations along normals are small. In the midship region (from b to c), the boundary layer starts to thicken, and, while the pressure variations along normals are not large, the boundary-layer displacement starts to influence the external flow.

In the stern region (from c to d), the boundary-layer further thickens, and large pressure variations along normals occur; furthermore, the external flow becomes strongly influenced by the boundary-layer displacement, and a strong cross-flow considerably affects the flow pattern. The stern region is also characterised by a tendency of the boundary layer to separate from the hull, and downstream curvature effects to travel upstream.

The interest of the present study is to focus attention on the flow in the stern region; however, because of the influence of the midship region on the stern one,

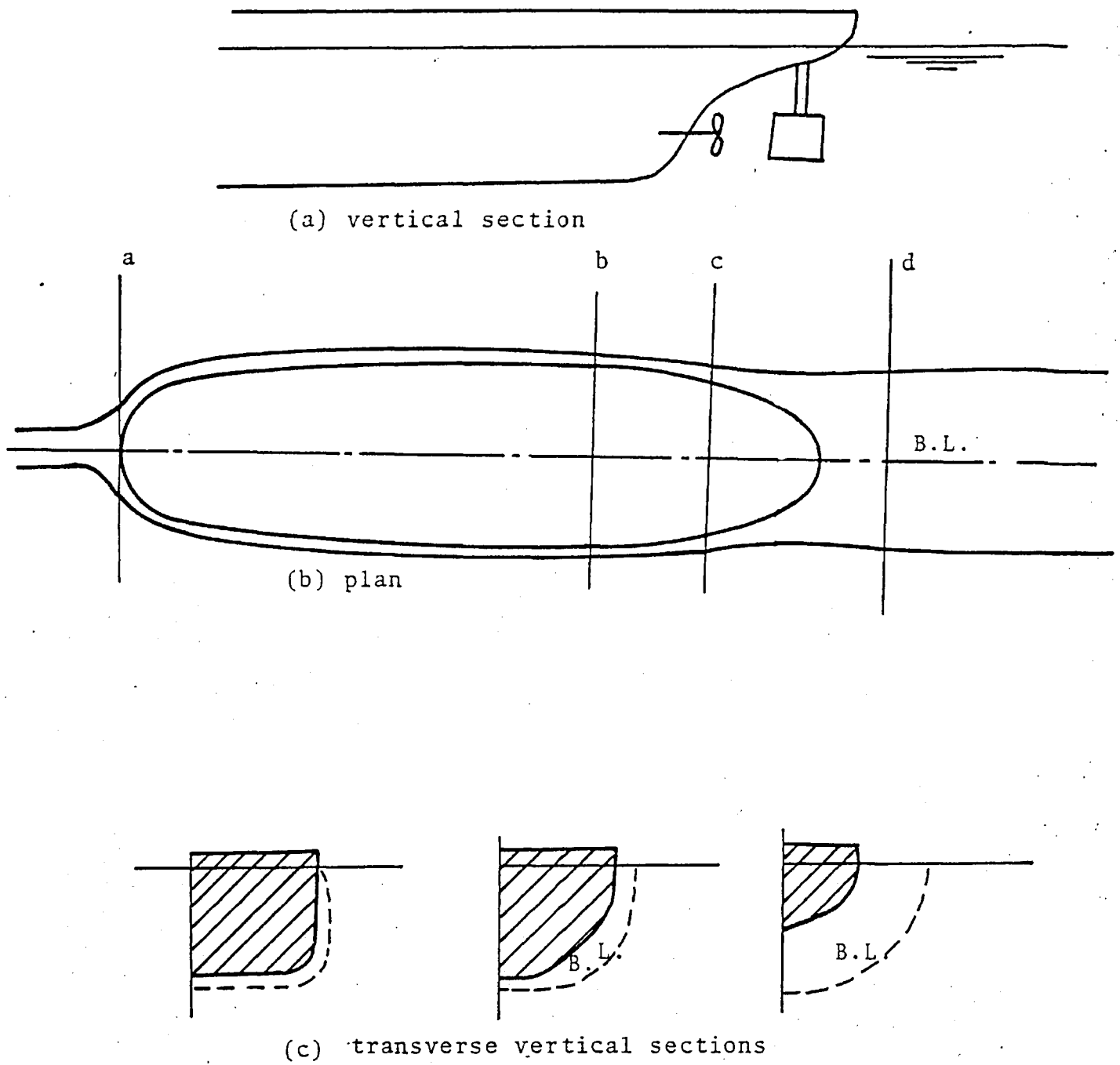


Figure 1.2.1 Schematic representation of flow around ships.

the midship region will be studied along with the stern region.

The present study describes the application of a finite-difference solution procedure to the partial-differential equations governing the steady, three-dimensional, turbulent flow around ships' sterns. The prediction procedure is composed of a numerical scheme for the solution of the partial-differential equations which describe the transport of mass and momentum in such flows, and a hypothesis for modelling the turbulence.

The present study has been restricted to flow situations which are characterised by a predominant flow direction along which there are no regions of flow "recirculation". The existence of a predominant flow direction ensures that:

- (a) flow properties are convected only from upstream regions to downstream and not vice versa; and that,
- (b) at moderate and large Reynolds numbers, molecular actions, such as conduction, diffusion and viscous action, along that direction, are small enough to be negligible.

These characteristics simplify the equations governing the flow and make them easier to be solved.

Validation of the prediction procedure is achieved by comparing predicted results for a real ship with experimental data. An assessment of the prediction procedure is then made.

For turbulent flows, the additional "stresses" due to turbulence have been represented through a two-equation turbulence model which comprised two additional partial-differential equations for the transport of the kinetic energy of turbulence and the volumetric rate of its dissipation.

1.3 Practical Relevance

In recent years, the interest in developing prediction methods for the flow around ships' hulls has grown rapidly. The major reason is that, with the advent of the age of large ships and supertankers the designers were met with lack of knowledge on the fundamental nature of flow. Such knowledge is very important in the design of the hull, propellers, etc. For example, of the many possible shapes of ships' hulls which satisfy the structural design conditions, only one will give the minimum drag at the design speed. Similarly,

for an accurate construction of the propeller as to efficiency and cavitation, the complicated flow approaching the propeller must be known.

Previous experiences in this field were mainly centred on model or prototype methods. Such experiences are very expensive to establish, and are time-consuming. Prediction methods are much faster and more economical than experimental ones, in gaining the required knowledge.

Prediction methods can also be very helpful in improving the performance of the already existing hulls, by finding the appropriate necessary alternations to the hull which will result in better performance.

1.4 Past Work

1.4.1 Classification of flows

It has been useful (Gosman et al, 1969) in numerical fluid dynamics to classify steady-flow phenomena into two classes: parabolic and elliptic. A flow is called parabolic if:

- (i) there exists a predominant direction of flow
(i.e. there is no reverse flow in that direction);
- (ii) the diffusion of momentum, mass, etc is negligible in that direction; and,

- (iii) the downstream pressure field has negligible effect on the upstream flow conditions.

Many duct flows, jet and boundary layer phenomena are of the parabolic kind; i.e. upstream conditions can determine the downstream flow properties, but not vice versa. A flow is called elliptic when at least the last two of the three prerequisites for parabolic flows are not satisfied.

There is a class of flow situation, however, in which although the flow is strictly elliptic, (because the downstream pressure field has a marked effect on the upstream flow conditions) the diffusion effect of the transport properties in the predominant direction is small enough to be neglected. This class of flow is called partially-parabolic (Spalding, 1975). Flow situations falling into the partially-parabolic class include the flow in strongly-curved ducts, the flow in stern regions, and others.

It has been customary (Spalding, 1974) to name such a flow with a predominant flow direction as a boundary-layer flow. It is also customary to classify such flows with respect to many points, such as dimensionality, number of equations to be solved, pressure gradients, etc.

From the view point of dimensionality, the boundary layer may be two-dimensional, two-and-a-half-dimensional, or three-dimensional (Spalding, 1974). Such classification is a practical and useful one because it enables the numerical analyst to choose the most economical method, and because numerical methods do actually exist for problems for all the indicated classes. Plane and axisymmetric flows are two-dimensional, even though three velocity components may be required in the computations; more generally they are three-dimensional.

There is however, an intermediate stage where, despite the three coordinate being retained, only two momentum equations are coupled by the pressure, and the third can be deduced from the other two; this class is termed two-and-a-half-dimensional (Spalding, 1974). The two-and-a-half-dimensional is, therefore, essentially three-dimensional with no variation of the pressure in one of the directions.

The boundary layer may also be classified according to the nature of the boundaries of the domain of integration. The domain may be wholly bounded by solid walls or symmetric surfaces, it may be wholly surrounded by a stream of irrotational fluid of great extent (free boundary) or by both type of boundaries.

In this thesis, the boundary layer is three-dimensional and free.

1.4.2 Review

In spite of the major importance of three-dimensional turbulent boundary layers in engineering, comparatively little work has been done in this field. The main effort has been directed towards the more common two-dimensional cases, which are, of course, easier to handle.

The first comprehensive survey on three-dimensional turbulent boundary layers was due to Nash and Patel (1972). In their systematic studies, Nash and Patel stated down the basic equations governing the flow; the different boundary-layer approximations; the physical models related to turbulence, and the general features of the three-dimensional turbulent boundary-layers. Thereafter, they presented a description of some integral and differential methods of solving these equations. Nash and Patel's studies did not contain any assessment of the available methods of solution, it merely serves as an introduction to the newcomers to the field.

Not long after Nash and Patel's work had been published, Larsson (1974a), presented a complete literature survey on the investigations of three-dimensional

turbulent boundary layers. Larsson's work was a foundation for his later contribution to the particular case of flow around ships.

The first major work on three-dimensional turbulent boundary layers was Euromech 60 Workshop (East, 1975). An assessment of the performance of available calculation methods of three-dimensional boundary-layers was achieved by applying these methods to one set of test cases. In total, six test cases were selected as the base for comparing different computational methods. The test cases were two-dimensional flow leading to separation, infinite swept flow leading to separation, infinite swept flow with decaying crossflow, conic flow beneath a leading-edge vortex, flow induced on a flat plate by a circular cylinder, and wavy wall flow. Two types of calculation methods were used, namely: integral and differential. Nine methods in total were used in these tests, three of which are integral and six are differential. Table 1 shows the different methods. The comparison of the different computation methods showed that integral methods are in error due to the assumptions regarding crossflow profiles; on the other hand, differential methods produce errors if the modelling of the direction of the shear-stress vector is not adequate. Furthermore, the restriction of four of the six differential methods to infinite

swept flows seriously reduced their utility, and the lack of any differential method written in non-orthogonal curvilinear coordinate is likely to make their application to some practical problems difficult.

Type		Authors
Integral methods	1	Michel and Cousteix
	2	Smith
	3	White
Differential	1	Fannelop and Humphreys
	2	Krause and Kordulla
	3	Lindhout et al
	4	Michel and Cousteix
	5	Schneider
	6	Spalding and Singhal
	7	Rodi and Rastogi

TABLE 1: EUROMECH 60 WORKSHOP

Regarding turbulence modelling, although the results obtained by simple mixing-length hypothesis were good in the simple test cases, the results obtained by the two-equation model (Launder and Spalding, 1973) were more accurate in the more complex cases. Moreover, Spalding's method has the advantage of computing a wide range of shear flows of which the test cases examined represent a very restricted set.

Further comparison of different solution methods were established in the EUROVISC working party on three-dimensional shear layers (Humphreys, 1979) where eight methods were applied to a single test case.

In the above-mentioned review, attention was given to the general three-dimensional turbulent boundary layer, with particular emphasis on some aerodynamic problems, e.g. wings.

Some attempts have been made to calculate the viscous resistance of ships. If the flow around the hull of a ship could be calculated with sufficient accuracy, it would be a fairly simple matter to calculate the viscous resistance.

In a recent review, Larsson (1978), examined the available calculation methods for three-dimensional

incompressible turbulent boundary layers. Twenty-four calculation methods were reviewed; thirteen of which were differential. Despite integral methods being the simplest numerically, the necessary empirical input, which is probably more dependent on the geometry than in any other problem, makes them less versatile; the differential methods, therefore, have the greatest potential. Moreover, most existing calculation methods are not fully three-dimensional*, thus their use in the calculation of the flow around ships is limited. Furthermore, many methods are restricted to orthogonal cartesian coordinates, while to be able to carry out accurate calculations on a non-developable surface, the coordinate system should be curvilinear.

Larsson has further examined the different turbulence models that can be applied for the calculation of flow around ships. For thin boundary layers, at midship, the use of a zero-equation turbulence model is well justified; for thick boundary layers, at stern, higher order equations should be used, (the most well-known is probably the one by Launder and Spalding, (1973)).

* Spalding (1974) demonstrated that all three-dimensional boundary-layer methods are of the restricted kind in which only two momentum equations are solved. These kind of boundary-layer methods were classified as two-and-a-half-dimensional (Section 1.5.1).

Summing up, there are only four fully three-dimensional differential methods and five integral ones; of these nine, only three methods can be applied near the stern, i.e. when the boundary layer is thick, they are namely: Larsson's (1974d, 1975); Spalding's (1975) and Tanaka et al's (1977). From the point of view of turbulence, all reviewed methods, except Spalding's method, do not include the higher order effects.*

1.4.3 Elaborations

(a) External effects (free-stream effects)

Theoretically speaking, the dynamic effects of the flow motion past streamlined body reaches to infinity. The flow surrounding the body can be thought of as having two regions, a boundary-layer one, and a potential-flow one. If the conditions and location on the surface separating the two regions are known,

* In Navier-Stoke's equations all terms may be assigned an order of magnitude, expressed as a power of δ/R , where δ is the boundary layer thickness, and R is the local radius of curvature of the surface. In the first order boundary-layer methods, only terms of leading order are retained. When δ becomes large and the ratio δ/R approaches unity, the neglected terms may be important.

then these conditions can be used as the boundary conditions for the boundary-layer calculations. Alternatively, if the boundary-layer thickness is known, then we can compute the potential-flow around the "displace-body"*. Since the boundary-layer calculation is dependent on the potential-flow solution, and vice versa, an iterative procedure must be employed.

The most common iterative procedure in recent years is the one known as "the method of equivalent sources". The method employs an iterative procedure between the potential-flow and the boundary-layer through which adjustments to the sources and sinks at the outer boundary are made. The basis for this method is the normal velocity at the edge of the boundary-layer. Given the distribution of this velocity, a corrective source distribution in the potential flow may be obtained. Of course, iterations have to be carried out between the boundary-layer (where the normal velocity is calculated) and the potential flow (where the pressure is obtained) until we ensure that the pressure-velocity relations at the free-boundary are in proper accordance with the dynamical laws of potential flow.

* The displacement body is defined as the body thickened by the displacement thickness (i.e. boundary-layer thickness).

The method should enable us to adjust the cross-stream velocities at the boundary to fit the source-sink field until the pressures at the boundaries resulting from the finite-difference solution (boundary-layer, internal solution) agree with the pressures at the boundaries corresponding to the source-sink solution (potential flow, external solution).

Such an iterative procedure, the method of equivalent sources, has been used in several methods published recently: Dyne, 1977; Hart and Hotta, 1977.

(b) Effect of surface roughness

One of the important arguments in the prediction of the boundary layer around ships, is the surface roughness of the hull. Designers of ships' hulls are interested in finding out the effect of surface roughness on the drag on ships. Almost all three-dimensional prediction methods have only allowances for smooth surfaces; however, in practice, hull surfaces are rough, and in many cases are irregularly rough (Larsson, 1978). Larsson reviewed several methods which accommodate roughness calculation; however, none of these methods was three-dimensional.

The prediction procedure used in this thesis does have the advantage of using wall functions, by which

different roughness levels can be ascribed to different parts of the hull.

1.5 The Present Contribution

The primary objective of the present work has been to develop and test a numerical prediction procedure which can be used to predict the three-dimensional turbulent flow around ships' hulls. It is aimed that the procedure satisfies all the necessary requirements of a good design tool; i.e. it must be simple, accurate, widely applicable and economical.

The work presented in this thesis can be divided mainly into the following two sections:

(i) Development of the prediction procedure

The prediction procedure has been developed to predict the flow around bodies such as the hull of a ship or a submarine, which are basically elongated but otherwise of arbitrary shape. The cross-sectional shape of the body may vary in any arbitrary, but smooth, way with the axial distance. The prediction procedure was limited, during the present study, to flow situations which are characterised by a predominant flow direction along which there are no regions of flow 'recirculation'. However, the extension of such procedure to include

regions of recirculation is not a difficult task. The prediction procedure was also limited to the first step in the iterative procedure between the boundary layer and the potential flow. The boundary layer calculations were made based on the potential flow calculated based on the true hull.

The prediction procedure employs a finite-difference method to arrive at a set of equations relating the dependent variables together. The equations are solved by the SIMPLE algorithm (Patankar and Spalding, 1972). The finite-difference method employs a novel non-orthogonal finite-difference grid, which was developed specifically for the current study.

(ii) Validation of the prediction procedure

In order to test the capabilities of the prediction procedure, the procedure was applied to predict the flow around a ship. The predictions were then compared with available measurements.

1.6 Outline of the Thesis

The remaining sections of the thesis, which describe the present study in more detail, are arranged thus: in the following chapter, Chapter 2, the partial-differential equations which govern the flow situations

considered in this thesis are derived in the non-orthogonal coordinate system. The 'turbulence-model' approach is also discussed in this chapter. In Chapter 3, the above equations are transformed into their finite-difference equivalents, and the method used for their solution is presented.

In Chapter 4, the application of the method to two simple test cases is reported. The validation of the procedure in predicting flow around ships is reported in Chapter 5.

Chapter 6 summarises the main contributions of the present study and lists the conclusions obtained thereof, along with a few suggestions for future work. The last part of the thesis provides a description of the Nomenclature used, a list of References and an Appendix.

CHAPTER 2 - THE MATHEMATICAL PROBLEM

2.1 Introduction

This chapter describes the formulation of the mathematical problem. The description consists firstly of a brief derivation of the partial differential equations which govern the transport of mass, momentum and other scalar fluid properties in steady three-dimensional boundary layer flows. In this derivation, the equations of the three-dimensional flow motion expressed in cylindrical forms, are used to arrive at a set of equations represented in a novel non-orthogonal curvilinear coordinate system. Thereafter, a brief description of the turbulence model used in the present study is given. Next, the auxiliary information necessary to compute the mathematical formulation of the problem are detailed. Finally, a summary of the assumptions involved in arriving at the mathematical description of the problem is provided.

2.2 The Governing Differential Equations

2.2.1 Choice of the coordinates

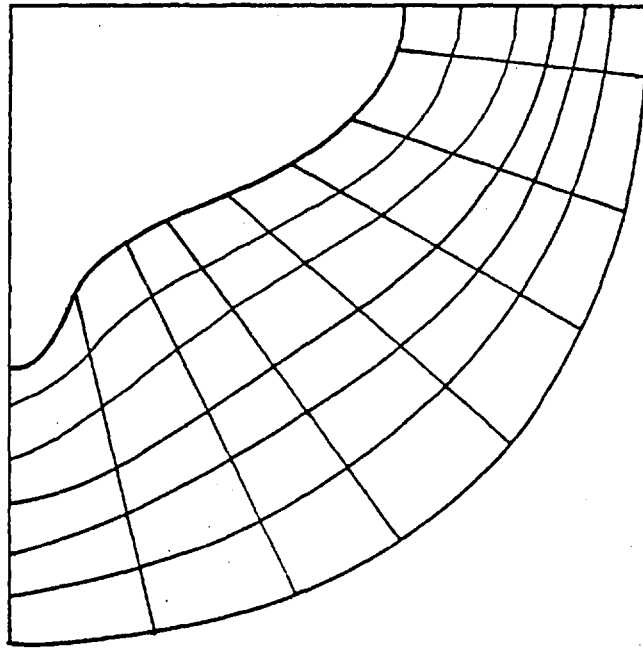
In the calculation of three-dimensional turbulent boundary layers on arbitrary shaped bodies, such as ship forms or shapes of aerodynamic interest, an important problem which has to be resolved is the

choice of a coordinate system. A cartesian or cylindrical coordinate system would present the difficulty in describing the body and excessive unnecessary storage would need to be employed.

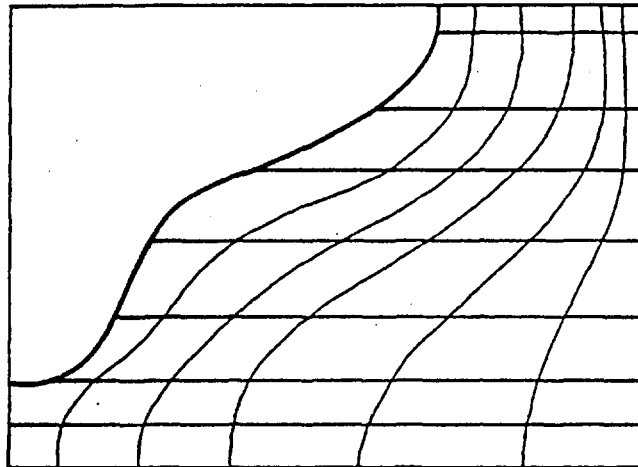
In ship research, for reasons of simplicity, the Navier-Stokes equations are usually written in a curvilinear coordinate system which is orthogonal in space. In such a system it is convenient to choose, for the Navier-Stokes equations as well as for boundary-layer calculations, a system of coordinates such that the surface of the body is one member of a family of ordinate surfaces.

The orthogonal system, despite being a favourite one, does have some well known disadvantages (Larsson, 1974a). The system suggested here, a non-orthogonal one, can be applied to ship forms where the frames are not generally orthogonal to the streamlines. This system would have one axis normal to the axis of the body. Figure 2.2.1 presents the alternative to both the normal coordinate and the orthogonal curvilinear one.

The choice of the cylindrical polar system to be the start of our curvilinear system stems from the fact that much of the hull can be represented with arcs.



a) cylindrical polar based system



b) cartesian based system

Figure 2.2.1 The non-orthogonal system (approach)

2.2.2 Basic equations

The partial-differential equations which govern steady, three-dimensional motion of fluids can be expressed as follows:

Continuity

$$\frac{\partial}{\partial x_i} (\rho U_i) = 0 \quad ; \quad (2.1)$$

Transport of fluid property

$$\frac{\partial}{\partial x_i} (\rho U_i \phi) = \bar{S}_\phi - \frac{\partial J_{\phi,i}}{\partial x_i} \quad ; \quad (2.2)$$

where U_i = velocity component in each of the three coordinate directions, i ;

x_i = corresponding coordinate axis;

$\bar{S}_\phi$ = source (and/or sink) of ϕ ;

$-J_{\phi,i}$ = flux of ϕ along the i th direction (in case of ϕ being a velocity, $-J_{\phi,i}$ would be the complete stress tensor, σ_{ij});

ρ = fluid density; and,

ϕ = transported scalar fluid property (may stand for a velocity component, kinetic energy of turbulence or its dissipation).

In the case of turbulent flows, the symbols for the various flow properties represent the time-averaged values.

For the transport of the velocity component U_j , for example, equation (2.2) takes the form:

$$\rho \frac{\partial}{\partial x_i} (U_i U_j) = F_j - \frac{\partial P}{\partial x_j} - \frac{\partial \tau_{ij}}{\partial x_i} ; \quad (2.3)$$

F_j being a body force acting on the fluid, and P the pressure.

These equations in cylindrical-polar coordinates and in accordance with Figure 2.2.2, takes the following form:

- Longitudinal momentum

$$\begin{aligned} & \frac{1}{r} \frac{\partial}{\partial r} (rvw) + \frac{1}{r} \frac{\partial}{\partial \theta} (uw) + \frac{\partial}{\partial z} (ww) \\ &= - \frac{1}{\rho} \frac{\partial P}{\partial z} + \frac{1}{r} \left\{ \frac{\partial}{\partial r} \left(r \frac{\tau_{rz}}{\rho} \right) + \frac{\partial}{\partial \theta} \left(\frac{\tau_{\theta z}}{\rho} \right) \right\} ; \quad (2.4) \end{aligned}$$

- Radial momentum

$$\begin{aligned} & \frac{1}{r} \frac{\partial}{\partial r} (rvv) + \frac{1}{r} \frac{\partial}{\partial \theta} (uv) + \frac{\partial}{\partial z} (wv) - \frac{u^2}{r} \\ &= - \frac{1}{\rho} \frac{\partial P}{\partial r} + \frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\tau_{rr}}{\rho} \right) + \frac{1}{r} \frac{\partial}{\partial \theta} \left(\frac{\tau_{r\theta}}{\rho} \right) - \frac{\tau_{\theta\theta}}{\rho r} \\ & ; \quad (2.5) \end{aligned}$$

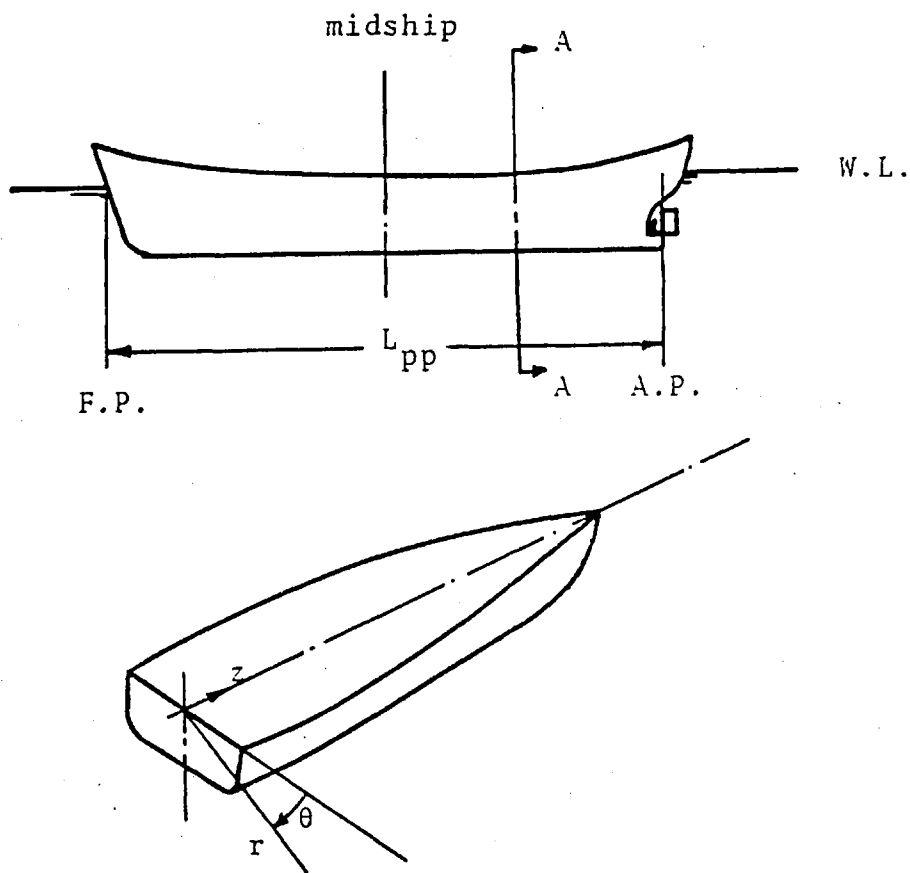


Figure 2.2.2 The coordinate system (r, θ, z)

• Angular momentum

$$\begin{aligned} & \frac{1}{r} \frac{\partial}{\partial r} (rvu) + \frac{1}{r} \frac{\partial}{\partial \theta} (uu) + \frac{\partial}{\partial z} (wu) + \frac{uv}{r} \\ &= - \frac{1}{\rho} \frac{P}{r \partial \theta} + \frac{1}{r^2} \frac{\partial}{\partial r} (r^2 \frac{\tau_{r\theta}}{\rho}) + \frac{1}{r} \frac{\partial}{\partial \theta} (\frac{\tau_{\theta\theta}}{\rho}); \quad (2.6) \end{aligned}$$

where τ 's represent shear stresses and are expressed in terms of the velocity gradients and an effective viscosity, which varies in the field.

The expressions for the τ 's are as follows:

$$\tau_{rr} = \Gamma_{\text{eff}} (2 \frac{\partial v}{\partial r}) \quad ; (2.7)$$

$$\tau_{\theta\theta} = \Gamma_{\text{eff}} 2 (\frac{1}{r} \frac{\partial u}{\partial \theta} + \frac{v}{r}) \quad ; (2.8)$$

$$\tau_{r\theta} = \tau_{\theta r} = \Gamma_{\text{eff}} (\frac{\partial u}{\partial r} - \frac{u}{r} + \frac{1}{r} \frac{\partial v}{\partial \theta}) \quad ; (2.9)$$

$$\tau_{z\theta} = \tau_{\theta z} = \Gamma_{\text{eff}} (\frac{1}{r} \frac{\partial w}{\partial \theta}) \quad ; (2.10)$$

$$\tau_{zr} = \tau_{rz} = \Gamma_{\text{eff}} (\frac{\partial w}{\partial r}) \quad ; (2.11)$$

where Γ_{eff} is an effective diffusion coefficient, in laminar flows $\Gamma_{\text{eff}} = \mu_{\text{laminar}}$; in turbulent flows, Γ_{eff} will be calculated in a different manner.

The above equations have been reduced from the three-dimensional form by neglecting axial diffusion of

momentum; such an assumption agrees with the well-known boundary layer assumptions.

The above equations are applicable both to the parabolic and partially-parabolic flows. Further simplification can be made in the case of parabolic flows*, where the axial pressure gradient, $-\frac{1}{\rho} \frac{\partial p}{\partial z}$, can be assumed to be uniform over the whole section, and is equal to $-\frac{1}{\rho} \frac{\partial \bar{p}}{\partial z}$ (Patankar and Spalding, 1972b).

* For parabolic flows, the downstream pressure field has little effect on the upstream flow condition, thus permitting the pressure in the momentum equation for the predominant flow direction to be decoupled from the pressure of the momentum equations in the other two directions. Thus, the pressure gradient appearing in the momentum equation for the predominant flow direction is:

$$-\frac{1}{\rho} \frac{\partial \bar{p}}{\partial z}$$

where $\bar{p}$ stands for the average static pressure over a cross-section. This decoupling of pressures is a necessary part of the parabolic procedure (Patankar and Spalding, 1972b). It permits a marching-integration technique to be employed. It is worth noting that the omission of this practice does not result in an increase in the accuracy for flows which are truly parabolic.

For partially-parabolic flows, the above stated pressure-decoupling practice cannot be applied. Therefore, no alternations to equations (2.4) to (2.6) is made.

Within the above framework, the partial-differential equations which govern steady, three-dimensional motion of fluids in cylindrical coordinates (r, θ, z) can be expressed in the general form:

$$\begin{aligned} & \frac{1}{r} \frac{\partial}{\partial r} (rv\phi) + \frac{1}{r} \frac{\partial}{\partial \theta} (u\phi) + \frac{\partial}{\partial z} (w\phi) \\ &= \frac{1}{r} \frac{\partial}{\partial r} (\Gamma_{\phi} r \frac{\partial \phi}{\partial r}) + \frac{1}{r^2} \frac{\partial}{\partial \theta} (\Gamma_{\phi} \frac{\partial \phi}{\partial \theta}) + \bar{S}_{\phi}. \end{aligned} \quad (2.12)$$

In the above equation, ϕ , is taken to represent any fluid property transported by the flow, including the three components of momentum per unit mass u , v and w .

Equation (2.12) also includes the continuity equation, for which $\phi=1$ and, Γ_{ϕ} and $\bar{S}_{\phi}$ are zeros. Γ_{ϕ} is the diffusion coefficient for ϕ and is given for each of the ϕ 's considered by the present study, together with the source/sink term $\bar{S}_{\phi}$, in Table 2.

2.2.3 The non-orthogonal curvilinear coordinate system

(a) Definition

The governing differential equations are expressed in a non-orthogonal curvilinear system of coordinates instead of the more usual cartesian or cylindrical-polar

Equation	ϕ	Γ	$\bar{S}_\phi$
Continuity	1	0	0
w-momentum	w	μ_{eff}	$-\frac{\partial P}{\partial z}$
u-momentum	u	μ_{eff}	$\frac{1}{r^2} \frac{\partial}{\partial \theta} (\mu_{\text{eff}} \frac{\partial u}{\partial \theta}) + \frac{\partial}{\partial r} (\mu_{\text{eff}} \frac{\partial v}{r \partial \theta})$ $+ \frac{2}{r^2} \frac{\partial}{\partial \theta} (\mu_{\text{eff}} v) - \frac{1}{r} \frac{\partial}{\partial r} (\mu_{\text{eff}} u)$ $+ 2 \frac{\mu_{\text{eff}}}{r^2} \frac{\partial v}{\partial \theta} + \frac{\mu_{\text{eff}}}{r} \frac{\partial u}{\partial r}$ $- \mu_{\text{eff}} \frac{u}{r^2} - \frac{uv}{r} - \frac{\partial P}{r \partial \theta}$
v-momentum	v	μ_{eff}	$\frac{\partial}{\partial r} (\mu_{\text{eff}} \frac{\partial v}{\partial r}) + \frac{1}{r} \frac{\partial}{\partial \theta} (\mu_{\text{eff}} \frac{\partial u}{\partial r})$ $+ \frac{\mu_{\text{eff}}}{r} \frac{\partial v}{\partial r} - 2 \frac{\mu_{\text{eff}}}{r^2} \frac{\partial u}{\partial \theta}$ $- \frac{1}{r^2} \frac{\partial}{\partial \theta} (\mu_{\text{eff}} u) - 2 \mu_{\text{eff}} \frac{v}{r^2}$ $+ \frac{u^2}{r} - \frac{\partial P}{\partial r}$

TABLE 2: THE DIFFUSION COEFFICIENTS AND THE SOURCE TERMS OF EQUATION (2.12).

systems so as to allow the geometry of the hull to be specified easily.

The elements of the present system (η, ξ, ζ) are defined in terms of orthogonal, cylindrical-polar coordinates (r, θ, z) , as follows:

$$\begin{aligned}\eta &= \frac{r-r_S}{r_N-r_S} && ; \\ \xi &= \frac{\theta-\theta_W}{\theta_E-\theta_W} && ; \quad (2.13) \\ \zeta &= z && .\end{aligned}$$

The definition (2.13) can best be appreciated with reference to Figure 2.2.3.

The flow region of interest is, therefore, contained wholly within the surfaces $\xi=0$, $\xi=1$, $\eta=0$ and $\eta=1$.

The subscripts N, S, E and W, refer respectively to the North, South, East and West boundaries of the calculation domain in the $r-\theta$ domain.

r_S defines the hull radius, where r_N defines the outer boundary. r_N and r_S are, in general, functions of both the angular and axial position, and define the height of the flow domain.

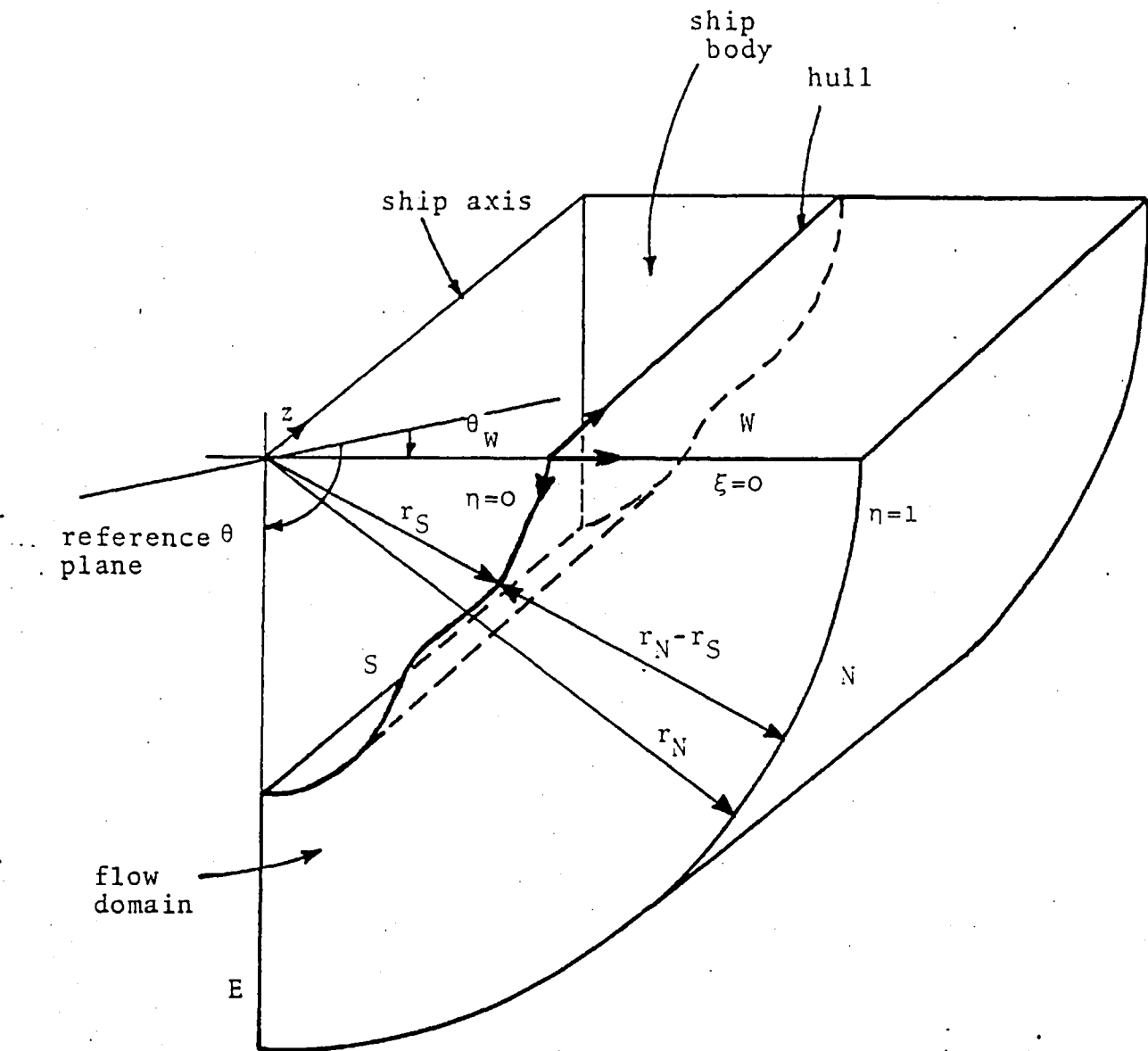


Figure 2.2.3 The η, ξ, ζ coordinate system

θ_E defines the vertical symmetry line whereas θ_W defines the water surface. θ_E and θ_W are, in general, functions of the axial position and define the angular width of the flow domain.

In the present contribution θ_E , θ_W and r_N are taken as constant, whereas r_S is made to follow the hull surface.

The coordinate system has one axis η normal to the axis of the body. The coordinates (ξ, η, ζ) satisfy the general relationship:

$$\begin{aligned}\frac{\partial}{\partial r} &= \frac{\partial}{\partial \xi} \frac{\partial \xi}{\partial r} + \frac{\partial}{\partial \eta} \frac{\partial \eta}{\partial r} + \frac{\partial}{\partial \zeta} \frac{\partial \zeta}{\partial r} & ; \\ \frac{\partial}{\partial \theta} &= \frac{\partial}{\partial \xi} \frac{\partial \xi}{\partial \theta} + \frac{\partial}{\partial \eta} \frac{\partial \eta}{\partial \theta} + \frac{\partial}{\partial \zeta} \frac{\partial \zeta}{\partial \theta} & ; \quad (2.14) \\ \frac{\partial}{\partial z} &= \frac{\partial}{\partial \xi} \frac{\partial \xi}{\partial z} + \frac{\partial}{\partial \eta} \frac{\partial \eta}{\partial z} + \frac{\partial}{\partial \zeta} \frac{\partial \zeta}{\partial z} & .\end{aligned}$$

The following mathematical consequences result from the definition (2.13) in accordance with the relations (2.14):

$$\begin{aligned}\frac{\partial}{\partial r} &= \frac{1}{\Delta \eta_N} \frac{\partial}{\partial \eta} & ; \\ \frac{\partial}{\partial \theta} &= \frac{1}{\Delta \xi_E} \frac{\partial}{\partial \xi} - \frac{1}{\Delta \eta_N} \left(\frac{\partial \Delta \eta_S}{\partial \theta} + \eta \frac{\partial \Delta \eta_N}{\partial \theta} \right) \frac{\partial}{\partial \eta} & ;\end{aligned}$$

$$\frac{\partial}{\partial z} = \frac{\partial}{\partial \zeta} - \frac{1}{\Delta \eta_N} \left(\frac{\partial \Delta \eta_S}{\partial z} + \eta \frac{\partial \Delta \eta_N}{\partial z} \right) \frac{\partial}{\partial \eta} ; \quad (2.15)$$

$$\text{where } \Delta \eta_S = r_S ;$$

$$\Delta \eta_N = r_N - r_S ; \quad (2.16)$$

$$\Delta \xi_E = \theta_E - \theta_W .$$

The following further definitions are also required:

$$G = \frac{1}{r} \left\{ \frac{\partial \Delta \eta_S}{\partial \theta} + \eta \frac{\partial \Delta \eta_N}{\partial \theta} \right\} ; \quad (2.17)$$

$$F = \left\{ \frac{\partial \Delta \eta_S}{\partial z} + \eta \frac{\partial \Delta \eta_N}{\partial z} \right\} .$$

G and F are dimensionless quantities and they represent a measure of the degree of non-orthogonality of the particular geometry investigated, in the angular and axial directions, respectively.

(b) Derivation

In this section, the derivation of the partial-differential equations governing the steady, three-dimensional turbulent flow around ships' hulls in the non-orthogonal curvilinear coordinate system is given.

The general form of the ϕ equation in the cylindrical-polar coordinates was expressed (from equation 2.12)

as:

$$\begin{array}{ccc}
 & \text{I} & \\
 \hline
 1 & 2 & 3 \\
 \\
 \frac{1}{r} \frac{\partial}{\partial r} (ru\phi) + \frac{1}{r} \frac{\partial}{\partial \theta} (u\phi) + \frac{\partial}{\partial z} (w\phi) & & \\
 \\
 & \text{II} & \text{III} \\
 \hline
 4 & 5 & 6 \\
 \\
 = \frac{1}{r} \frac{\partial}{\partial r} (\Gamma_\phi r \frac{\partial \phi}{\partial r}) + \frac{1}{r^2} \frac{\partial}{\partial \theta} (\Gamma_\phi \frac{\partial \phi}{\partial \theta}) + \bar{S}_\phi & . & (2.18)
 \end{array}$$

Substitution of transformations (2.15) along with relations (2.16) and (2.17) into (2.18) results in the transformation of the governing equations to the η, ξ, ζ coordinate system as follows:

LHS

$$\begin{array}{l}
 1 \rightarrow \frac{1}{r\Delta\eta_N} \frac{\partial}{\partial \eta} (rv\phi) ; \\
 2 \rightarrow \frac{1}{r\Delta\xi_E\Delta\eta_N} \frac{\partial}{\partial \xi} (u\phi\Delta\eta_N) - \frac{1}{r\Delta\eta_N} \frac{\partial}{\partial \eta} (u\phi Gr) ; \\
 3 \rightarrow \frac{\partial}{\partial \zeta} (w\phi) - \frac{1}{r\Delta\eta_N} \frac{\partial}{\partial \eta} (w\phi Fr) + \frac{w\phi}{r\Delta\eta_N} \frac{\partial (Fr)}{\partial \eta} ;
 \end{array}$$

adding 1, 2, 3 and rearranging, thus

$$\begin{aligned} I &= \frac{1}{r\Delta\eta_N} \frac{\partial}{\partial\eta} \{r\phi(v-uG-wF)\} \quad ; \\ &+ \frac{1}{r\Delta\eta_N\Delta\xi_E} \frac{\partial}{\partial\xi} \{u\phi\Delta\eta_N\} \quad ; \\ &+ \frac{1}{r\Delta\eta_N} \frac{\partial}{\partial\zeta} \{r\Delta\eta_N w\phi\} \quad . \end{aligned}$$

Similarly, for the RHS:

$$\begin{aligned} 4 &\rightarrow \frac{1}{r\Delta\eta_N} \frac{\partial}{\partial\eta} \left(r \frac{\Gamma_\phi}{\Delta\eta_N} \frac{\partial\phi}{\partial\eta} \right) \\ 5 &\rightarrow \frac{1}{r\Delta\eta_N\Delta\xi_E} \frac{\partial}{\partial\xi} \left(\Delta\eta_N \frac{\Gamma_\phi}{r\Delta\xi_E} \frac{\partial\phi}{\partial\zeta} \right) + \frac{1}{r\Delta\eta_N} \frac{\partial}{\partial\eta} \left(G^2 r \frac{\Gamma_\phi}{\Delta\eta_N} \frac{\partial\phi}{\partial\eta} \right) \\ &- \frac{1}{r\Delta\eta_N\Delta\xi_E} \frac{\partial}{\partial\xi} \left(G\Delta\eta_N \frac{\Gamma_\phi}{\Delta\eta_N} \frac{\partial\phi}{\partial\eta} \right) - \frac{1}{r\Delta\eta_N} \frac{\partial}{\partial\eta} \left(Gr \frac{\Gamma_\phi}{r\Delta\xi_E} \frac{\partial\phi}{\partial\zeta} \right) \end{aligned}$$

adding 4, 5 and rearranging*, thus

$$\begin{aligned} II &= \frac{1}{r\Delta\eta_N} \frac{\partial}{\partial\eta} \{ (1+G^2) r \frac{\Gamma_\phi}{\Delta\eta_N} \frac{\partial\phi}{\partial\eta} \} + \frac{1}{r\Delta\eta_N\Delta\xi_E} \frac{\partial}{\partial\xi} \\ &\{ \Delta\eta_N \frac{\Gamma_\phi}{r\Delta\xi_E} \frac{\partial\phi}{\partial\zeta} \} - \frac{1}{r\Delta\eta_N} \frac{\partial}{\partial\eta} \{ Gr \frac{\Gamma_\phi}{r\Delta\xi_E} \frac{\partial\phi}{\partial\zeta} \} \\ &- \frac{1}{r\Delta\eta_N\Delta\xi_E} \frac{\partial}{\partial\xi} \{ G\Delta\eta_N \frac{\Gamma_\phi}{\Delta\eta_N} \frac{\partial\phi}{\partial\eta} \} \end{aligned}$$

* Note that the two diffusion terms of equation (2.18) have become four terms. The last two will be added to the source terms.

The source term III can be done in a similar way.

(c) The general form

The governing equations in the curvilinear coordinate system are summarised in the following general form:

$$\begin{aligned}
 & \frac{1}{r\Delta\eta_N} \frac{\partial}{\partial\eta} \{r\phi(v-uG-wF)\} \\
 & + \frac{1}{r\Delta\eta_N\Delta\xi_E} \frac{\partial}{\partial\xi} \{u\phi\Delta\eta_N\} \\
 & + \frac{1}{r\Delta\eta_N} \frac{\partial}{\partial\zeta} \{r\Delta\eta_N w\phi\} \\
 & = \frac{1}{r\Delta\xi_E\Delta\eta_N} \frac{\partial}{\partial\xi} \left\{ \frac{\Gamma_\phi}{r\Delta\xi_E} \Delta\eta_N \frac{\partial\phi}{\partial\xi} \right\} \\
 & + \frac{1}{r\Delta\eta_N} \frac{\partial}{\partial\eta} \left\{ \frac{\Gamma_\phi}{\Delta\eta_N} r(1+G^2) \frac{\partial\phi}{\partial\eta} \right\} \\
 & + S_\phi \quad ; \quad (2.18)
 \end{aligned}$$

where S_ϕ is the new source term; the full expression of S_ϕ and Γ_ϕ , for various dependent variables are given in Table 3.

The resulting equations in the η, ξ, ζ coordinate sytem are similar to the original equation in the r, θ, z coordinate with additional corrections for convection, shear stress and pressure gradients due to the non-orthogonality. When G and F equal zero, equation (2-18) reduces to the cylindircal-polar form.

Equation	ϕ	Γ	S_ϕ
Continuity	1	-	0
u-Momentum	u	μ_{eff}	$ \begin{aligned} & -\frac{1}{r\Delta\xi_E} \frac{\partial p}{\partial \xi} + \frac{1}{r\Delta\xi_E} \frac{\partial}{\partial \xi} \left(\frac{\mu_{\text{eff}}}{r\Delta\xi_E} \frac{\partial u}{\partial \xi} \right) + \frac{1}{\Delta\eta_N} \frac{\partial}{\partial \eta} \left(\frac{\mu_{\text{eff}}}{r\Delta\xi_E} \frac{\partial v}{\partial \xi} \right) \\ & + \frac{2}{r^2} \frac{1}{\Delta\xi_E} \frac{\partial}{\partial \xi} (\mu_{\text{eff}} v) - \frac{1}{r\Delta\eta_N} \frac{\partial}{\partial \eta} (\mu_{\text{eff}} u) \\ & + \frac{\mu_{\text{eff}}}{r\Delta\eta_N} \frac{\partial u}{\partial \eta} - \frac{\mu_{\text{eff}} u}{r^2} - \frac{uv}{r} + G^2 \left\{ \frac{1}{\Delta\eta_N} \frac{\partial}{\partial \eta} \left(\frac{\mu_{\text{eff}}}{\Delta\eta_N} \frac{\partial u}{\partial \eta} \right) \right\} \\ & - G \left\{ \frac{2}{r\Delta\xi_E} \frac{\partial}{\partial \xi} \left(\frac{\mu_{\text{eff}}}{\Delta\eta_N} \frac{\partial u}{\partial \eta} \right) + \frac{2}{\Delta\eta_N} \frac{\partial}{\partial \eta} \left(\frac{\mu_{\text{eff}}}{r\Delta\xi_E} \frac{\partial u}{\partial \xi} \right) \right. \\ & + \left. \frac{1}{\Delta\eta_N} \frac{\partial}{\partial \eta} \left(\frac{\mu_{\text{eff}}}{\Delta\eta_N} \frac{\partial v}{\partial \eta} \right) \right\} + \frac{\mu_{\text{eff}}}{\Delta\eta_N} \frac{\partial u}{\partial \eta} \left(-\frac{2}{r\Delta\xi_E} \frac{\partial G}{\partial \xi} - \frac{G^2}{r} \right) \\ & - \left\{ \frac{\mu_{\text{eff}}}{\Delta\eta_N} \frac{\partial v}{\partial \eta} \right\} \left(\frac{1}{\Delta\eta_N} \frac{\partial G}{\partial \eta} + \frac{2G}{r} \right) - \frac{2G}{r\Delta\eta_N} \frac{\partial}{\partial \eta} (\mu_{\text{eff}} v) \\ & - \frac{\mu_{\text{eff}}}{r\Delta\xi_E} \frac{\partial u}{\partial \xi} \frac{1}{r\Delta\xi_E \Delta\eta_N} \frac{\partial \Delta\eta_N}{\partial \xi} + \frac{G}{\Delta\eta_N} \frac{\partial p}{\partial \eta} \end{aligned} $
v-Momentum	v	μ_{eff}	$ \begin{aligned} & -\frac{1}{\Delta\eta_N} \frac{\partial p}{\partial \eta} + \frac{1}{\Delta\eta_N} \frac{\partial}{\partial \eta} \left(\frac{\mu_{\text{eff}}}{\Delta\eta_N} \frac{\partial v}{\partial \eta} \right) + \frac{1}{r\Delta\xi_E} \frac{\partial}{\partial \xi} \left(\frac{\mu_{\text{eff}}}{\Delta\eta_N} \frac{\partial u}{\partial \eta} \right) \\ & + \frac{\mu_{\text{eff}}}{r\Delta\eta_N} \frac{\partial v}{\partial \eta} - \frac{2\mu_{\text{eff}}}{r^2 \Delta\xi_E} \frac{\partial u}{\partial \xi} - \frac{1}{r^2 \Delta\xi_E} \frac{\partial}{\partial \xi} (\mu_{\text{eff}} u) - \frac{2\mu_{\text{eff}} v}{r^2} \\ & + \frac{u^2}{r} - G \left\{ \frac{1}{r\Delta\xi_E} \frac{\partial}{\partial \xi} \left(\frac{\mu_{\text{eff}}}{\Delta\eta_N} \frac{\partial v}{\partial \eta} \right) + \frac{1}{\Delta\eta_N} \frac{\partial}{\partial \eta} \left(\frac{\mu_{\text{eff}}}{r\Delta\xi_E} \frac{\partial v}{\partial \xi} \right) \right. \\ & + \left. \frac{1}{\Delta\eta_N} \frac{\partial}{\partial \eta} \left(\frac{\mu_{\text{eff}}}{\Delta\eta_N} \frac{\partial u}{\partial \eta} \right) \right\} - \frac{\mu_{\text{eff}}}{\Delta\eta_N} \frac{\partial v}{\partial \eta} \left[\frac{G^2}{r} \right] - \frac{\mu_{\text{eff}}}{\Delta\eta_N} \frac{\partial v}{\partial \eta} \left(\frac{1}{r\Delta\xi_E} \frac{\partial G}{\partial \xi} \right. \\ & + \left. \frac{G}{\Delta\eta_N} \frac{\partial G}{\partial \eta} \right) + 2 \frac{\mu_{\text{eff}}}{r} \frac{G}{\Delta\eta_N} \frac{\partial u}{\partial \eta} - \frac{\mu_{\text{eff}}}{r\Delta\xi_E} \frac{\partial v}{\partial \xi} \frac{1}{r\Delta\xi_E \Delta\eta_N} \frac{\partial \Delta\eta_N}{\partial \xi} \\ & + \frac{G}{r\Delta\eta_N} \frac{\partial}{\partial \eta} (\mu_{\text{eff}} u) \end{aligned} $

Equation	ϕ	Γ	S_ϕ
w-Momentum	w	μ_{eff}	$- G \left\{ \frac{1}{r\Delta\xi_E} \frac{\partial}{\partial\xi} \left(\frac{\mu_{\text{eff}}}{\Delta\eta_N} \frac{\partial w}{\partial\eta} \right) + \frac{1}{\Delta\eta_N} \frac{\partial}{\partial\eta} \left(\frac{\mu_{\text{eff}}}{r\Delta\xi_E} \frac{\partial w}{\partial\xi} \right) \right\}$ $- \frac{\mu_{\text{eff}}}{r\Delta\xi_E} \frac{\partial w}{\partial\xi} \frac{1}{r\Delta\xi_E \Delta\eta_N} \frac{\partial\Delta\eta_N}{\partial\xi} - \left(\frac{\mu_{\text{eff}}}{\Delta\eta_N} \frac{\partial w}{\partial\eta} \right) \left(\frac{G^2}{r} + \frac{1}{r\Delta\xi_E} \frac{\partial G}{\partial\xi} \right)$ $+ \frac{G}{\Delta\eta_N} \frac{\partial G}{\partial\eta} \Big) - \frac{\partial p}{\partial\zeta} + \frac{F}{\Delta\eta_N} \frac{\partial p}{\partial\eta}$
Kinetic energy	k	$\frac{\mu_{\text{eff}}}{\sigma_{\text{eff},k}}$	$- G \left[\frac{1}{\Delta\eta_N} \frac{\partial}{\partial\eta} \left(\frac{\Gamma_k}{r\Delta\xi_E} \frac{\partial k}{\partial\xi} \right) + \frac{1}{r\Delta\xi_E} \frac{\partial}{\partial\xi} \left(\frac{\Gamma_k}{\Delta\eta_N} \frac{\partial k}{\partial\eta} \right) \right]$ $- \frac{1}{r\Delta\xi_E \Delta\eta_N} \frac{\partial\Delta\eta_N}{\partial\xi} \frac{\Gamma_k}{r\Delta\xi_E} \frac{\partial k}{\partial\xi} - \left(\frac{1}{r\Delta\xi_E} \frac{\partial G}{\partial\xi} + \frac{G}{\Delta\eta_N} \frac{\partial G}{\partial\eta} \right) \cdot$ $\frac{\Gamma_k}{\Delta\eta_N} \frac{\partial k}{\partial\eta} - \frac{G^2}{r} \frac{\Gamma_k}{\Delta\eta_N} \frac{\partial k}{\partial\eta} + GE - \epsilon$
Dissipation rate	ϵ	$\frac{\mu_{\text{eff}}}{\sigma_{\text{eff},\epsilon}}$	$- G \left[\frac{1}{\Delta\eta_N} \frac{\partial}{\partial\eta} \left(\frac{\Gamma_\epsilon}{r\Delta\xi_E} \frac{\partial\epsilon}{\partial\xi} \right) + \frac{1}{r\Delta\xi_E} \frac{\partial}{\partial\xi} \left(\frac{\Gamma_\epsilon}{\Delta\eta_N} \frac{\partial\epsilon}{\partial\eta} \right) \right]$ $- \frac{1}{r\Delta\xi_E \Delta\eta_N} \frac{\partial\Delta\eta_N}{\partial\xi} \frac{\Gamma_\epsilon}{r\Delta\xi_E} \frac{\partial\epsilon}{\partial\xi} - \left(\frac{1}{r\Delta\xi_E} \frac{\partial G}{\partial\xi} \right.$ $\left. + \frac{G}{\Delta\eta_N} \frac{\partial G}{\partial\eta} \right) \frac{\Gamma_\epsilon}{\Delta\eta_N} \frac{\partial\epsilon}{\partial\eta} - \frac{G^2}{r} \frac{\Gamma_\epsilon}{\Delta\eta_N} \frac{\partial\epsilon}{\partial\eta}$ $+ C_1 GE \frac{\epsilon}{k} - C_2 \frac{\epsilon^2}{k}$

TABLE 3 - THE DIFFUSION COEFFICIENTS AND THE SOURCE TERMS OF EQUATION (2.18)

2.3 Turbulence Modelling

2.3.1 Approach

When the flow is turbulent, the velocity components and pressure fluctuate in some nearly random fashion with respect to space and time around the appropriately defined mean values. Theoretically speaking, it should be possible to predict turbulent flow directly by solving the same set of equations as those of the corresponding unsteady laminar flow. However, in practice, such a solution is an impossibility because the equations are partial, non-linear and simultaneous. Also, in fully three-dimensional flows, some of the important turbulence processes are taking place at very small spatial distances and in small time steps (see for example, Hinze (1959), Tennekes and Luley (1973)).

Thus, a numerical solution of the exact time dependent equations for even the simplest turbulent flows would still require prohibitive computational storage and time.

Engineers, designers and researchers, are normally interested in only the time-averaged properties of turbulence, and not the details of turbulence. Therefore, the present approach in calculating turbulent

flows is to solve the equations which govern the transport of mean flow variables; these equations are obtained by time-averaging the transport equations for the instantaneous values of the flow variables.

Mathematically, this means expressing the general variable ϕ as:

$$\phi = \bar{\phi} + \phi' \quad ; \quad (2.19)$$

where $\bar{\phi}$ is the time-averaged value and ϕ' is the fluctuating component. The process of time-averaging, however, introduces statistical correlations involving the fluctuating velocities and scalar quantities of which magnitudes are unknown (such as $\overline{u'v'}$, $\overline{v'w'}$, ... etc). We must at this stage, approximate these correlations in terms of quantities we can determine. This process is known as "turbulence modelling".

2.3.2 The k- ϵ turbulence model

Research into the mathematical modelling of turbulence has, in recent years, led to the use of multi-equation turbulence models. Several models were developed by researchers, a survey of such models can be found in (Launder & Spalding, 1972). Of these, the most popular has been the k- ϵ model developed initially by

Harlow and Nakayama, 1968. The assumption is made that the turbulence is adequately characterised by two quantities: turbulence energy k , and its rate of dissipation, ϵ .

In this turbulence model the turbulence fluxes are related to the gradients of the flow properties through a viscosity which, at any location in the flow, is assumed to be isotropic. The distribution of this viscosity over the flow domain is calculated from the values of k and ϵ through the following formula:

$$\mu_t = \rho c_\mu \frac{k^2}{\epsilon} \quad ; \quad (2.20)$$

where μ_t is the turbulence viscosity and c_μ is a constant.

The model used in the present investigation is a modification to that of Harlow and Nakayama, 1968. The merits of the model are as described in (Launder and Spalding, 1973).

The k - ϵ has and is still enjoying a fair degree of success in the prediction of turbulent flows. Among the diverse flow situations investigated in recent years using the k - ϵ model are the following: free turbulent flows (Launder et al, 1972), flow in straight

and curved ducts (Sharma, 1974), (Tatchell, 1975) and (Pratap and Spalding, 1975), turbulent buoyant jets (McGuirk, 1975), three-dimensional can combustor (Serag-El-Din, 1977), turbulent buoyant flow in pipes (Abdelmeguid and Spalding, 1979), and unsteady turbulent heat transfer in pipes (El-Hadidy, 1980). In general, the performance of the model is satisfactory, and its choice for the present investigation is aided by the diversity of its applications.

In this model, the partial-differential equations governing the transport of turbulent kinetic energy k , and its dissipation rate ϵ , are:

- Kinetic energy of turbulence

$$\begin{aligned} & \frac{1}{r} \frac{\partial}{\partial \theta} (uk) + \frac{1}{r} \frac{\partial}{\partial r} (rvk) + \frac{\partial}{\partial z} (wk) \\ &= \frac{1}{r} \frac{\partial}{\partial r} (r\Gamma_k \frac{\partial k}{\partial r}) + \frac{1}{r^2} \frac{\partial}{\partial \theta} (\Gamma_k \frac{\partial k}{\partial \theta}) + GE - \epsilon. \end{aligned} \quad (2.21)$$

- Dissipation rate

$$\begin{aligned} & \frac{1}{r} \frac{\partial}{\partial \theta} (u\epsilon) + \frac{1}{r} \frac{\partial}{\partial r} (rv\epsilon) + \frac{\partial}{\partial z} (w\epsilon) \\ &= \frac{1}{r} \frac{\partial}{\partial r} (r\Gamma_\epsilon \frac{\partial \epsilon}{\partial r}) + \frac{1}{r^2} \frac{\partial}{\partial \theta} (\Gamma_\epsilon \frac{\partial \epsilon}{\partial \theta}) \\ &+ C_1 GE \frac{\epsilon}{k} - C_2 \frac{\epsilon^2}{k} ; \end{aligned} \quad (2.22)$$

where GE represents the general of turbulence energy, and is defined as:

$$GE = \mu_t \left[2 \left\{ \left(\frac{1}{r} \frac{\partial u}{\partial \theta} + \frac{v}{r} \right)^2 + \left(\frac{\partial v}{\partial r} \right)^2 \right\} + \left\{ r \frac{\partial}{\partial r} \left(\frac{u}{r} \right) + \frac{1}{r} \frac{\partial v}{\partial \theta} \right\}^2 + \left\{ \left(\frac{1}{r} \frac{\partial w}{\partial \theta} \right)^2 + \left(\frac{\partial w}{\partial r} \right)^2 \right\} \right]. \quad (2.23)$$

Γ_k and Γ_ϵ represent the effective diffusion coefficients for k and ϵ respectively; and C_1 and C_2 are constants in the turbulence model.

The value of the effective diffusion coefficient, Γ , may be expressed by the following relation:

$$\Gamma_\phi = \frac{\mu_\ell}{Pr_{\ell\phi}} + \frac{\mu_t}{Pr_{t\phi}} \quad ; \quad (2.24)$$

where the subscript ϕ denotes the values for the variable ϕ ; μ_ℓ and μ_t are the laminar and turbulent viscosity; and Pr_ℓ and Pr_t represent the laminar and turbulent Prandtl/Schmidt number.

The model contains five empirical constants which are assigned the following values:-

$$C_1 = 1.44; C_2 = 1.92; C_\mu = 0.09; \\ Pr_k = 1.; \text{ and } Pr_\epsilon = 1.23 \quad . \quad (2.25)$$

The value of Pr_λ is dependent on the molecular properties of the fluid, and is equal to unity for the velocities and for k and ϵ .

The length scale of turbulence, in this model, is given by:

$$\lambda = c_\mu \frac{k^{3/2}}{\epsilon} \quad (2.26)$$

2.3.3 The near-wall treatment

Turbulence flows near a wall are distinguished in two ways from the flow away from the wall:

(a) The flow properties often show steep variations in the vicinity of the wall, thus necessitating the use of an extremely fine grid for their accurate computation. Such rigorous incorporation of the effects of the vicinity of a wall to turbulence proves expensive in computer time.

(b) The effect of molecular viscosity becomes prominent because of the damping effect of wall turbulence. The present model is, however, valid only for fully turbulent flow, and alternative modelling would be required. The use of a low-Reynolds number turbulence model (e.g. Jones and Launder, 1973) is a possible solution; however, to obtain good accuracy,

it is necessary to employ a very fine grid in the vicinity of the wall, and hence devote substantial computational effort to this narrow region.

One economical method of accounting for these effects is by way of "wall functions", (Patankar and Spalding, 1972). Wall functions have been proved to be satisfactory in two-dimensional situations. It is these wall functions, with their appropriate extension to three-dimensional cases that have been employed in the present study.

The features of the method are as follows. The velocity variation in the near wall region is presumed to obey the logarithmic law given below:

$$\frac{U_P}{U_\tau} = \frac{1}{\kappa} \ln \left\{ \frac{E\delta U_\tau}{\nu} \right\} \quad ; \quad (2.27)$$

where U_P is the resultant velocity component parallel to the wall shear stress;

U_τ is the shear velocity;

δ is the normal distance from the wall;

ν is the laminar diffusivity;

κ is the Von Karman's constant appearing in the logarithmic law of the wall, ($\kappa=0.4$); and

E is the constant function of wall roughness, ($E=9.0$ for smooth walls).

The shear velocity, U_τ , in the model, is defined as:

$$U_\tau = c_\mu^{\frac{1}{4}} k_p^{\frac{1}{2}} ; \quad (2.28)$$

where k_p represents the value of turbulence energy in the near wall region, and c_μ is the constant (=0.09) in the turbulence model.

The shear stress at the wall is defined as:

$$\tau_w = \rho c_\mu^{\frac{1}{2}} k_p ; \quad (2.29)$$

where symbols are as defined before.

The rate of dissipation of kinetic energy, ϵ_p , near the wall is fixed by reference to the requirement that the length scale varies linearly with distance from the wall; the corresponding expression for ϵ_p is then given by:

$$\epsilon_p = \frac{U_\tau^3}{\kappa \delta} . \quad (2.30)$$

The quantity k_p is calculated from the regular balance equation but with the following changes: the generation term in the kinetic energy equation is then modified to account for the values of wall shear-stress calculated from (2.29).

2.4 Auxiliary Information

In addition to the set of equations governing the flow, represented by equations (2.18) and (2.22), the complete specification of the mathematical problem also requires the knowledge of some auxiliary relationships (e.g. (2.13), (2.17)...), and the appropriate boundary and initial conditions. These conditions are wall conditions, symmetry-plane conditions, free-surface conditions, free boundary conditions, initial conditions, and exit conditions.

(a) Wall conditions (the hull boundary)

The velocities at the hull are zero from the no-slip conditions. For k , a zero diffusive flux at the hull is used. For ϵ , the empirical evidence that a typical length scale of turbulence such as ℓ varies linearly with the distance from the wall (see Section 2.3.3) is used to calculate ϵ itself at the near wall point.

(b) Symmetry-plane conditions

The east boundary of the calculation domain is a symmetry plane. Downstream of the stern, the boundary which was previously the hull (the south boundary) becomes a symmetry plane as well.

The symmetry-planes are of course zero-gradient boundaries, since there is no flux of any quantity through them.

(c) Free-surface conditions (water level)

The west boundary of the calculation domain (the free-surface) has been treated as symmetry plane also (i.e. this is the case of zero Froude number flow). Such a treatment is an approximation justified by the ignorance of the true conditions here.

(d) Free-boundary conditions

The conditions at the free boundary of the calculation domain (north boundary) is also unknown. If the free-boundary is located far away from the ship's hull, then the conditions prevailing there would be the same as the undisturbed flow; nearer to the hull, the conditions would be those satisfying an inviscid flow solution. Therefore, the flow variables (u and w) at this boundary are given the values calculated from the potential flow solution around the hull. The v -velocity is computed from the local continuity, and P is thereafter calculated to cause P_0 (the stagnation pressure) to be equal to the undisturbed one. A zero gradient condition is applied for P , k and ϵ .

(e) Initial conditions

Those are the values of the dependent variables corresponding to the starting position of calculation along the predominant flow direction (i.e. ζ). The calculation starts at midship and the conditions of the dependent variables there are either known from measurements or required of some specification.

In the absence of any knowledge of these conditions, particular assumptions can be put forth to specify some conditions.

At the starting station, the specification of five dependent variables, u , v , w , k and ϵ are required. For long ships, one can assumed, without much error, that the flow characteristics are the same as the fully developed flow over a flat plate. Such an assumption is justified by assuming that the section of ship with constant geometry (i.e. at, and around midship), the boundary layer gets developed in a manner similar to its development over a flat plate. This assumption stems out the following initial conditions:

- For u and v : equal to zero, everywhere.
- For w : it is assumed that, at the inlet section, the boundary layer is uniform around the girth of the hull and corresponds to a girth along

a flat plate of length z_{len} . Thus w is calculated from:

$$\frac{w}{U_{\infty}} = \left(\frac{y}{\delta}\right)^{1/7}, \quad (2.32)$$

where y denotes distance from the hull, $\delta = \delta(z)$ denotes the boundary layer thickness at distance z_{len} from the plate's leading edge; and U_{∞} is the free-stream velocity. The flat plate 1/7th power-law profile is then scaled to produce a velocity at the free boundary which is the same as the potential-flow solution. δ is calculated from the following relations (Schlichting, 1960):

$$\delta = .37 z_{len} \left(\frac{U_{\infty} z_{len}}{\nu} \right)^{1/5}. \quad (2.33)$$

- For k : uniform and equal to 0.4% of the free-stream velocity squared.
- For ϵ : the Escudier formula (Escudier, 1966) is used to calculate the distribution of the length scale, l_m .

$$\begin{aligned} \text{For } \frac{y}{\delta} \leq \frac{c_{\mu}}{\kappa} \quad , \quad l_m &= \kappa y ; \\ \frac{y}{\delta} > \frac{c_{\mu}}{\kappa} \quad , \quad l_m &= c_{\mu} \delta ; \end{aligned} \quad (2.34)$$

where κ is the von Karman constant (≈ 0.4) and ℓ_m is equal to $\ell c_\mu^{-1/4}$. The starting value of ϵ are now calculated from:

$$\epsilon = \frac{c_\mu^{3/4} \kappa^{3/2}}{\ell_m} \quad (2.35)$$

(f) The exit boundary conditions

At the exit of the calculation domain (nearly 1/5th of a ship length downstream of the stern), an additional boundary condition is needed for the pressure. It is assumed that the exit pressure is uniform, at the value given by the potential-flow solution.

2.5 Closure

In summary, the mathematical framework of the present work has been presented in this chapter. The partial-differential equations that govern the transport of mass, momentum... etc, in three-dimensional boundary-layer flows have been stated in their cylindrical-polar form.

These equations are then transformed into a novel curvilinear non-orthogonal coordinate system which is more convenient than the cylindrical-polar one. The details of the turbulence model employed in the present calculations have been also described.

Closure to the mathematical problem is achieved by the auxiliary information. It is left to seek a solution procedure for the mathematical model. Such a procedure will be described and outlined in the following chapter.

CHAPTER 3 - THE SOLUTION PROCEDURE

3.1 Introduction

In the previous chapter, the partial-differential equations which govern the three-dimensional flows considered in this thesis were set out, along with the accompanying auxiliary relations. The purpose of this chapter is to provide the details of the solution procedure which has been employed in solving these equations.

The first step in the development of the solution procedure is to obtain discretised equivalents of the partial-differential equations (i.e. finite-difference equations). The entire flow field of interest is, therefore, covered by a finite-difference grid. Owing to the irregular geometry of the flow domain, a non-orthogonal grid, based on the non-orthogonal coordinate system developed in Chapter 2 was employed. Each grid point is surrounded by a control volume. The finite-difference equations are thus obtained by integration of the differential equations over the respective control volumes. The resulting finite-difference equations link the flow variables at a particular node with the surrounding ones, and are non-linear and strongly coupled.

The next step is to employ a calculation procedure to solve these equations. The calculation procedure described here is based on the method described earlier by Patankar and Spalding, (1972a), and with its extension for partially-parabolic flows as reported by Spalding (1975), Patankar et al (1975) and Pratap and Spalding (1976).

The solution procedure, therefore, comprises of the following main stages:

- (a) Subdivision of the flow domain into smaller regions by a finite-difference grid, and integration of the differential equations over the finite-difference "cells"; and
- (b) solution of the algebraic equations obtained from the integration over step (a).

A detailed description of each of these stages is presented below.

3.2 The Finite-Difference Scheme

The first step in the development of the solution procedure is to obtain discretised equivalents for the partial-differential equations governing the flow. This is described in the following sub-sections.

3.2.1 The finite-difference grid

The finite-difference grid consists of a number of intersecting grid lines arranged over the three-dimensional flow domain. The arrangement of the finite-difference grid system used is indicated in Figure 3.2.1a and Figure 3.2.1b. Figure 3.2.1a shows the family of grid lines in the cross-stream plane, whereas Figure 3.2.1b shows the family of grid lines in the longitudinal direction. The intersection of these lines, known as 'grid-nodes', serve as reference location for the identification of the discrete values of the flow variables. Since the choice of the grid affects accuracy and economy, the grid spacings need not necessarily be uniform over the flow domain, but can vary so that there is a high concentration of grid nodes in regions of steep variations of flow properties. The same freedom applies in choosing the increment in the ζ -direction (or forward step). The only limits placed on this choice are those of accuracy, stability and economy. The finite-difference grid is chosen after experimentation with finer and coarser grids so as to make the results of the computation substantially independent of the grid employed.

3.2.2 The locations of variables

Figure 3.2.2 shows the manner in which the flow variables

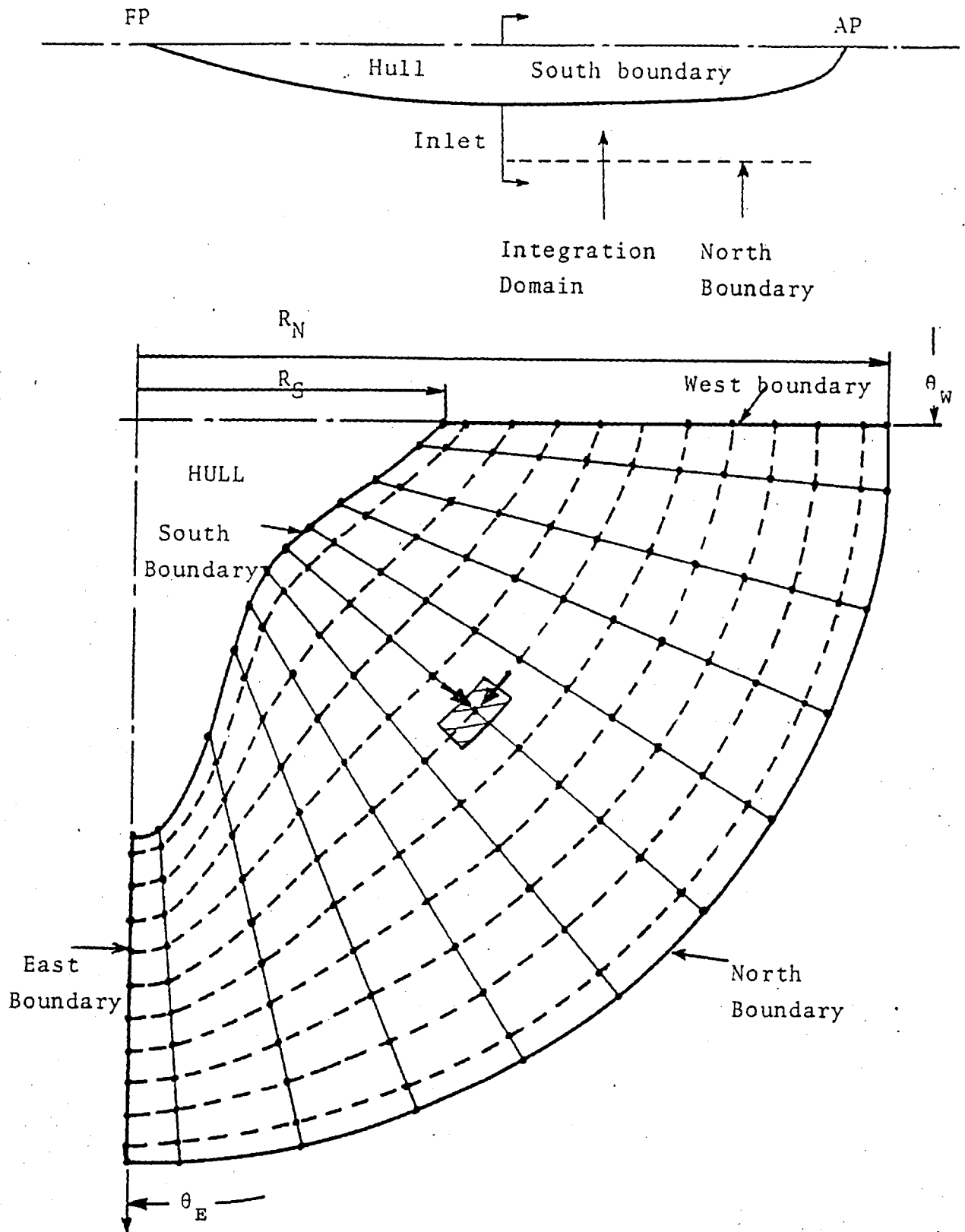


Figure 3.2.1.a The Grid (cross-stream)

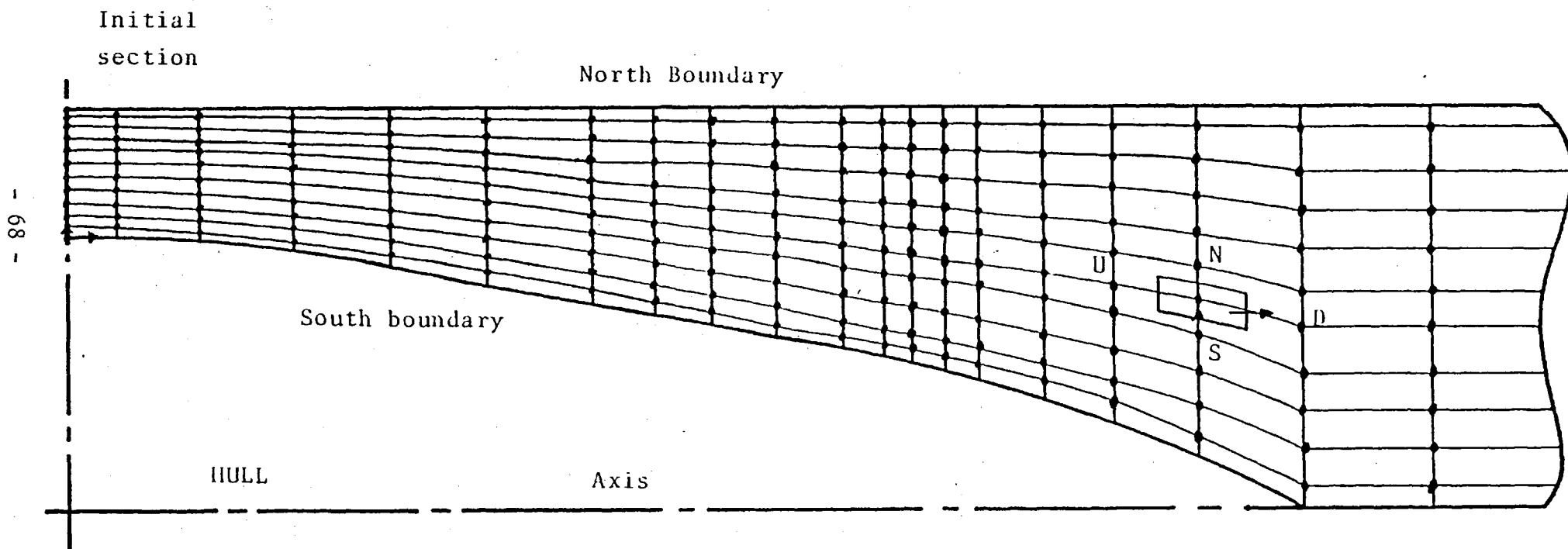


Figure 3.2.1.b The Grid (Longitudinal direction)

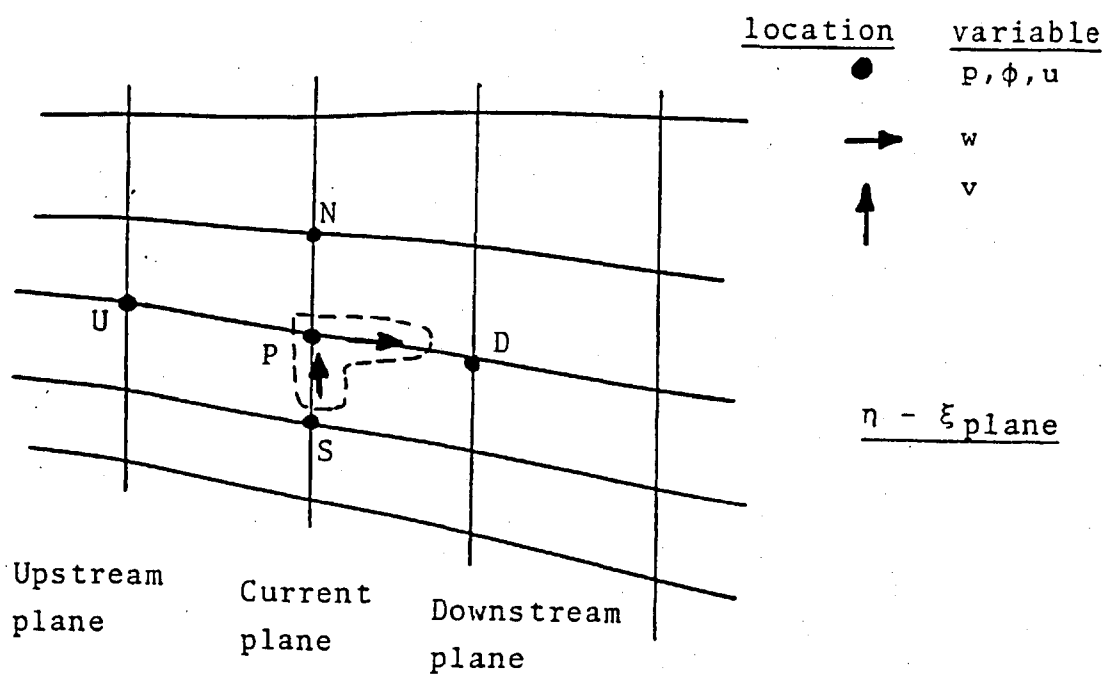
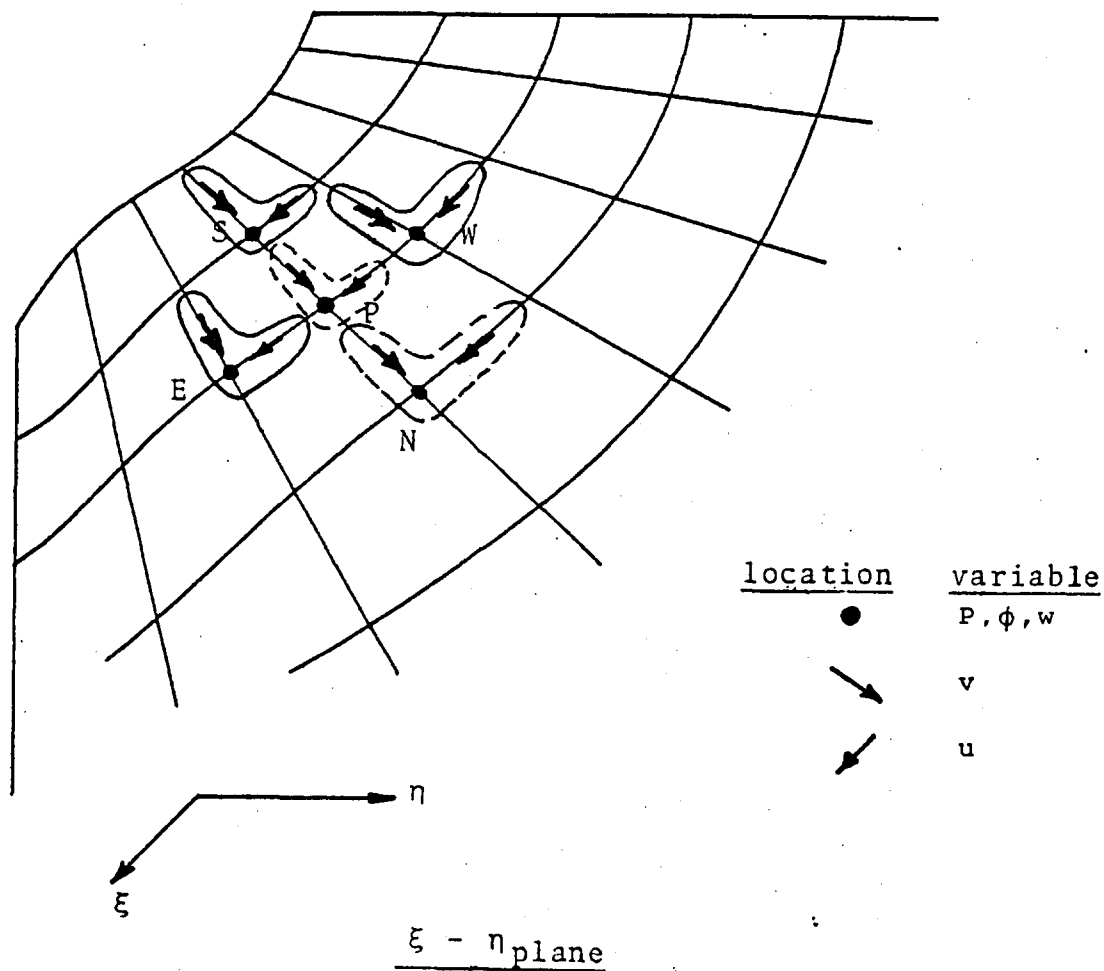


Figure 3.2.2 Location of variables on finite-difference grid

are located in the finite-difference grid. The pressure, and all scalar properties such as turbulence kinetic energy, its dissipation etc, are stored at the grid nodes, while the velocity-components are placed midway between adjacent grid nodes. The u and v velocities are stored midway behind the grid node, while the w velocities are stored midway ahead of the grid node. The "staggered-grid" system adopted here has the following advantages:

- (a) the velocity components are conveniently located for calculating the convective fluxes of flow properties stored at the grid node;
- (b) the calculation of mass balance over a region surrounding a grid node is made easy because the velocities normal to the boundaries of this region are located directly on the boundaries;
- (c) the pressures are stored so as to make it easy to calculate the pressure gradients that affect the velocity components.

The dashed enclosures shown in the figure describe the manner in which the variables have been grouped for the purpose of identification: the variables enclosed by a given dashed line are denoted by the same subscript.

3.2.3 The control volumes

The "control volumes" represent small arbitrary regions surrounding each node, over which the differential equations are integrated in order to obtain their finite-difference counterpart. The control volumes are different for each flow variables. Figure 3.2.3 shows the control volumes for the velocity components and for a general variable, ϕ . The boundaries of these control volumes are defined as follows. For variables located at the grid nodes, the boundaries of the control volumes, in all three directions, lie half-way between the grid nodes; thus the interfaces of the control volumes always pass through locations at which the normal-velocities are stored. The control volumes for velocity components are defined in a similar way, but the above practice is adopted only along those directions which are perpendicular to the velocity; for the direction along the velocity, the edge of the control volume pass through the grid nodes which lie on either side of the velocity. Figure 3.2.4 shows the main control volume for the pressure, and other ϕ variables.

3.2.4 Location of flow variables near the boundary

The staggered grid system explained above for locating the velocity components presents a few difficulties in

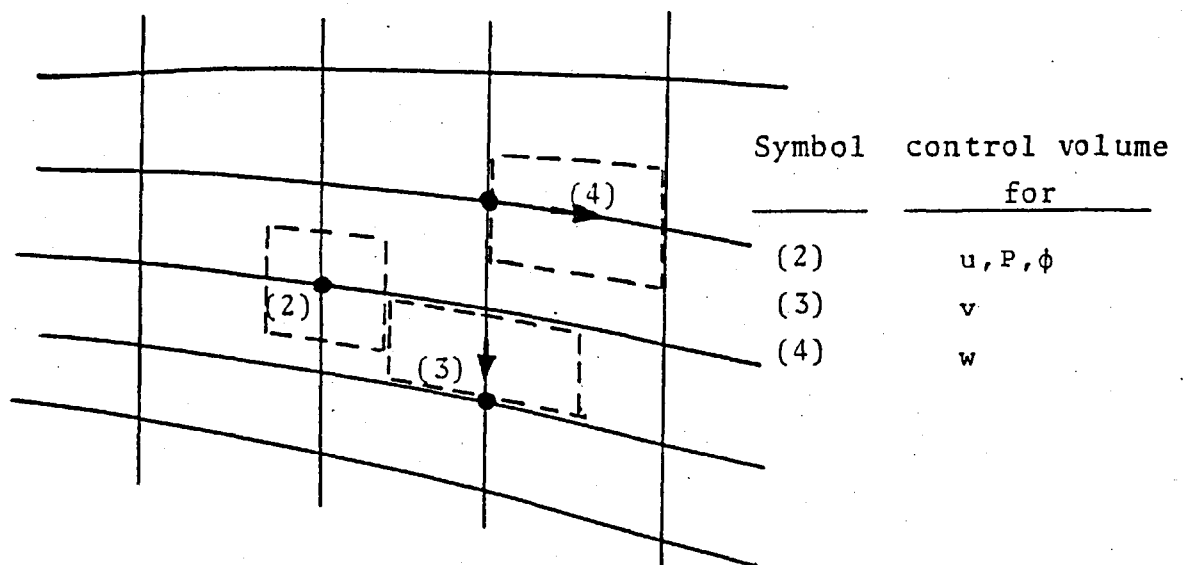
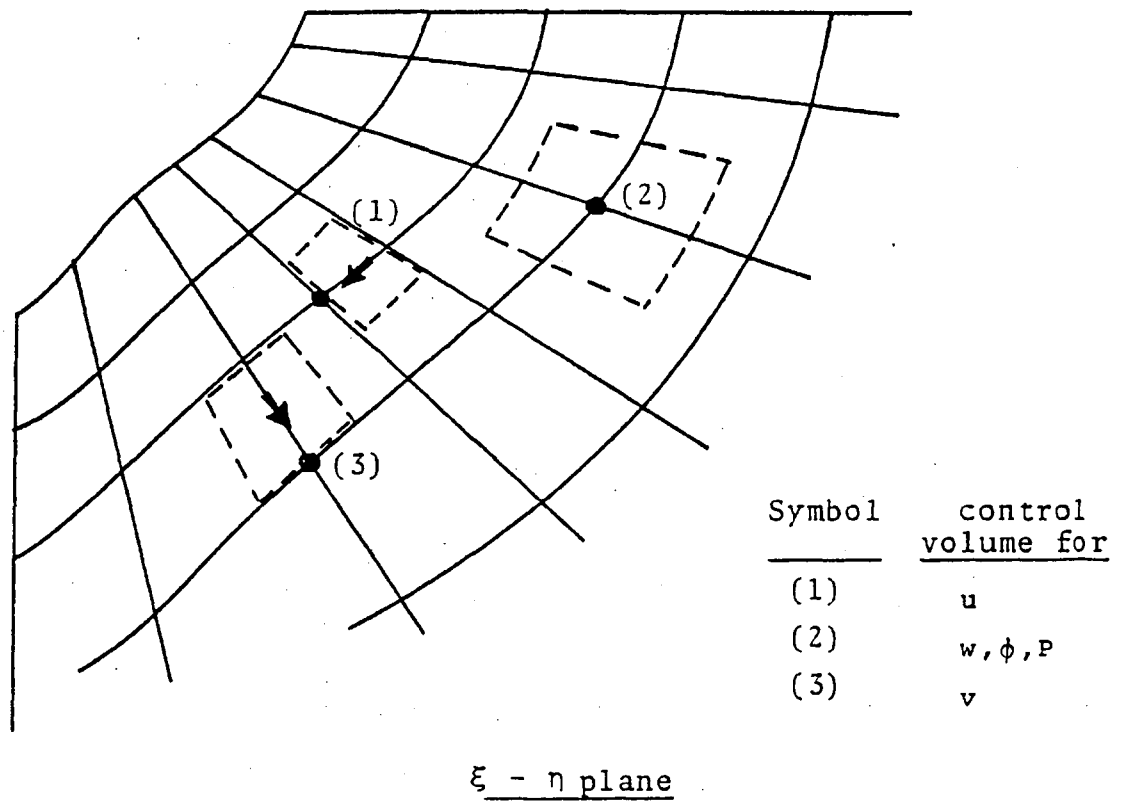


Figure 3.2.3 Definition of control volumes

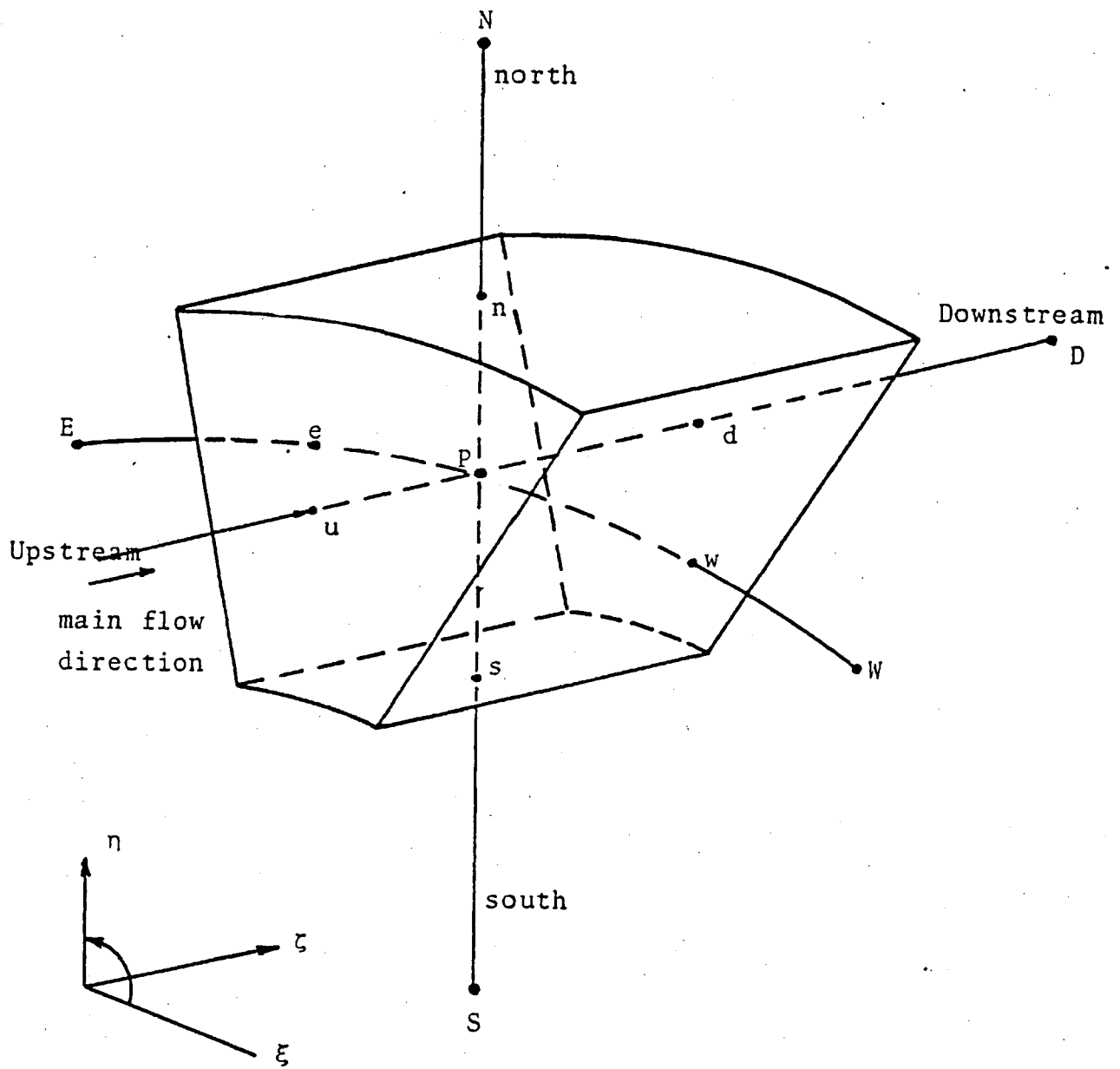


Figure 3.2.4 The main control volume

calculating the flow variables situated adjacent to the boundaries of the flow domain. In order to overcome these difficulties, slightly different practices are employed for locating the near-boundary velocities. Figure 3.2.5 illustrates the locations of the velocities near the boundaries. The new practice consists of altering the locations of the velocities normal to the boundaries from their staggered positions to locations directly on the boundary. The new locations of the near-boundary velocities also modify the control volumes for the variables situated near the boundary; those changes are shown in Figure 3.2.5

3.2.5 Integration of the differential equations

To obtain the finite-difference form of the set of the partial-differential equations which govern the flow motion, each of these equations is integrated for each variable over the appropriate control volumes surrounding each node. Since the transport equations have a generalised form (see Section 2.2.3), the procedures for their integration are identical; thus the steps in the integration procedure are explained below with reference to the differential equation for general variable ϕ .

Basic decisions

In the integration, the following decisions are taken regarding the manner in which the variables varies between grid nodes.

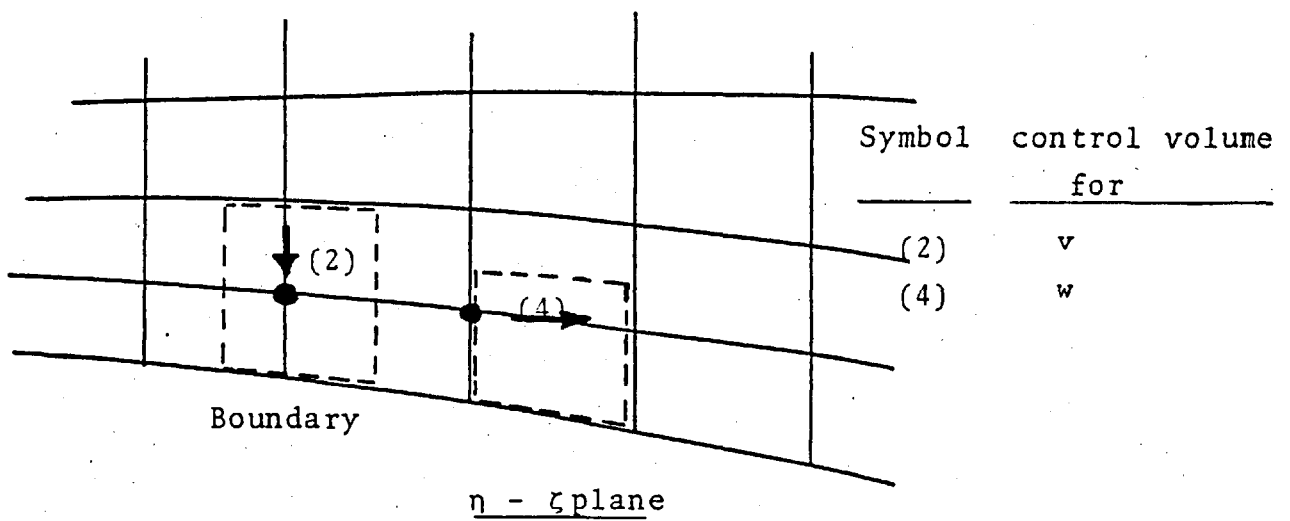
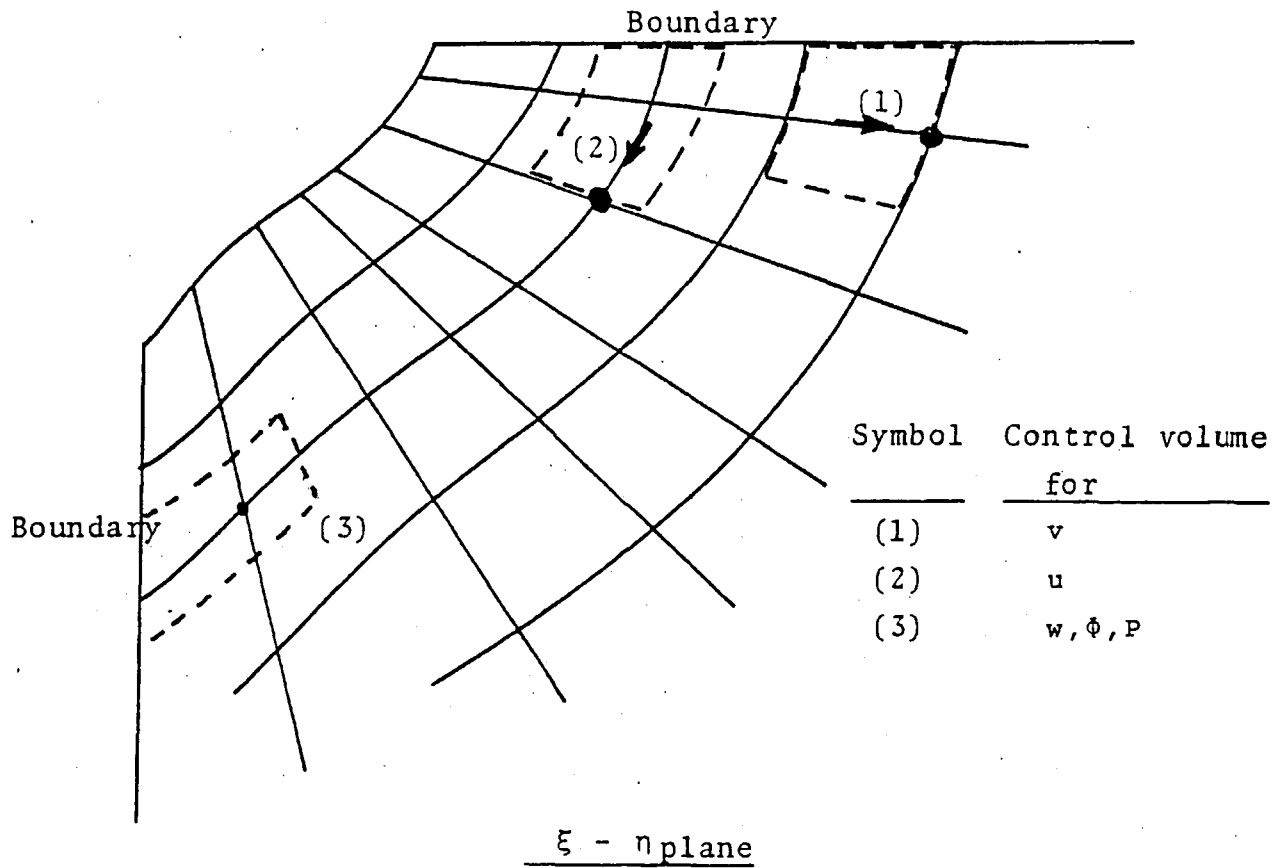


Figure 3.2.5 Near boundary control volumes

- (i) In the longitudinal direction (ζ), ϕ varies in a step wise manner; the current plane (at $\zeta=\zeta_D$) values of ϕ prevails over the interval ζ_P to ζ_D , except at $\zeta=\zeta_D$ (see Figure 3.2.2).
- (ii) For the calculation of convection of ϕ in the longitudinal (ζ) direction, the variation of ϕ in the cross-stream (ξ, η) plane is also taken to be step-wise. Thus, in the $\xi-\eta$ plane the value of ϕ is assumed to remain uniform and equal to ϕ_P over the control volume surrounding the point P, and to change sharply to $\phi_N, \phi_S, \phi_E, \phi_W$ or ϕ_U at the edges of the control volume.
- (iii) For the cross-stream convection from the $\xi-\zeta$ and $\eta-\zeta$ faces of the control volume, an upwind difference scheme is employed. According to this, the value of ϕ convected across the face is the value at the neighbouring node from which the flow is approaching towards the face concerned. Details of implementation of this practice will be presented in Section 3.2.6.
- (iv) For cross-stream diffusion from the $\xi-\zeta$ and $\eta-\zeta$ faces of the control volume, the value of ϕ is assumed to vary linearly between grid nodes; this implies that for an interface which is midway between two grid nodes, the value of ϕ that is diffused across the interface is the arithmetic mean of the two values of ϕ at the grid nodes.

The integrated equation for ϕ

From equation (2.18) the general form of the ϕ equation is:

$$\begin{aligned} & \frac{1}{r\Delta\xi_E\Delta\eta_N} \frac{\partial}{\partial\xi} \{\Delta\eta_N u\phi\} + \frac{1}{r\Delta\eta_N} \frac{\partial}{\partial\eta} \{r(v-uG-wF)\phi\} \\ & + \frac{1}{r\Delta\eta_N} \frac{\partial}{\partial\zeta} \{r\Delta\eta_N w\phi\} = \frac{1}{r\Delta\xi_E\Delta\eta_N} \frac{\partial}{\partial\xi} \left\{ \frac{\Gamma\phi}{r\Delta\xi_E} \Delta\eta_N \frac{\partial\phi}{\partial\xi} \right\} \\ & + \frac{1}{r\Delta\eta_N} \frac{\partial}{\partial\eta} \left\{ \frac{\Gamma\phi}{\Delta\eta_N} r(1+G^2) \frac{\partial\phi}{\partial\eta} \right\} + S_\phi . \end{aligned} \quad (3.1)$$

Equation (3.1) is now integrated over the control volume for the ϕ variable appropriate to the arbitrary grid node P. The control volume face in the cross-stream plane is shown in Figure 3.2.6. The notation in the figure is self-explanatory. Note that while the captions N, E, etc refer to the nodal points, the small letters s, n, e, etc denote cell interfaces. The volume of the cell is given by:

$$\Delta V = r\Delta\xi_E \Delta\eta_N \delta\xi\delta\eta\delta\zeta . \quad (3.2)$$

Bearing in mind the decisions taken earlier, (except (iii), which will be implemented in the next section), the integrated equation may be expressed as an algebraic equation for each grid location, in the

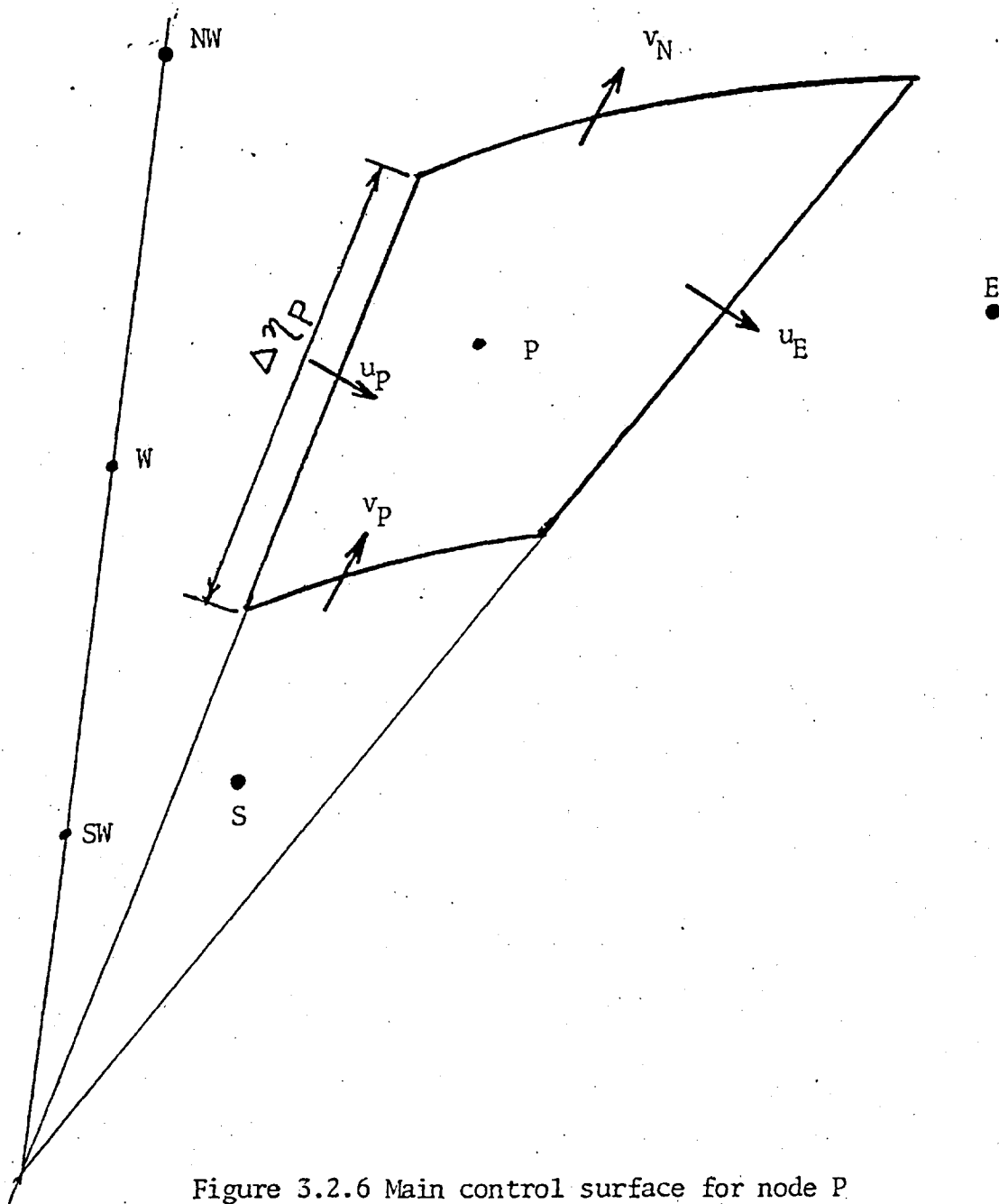


Figure 3.2.6 Main control surface for node P

following form:

$$\begin{aligned}
 & F_P^{\zeta} \phi_P - F_{PU}^{\zeta} \phi_{PU} + L_N^{\eta} \phi_N - L_S^{\eta} \phi_S \\
 & + L_E^{\xi} \phi_E - L_W^{\xi} \phi_W \\
 & = T_N^{\eta} (\phi_N - \phi_P) - T_S^{\eta} (\phi_P - \phi_S) \\
 & + T_E^{\xi} (\phi_E - \phi_P) - T_W^{\xi} (\phi_P - \phi_W) \\
 & + S_{\phi}^U + S_{\phi}^P \phi_P \quad . \quad (3.3)
 \end{aligned}$$

In the above equation, the symbols L, T and F are defined as follows:

$$F_P^{\zeta} = \{w\}_P r \delta \eta \delta \xi \Delta \eta_P \Delta \xi_P \delta \zeta \quad ;$$

$$F_{PU}^{\zeta} = \{w\}_{PU} r \delta \eta \delta \xi \Delta \eta_P \Delta \xi_P \delta \zeta \quad ;$$

$$L_J^{\eta} = \{v-wF-uG\}_J (r \delta \xi)_J \delta \zeta \Delta \xi_E \quad ;$$

$$L_I^{\xi} = \{u\}_I (\delta \eta \Delta \eta)_I \delta \zeta \quad ;$$

$$T_J^{\eta} = \left\{ \Gamma_{\phi} \frac{(1+G^2)}{\delta \eta \Delta \eta} r \delta \xi \right\}_J \delta \zeta \Delta \xi_E \quad ;$$

$$T_I^{\eta} = \left\{ \Gamma_{\phi} \frac{\delta \eta \Delta \eta}{r \delta \xi} \right\}_I \frac{\delta \zeta}{\Delta \xi_E} \quad ;$$

$$\begin{aligned}
 S_{\phi}^P &= S_{\phi}^P r \delta \xi \delta \eta \delta \zeta \Delta \xi_E \Delta \eta_P ; \\
 S_{\phi}^U &= S_{\phi}^U r \delta \xi \delta \eta \delta \zeta \Delta \xi_E \Delta \eta_P ; \quad (3.4)
 \end{aligned}$$

where subscripts E, W, N, S, P and U, and the subscripts e, w, n, s refer to the corresponding nodes and locations as shown in Figure 3.2.6; and $\Delta \xi_E \delta \xi$, $\Delta \eta_N \delta \eta$, $\delta \zeta$ are defined as the dimensions of the control volume; $\delta \xi$ and $\delta \eta$ are dimensionless quantities. The dimension $\Delta \eta_N$ is the height of the flow domain at a particular ξ line. The expression for F_P may be obtained by integrating the continuity equation and is as follows:

$$F_P^{\zeta} = F_{PU}^{\zeta} - L_n^{\eta} + L_s^{\eta} - L_e^{\xi} + L_w^{\xi} ; \quad (3.5)$$

or it can be obtained by putting $\phi=1$ in equation (3.4).

S^{ϕ} is the integrated form of the source (and/or sink) of ϕ ; although sources may be non-linear functions of ϕ , in general, they are always expressed in the following linearised form for reasons of stability:

$$S^{\phi} = S_U^{\phi} + S_P^{\phi} \phi_P . \quad (3.6)$$

S_U^{ϕ} and S_P^{ϕ} are volume average values of the source terms and are calculated from values already stored

in the computer at the time they are needed. Expressing source terms in the linearised form enhances the stability of the finite-difference equations, and is recommended as a useful practice to be applied whenever possible. Further, it is necessary that S_P^ϕ should be always negative if stability is to be ensured.

It is to be noted here that, in the normal course of events with an orthogonal grid, the finite-difference equation connecting the value of ϕ at P with its neighbours would only make reference to the nearest neighbours M, S, E and W; however, it is now expected that, owing to the additional G-terms, the equation in the non-orthogonal grid will also include values of ϕ at the more remote neighbours NW, NE, SW and SE.

The expression (3.4) may be substituted into equation (3.3) to yield the following equation for ϕ_P :

$$\phi_P = A_E^\phi \phi_E + A_W^\phi \phi_W + A_N^\phi \phi_N + A_S^\phi \phi_S + B^\phi ; \quad (3.7)$$

$$\text{where } A_E^\phi = A'_E / A'_P ,$$

$$A_W^\phi = A'_W / A'_P ,$$

$$A_N^\phi = A'_N / A'_P ,$$

$$A_S^\phi = A'_S / A'_P ,$$

$$B^\phi = B' / A'_P ; \quad (3.8)$$

$$\begin{aligned}
 \text{and } A'_E &= T_e^\xi - L_e^\xi, \\
 A'_W &= T_w^\xi + L_w^\xi, \\
 A'_N &= T_n^\eta - L_n^\eta, \\
 A'_S &= T_s^\eta + L_s^\eta, \\
 B' &= F_{PU}^\zeta \phi_{PU} + S_U^\phi, \\
 A'_P &= A'_E + A'_W + A'_N + A'_S + F_{PU}^\zeta - S_P^\phi. \quad (3.9)
 \end{aligned}$$

Equation (3.7) represents the final form of the required finite-difference equivalent of the differential equation (3.1). It relates ϕ_P to the ϕ values at neighbouring nodes on the numerical grid. Further remarks on evaluation of 'A' coefficients are made in the next section.

Equations for the velocity components

The finite-difference equations for the velocity components are expressed in the same manner of equation (3.8) but contain an additional term representing the pressure gradient.

ξ -direction momentum equation:

$$\begin{aligned}
 u_P &= A_{NN}^u u_N + A_{SS}^u u_S + A_{WW}^u u_W + A_{EE}^u u_E + B^u \\
 &+ D_P^u (P_W - P_P) + D_P^{uv} (P_N + P_{NW} - P_S - P_{SW}). \quad (3.10)
 \end{aligned}$$

η -direction momentum equation:

$$v_P = A_N^V v_N + A_S^V v_S + A_W^V v_W + A_E^V v_E + B^V + D_P^V (P_S - P_P) \quad (3.11)$$

ζ -direction momentum equation:

$$w_P = A_N^W w_N + A_S^W w_S + A_E^W w_E + A_W^W w_W + B^W + D_P^W (P_{PD} - P_P) + D_P^{WV} (P_N - P_S) \quad (3.12)$$

The A and B coefficients in the above equations have similar meanings to those in equation (3.8). The new symbols D_P^u , D_P^{uv} , D_P^v , D_P^w and D_P^{wu} are defined as follows:

$$\begin{aligned} D_P^u &= (\delta\eta\Delta\eta)_w \delta\zeta/A_P'^u ; \\ D_P^{uv} &= .25 (G\delta\xi)_w \Delta\xi_E \delta\zeta/A_P'^u ; \\ D_P^v &= (r\delta\xi)_n \Delta\xi_E \delta\zeta/A_P'^v ; \\ D_P^w &= - (r\delta\xi\delta\eta\Delta\eta)_P \Delta\xi_E/A_P'^w ; \\ \text{and } D_P^{WV} &= .5 F_P (r\delta\xi)_P \Delta\xi_E/A_P'^w . \end{aligned} \quad (3.13)$$

The A's have the same meanings as in equation (3.7). Note the following modifications for the u, v and w equations:

- (i) The control volume cells for u and v are different from those for other ϕ 's, as illustrated in Figure 3.2.3. Hence, the appropriate inter-nodal distances and cell dimensions must be used in evaluating the A's and B's.
- (ii) The transport coefficient and mass fluxes for the u and v control volumes must be calculated in an appropriate manner.
- (iii) In all the three equations, the components of pressure is left as explicit; however, they are truly calculated as part of B's as will be seen later.

It is also to be noted that D_p^{uv} and D_p^{wv} represent the additional pressure gradient contribution that drives the respective velocity due to the non-orthogonality of the grid. When the grid is orthogonal D_p^{uv} and D_p^{wv} will be equal to zero (because G and F factors are zero).

The continuity equation

The discretisation of the continuity equation is quite simple. It merely states that the inflows and outflows of mass are locally in balance for each control volume in the flow domain. The discretised

form of the continuity equation may be obtained directly by substituting a value of unity for ϕ in the integral equation (3.3) and setting T's and S_ϕ 's to zero.

The resulting expression is:

$$F_P^\zeta - F_{PU}^\zeta + L_N^\eta - L_S^\eta + L_E^\xi - L_W^\xi = 0 ; \quad (3.14)$$

where L_I^J represents the mass velocity along the J direction at location I, and is defined in equation (3.4).

3.2.6 Upwind-differencing practice

The upwind-differencing practice was briefly introduced in the previous section (point (iii) under the heading of basic decisions). In this section, its need and its implementation are explained.

The finite-difference equations (equation (3.7)) implies that when the source terms S_U^ϕ and S_P^ϕ , are zero, ϕ_P will be a weighted average of ϕ_E , ϕ_W , ϕ_S , ϕ_N and ϕ_{PU} . This means that the value of ϕ_P must always lie between the highest and the lowest values of ϕ at the surrounding nodes. Thus, the weighting factor ('A' coefficient) of each variable must always be positive and less than one. However, the expression of 'A' coefficients (equations (3.7) and (3.9)) do not ensure this. For

instance A_N can take a negative value when L_n^η is positive and greater than T_n^η (since T_n^η is always positive by definition of the diffusion law). Similarly, A_S can be negative if L_s^η is negative and greater than T_n^η . Similar arguments apply for A_E and A_W .

To overcome this obstacle, an upwind-differencing practice is employed; according to which the convective coefficients (L's) are checked and corrected in the following manner:

- if L_n^η is positive, set it equal to zero;
- if L_s^η is negative, set it equal to zero;
- if L_e^ξ is positive, set it equal to zero; and
- if L_w^ξ is negative, set it equal to zero.

Thus, the weighting effect of the convection of the neighbouring node towards which the fluid is flowing is neglected in the calculation of ϕ_P .

3.3 The Solution Method

3.3.1 Main features

The aim of the solution procedure described here is to seek a simultaneous solution of the finite-difference equations expressed by equations (3.10), (3.11), (3.12) and (3.14). The unknowns in these

equations are the three velocity components u , v and w , the pressure and the scalar flow properties (such as the diffusion coefficients). Since the three momentum equations are coupled through the unknown pressure field, it has been necessary to employ an iterative procedure for their solution in which the pressure field is guessed, and is corrected so as to bring the velocities in conformity with the continuity equation. The equations being non-linear, their correction-procedure necessitates more than one iteration. The calculation procedure described here is based on the method reported earlier by Patankar and Spalding, 1972a, and with its extension for partially-parabolic flows as reported by Spalding, 1975 and Pratap and Spalding, 1976; further, some modifications were found necessary due to the non-orthogonality of the grid. The main features of the present solution procedure are as follows:

- (1) The solution procedure employs a guess and correct procedure for the calculation of the correct pressure-field. Several 'sweeps' are made through the flow domain and the pressure field is corrected each time.
- (2) The flow variables are calculated by 'marching' through the flow domain, along the predominant flow direction; in this marching, the variables

at any longitudinal station are calculated partially from values at an upstream location, and from values in store at this particular location.

- (3) The finite-difference equations are solved by a point-by-point algorithm (PBP), where the values of the surrounding nodes to the node under consideration are taken from the previous sweep.
- (4) The continuity equation (known as the pressure correction equation) is solved by a linear equation solver algorithm.
- (5) The flow field is stored three-dimensionally and is 'updated' in each marching of the flow domain. Basically, only the pressure field is needed in a three-dimensional array, however, the author took the advantage of the computing system available for the research, and used a sort of fast core-off random access memory to store the whole field three-dimensionally. This feature will speed up convergence, and thus reduce the computational cost.

Further details of the calculation steps are provided below.

3.3.2 Sequence of calculation steps

The sequence of calculation steps in the partially-parabolic flows is as follows:

- (1) The pressure field which is stored in a three-dimensional array is assigned guessed values at the start of the integration.
- (2) Using the guessed values of the pressure field, the momentum equations in the ξ , η and ζ directions at a downstream station are solved; the coefficients in the equations are evaluated on the basis of flow properties at the upstream longitudinal station. If local iterations are performed at each axial plane, then, the coefficients are evaluated from the flow properties of the preceding iteration. The sequence of the calculations has been to solve first for the u and v velocities and then for the w velocity which is situated ahead of the cross-stream plane containing the u and v velocities.
- (3) The pressure field is corrected to bring the velocities in conformity with the continuity equation. These corrections are applied for one cross-stream plane at a time, and are calculated as follows:

- (a) The pressure and velocity fields are first expressed as:

$$\begin{aligned} P &= P^* + P' & ; \\ u &= u^* + u' & ; \\ v &= v^* + v' & ; \\ w &= w^* + w' & ; \end{aligned} \quad (3.15)$$

where the "starred" quantities are the ones which are either guessed (pressure) or calculated by the guessed pressure field (velocity components); it is very unlikely that these quantities will satisfy the continuity equation. The primed quantities represent the required corrections to the approximate starred values to satisfy continuity.

- (b) The corrections to the velocities are related to the pressure corrections by substituting equation (3.15) in the momentum equations. The result of substitution is the following simplified expressions*:

* The complete expression for these terms are of the following form:

$$\begin{aligned} u_P &= u_P^* + A_N^u u_N' + A_S^u u_S' + A_E^u u_E' + A_W^u u_W' \\ &\quad + D_P^u (P_W' - P_P') + D_P^{uv} (P_N' + P_{NW}' - P_S' - P_{SW}') \end{aligned}$$

In the conventional SIMPLE algorithm u_N', u_S', u_E' and u_W' were taken as zeros and E_P^u will therefore be equal to D_P^u . To promote convergence, Tatchell, (1975), found that the assumption $u_N' = \gamma_N^u u_P'$, $u_S' = \gamma_S^u u_P'$ etc, where

* Continued...

$\gamma_N^u, \gamma_S^u, \dots$ etc, are constants, yields faster convergence and promotes stability. In this case the expression of E_P^u will equal to:

$$E_P^u = D_P^u / \{1 - \gamma_N^u A_N^u - \gamma_S^u A_S^u - \gamma_W^u A_W^u - \gamma_E^u A_E^u\}.$$

In the present work, all values of γ have been set equal to unity, assuming that the velocity corrections at the neighbouring nodes are likely to be almost the same as those at the node under consideration, i.e.

$$E_P^u = D_P^u / \{1 - A_N^u - A_S^u - A_W^u - A_E^u\};$$

similar expressions for E_P^{uv}, E_P^v, E_P^w and E_P^{wv} can be found.

$$u_P = u_P^* + E_P^u (P'_W - P'_P) + E_P^{uv} (P'_N + P'_{NW} - P'_S - P'_{SW});$$

$$v_P = v_P^* + E_P^v (P'_S - P'_P);$$

$$w_P = w_P^* + E_P^w (P'_{PD} - P'_P) + E_P^{wv} (P'_N - P'_S) \quad . (3.16)$$

(c). The relations expressed above are then substituted in the finite-difference form of the continuity equation (3.14); and assuming that upstream and downstream pressure corrections are zeros, one arrives at the expression for the pressure correction equation as:

$$A_P^P P'_P = A_E^P P'_E + A_W^P P'_W + A_N^P P'_N + A_S^P P'_S + A_W^{PY} (P'_{NW} - P'_{SW}) - A_E^{PY} (P'_{NE} - P'_{SE}) + \dot{m}_P; \quad (3.17)$$

where $P'_E, P'_W, P'_N, P'_S, P'_{NW}, P'_{SW}, P'_{NE}$ and P'_{SE} are corrections to the pressures at the nodes E, W, N, S, NW, SW, NE and SE; and the 'A''s

represent the products of the 'E' coefficients and the cell areas. The quantity $\dot{m}_p$ expresses the mass imbalance over the region surrounding the grid node P.

- (d) Equation (3.17) which has the same form as (3.8) is solved using a linear equation solver algorithm; the pressure and velocities are thereafter corrected accordingly.
- (e) Steps (b) to (d) are repeated until the continuity errors reduce practically to zero.
- (f) The equations for variables such as kinetic-energy of turbulence, etc, are solved so as to provide distributions appropriate to the downstream station.
- (g) The operations at the current station are now completed. The next downstream station is chosen and the process is repeated until the region of interest is covered. Once the flow domain has been covered, the calculation of one sweep has come to its end.
- (h) After completing a sweep (i.e. plane by plane march up to the last downstream section) the flow domain is swept once again, each time employing an improved pressure field (stored

from the preceding sweep). The procedure is terminated when the corrections to the pressure field become smaller than a pre-assigned value.

3.3.3 Methods of solving the difference equations

Reference has been made above to solve finite-difference equations of the form represented in equations (3.18) and (3.17). The general form of these equations is:

$$\Psi_P = A_N \Psi_N + A_S \Psi_S + A_E \Psi_E + A_W \Psi_W + P \quad ; \quad (3.18)$$

where Ψ can either be ϕ or P' . Two different algorithms were used: a PBP (point-by-point) algorithm and a linear equations solver algorithm. The PBP algorithm was used for solving the momentum equations; whereas, the linear equation solver algorithm was used for the pressure correction equation.

PBP algorithm

Let the value of Ψ available in store (old values) be $\bar{\Psi}$, then the new Ψ 's are calculated from the old values of Ψ 's as follows:

$$\Psi_P = A_N \bar{\Psi}_N + A_S \bar{\Psi}_S + A_E \bar{\Psi}_E + A_W \bar{\Psi}_W + P \quad ; \quad (3.19)$$

after calculating all values of Ψ , these values are then replaced in the store to become $\bar{\Psi}$. When convergence occurs, Ψ 's will be equal to $\bar{\Psi}$'s.

Linear solver

Suppose that Ψ_P can be linked to Ψ_E and Ψ_N through the following relation:

$$\Psi_P = N_P \Psi_N + E_P \Psi_E + Q_P, \quad (3.20)$$

where N , E and Q are coefficients to be found later, then:

$$\Psi_S = N_S \Psi_P + E_S \Psi_{SE} + Q_S ;$$

$$\text{and } \Psi_W = N_W \Psi_{NW} + E_W \Psi_P + Q_W. \quad (3.21)$$

Substituting the relation (3.21) into (3.18) yields:

$$\begin{aligned} \Psi_P = & A_N \Psi_N + A_S (N_S \Psi_P + E_S \Psi_{SE} + Q_S) \\ & + A_E \Psi_E + A_W (N_W \Psi_{NW} + E_W \Psi_P + Q_W) + B ; \end{aligned} \quad (3.22)$$

then,

$$\begin{aligned} \Psi_P (1 - A_S N_S - A_W N_W) = & A_N \Psi_N + A_E \Psi_E \\ & + \underline{P} + A_S (E_S \Psi_{SE} + Q_S) + A_W (N_W \Psi_{NW} + Q_W). \end{aligned} \quad (3.23)$$

Comparing the relation (3.23) to (3.20), one obtains:

$$\begin{aligned} N_P &= \frac{A_N}{D_P} ; \\ E_P &= \frac{A_E}{D_P} ; \\ Q_P &= \frac{\{P + A_S(E_S \Psi_{SE} + Q_S) + A_W(N_W \Psi_{NW} + Q_W)\}}{D_P} ; \end{aligned} \quad (3.24)$$

where,

$$D_P = 1 - A_S N_S - A_W E_W.$$

The computational procedure is therefore as follows:

- (1) The solution starts at the bottom-left-hand corner of the finite-difference grid in the following steps:
 - (a) Compute D_P (N_S and E_W are known at this corner).
 - (b) Compute N_P , E_P .
 - (c) Repeat steps (a) and (b) until the whole domain is covered.
- (2) Calculate Q_P using the in-store values of Ψ .
- (3) Then use $\Psi_P = N_P \Psi_N + E_P \Psi_E + Q_P$ to obtain the new Ψ 's, and replace these values in store. This computational step starts at the top-right-hand corner

of the finite-difference grid (Ψ_N , Ψ_E are known at this corner).

- (4) Repeat steps (2) and (3) until convergence.

3.4 Incorporation of Auxiliary Information

3.4.1 Initial conditions

Initial conditions of all variables can be supplied to the calculation procedure in a simple manner. For example, a distribution of experimentally-determined velocities at the inlet section may be supplied as a function of grid position.

In the absence of any knowledge of the initial profiles (as it is the case of the present investigation), the approach indicated in Section 2.4 is used.

3.4.2 Boundary conditions

Reference was made in Section 2.4 to two types of boundary conditions encountered in this thesis, namely: whether the value of a variable, or the value of its flux is specified. The incorporation of either kind of boundary condition into the calculation procedure is easy, and is achieved by modifying the source term in the finite-difference equation for the near boundary control volumes; the appropriate coefficients are then set to zeros.

The treatment of wall boundaries were given in detail in Section 2.5.3. It is to be mentioned here that, in the non-orthogonal coordinate system adopted for this work, all the three-velocity components u , v and w will give rise to wall shear depending on the non-orthogonality of the grid.

3.5 Remarks on the Numerical Procedure

3.5.1 Accuracy of the procedure

Analytical exact solutions are, in general, not available to confirm the accuracy of the numerical results obtained by the calculation procedure. The assessment of the accuracy of the predictions was therefore made against itself. Solutions obtained by finite-difference methods are grid dependent except for very small finite-difference grids. The basic aim is, therefore, to establish a grid with the smallest possible number of nodes for which the finite-difference solutions are essentially those of the original differential equations. This is achieved in part by obtaining solutions with increasing numbers of grid nodes until a stage is reached where the solution exhibits negligible change with further increase in the number of grid nodes, or with small adjustments in their distributions. Two basic criteria were considered in the exploration of a suitable finite-difference grid; firstly, grid boundaries were located

where there was little influence from the hull, and secondly, for computing economy the number of grid nodes were minimised. A compromise between these criteria was achieved by a non-uniform distribution of grid lines in such a way that the nodes were concentrated near walls and less nodes nearer to the free-boundary. To this end, solutions were obtained with successive refinements of grid size, i.e. reduction in the spacing between nodes in the η - and ξ -directions, as well as in the ζ -direction. When solutions are observed, in respect of several details, to be unaffected by further refinement, they are presumed to be grid-independent. Two criteria are chosen as a test for grid-independence, namely: the stagnation pressure coefficient*, C_{p_o} , and axial velocity profiles. Grid refinement tests are made first in the axial direction, then in the cross-stream direction.

To illustrate the grid refinement tests and their effect, the tests made for the test case reported in Chapter 5 are shown in Figures 3.5.1 and 3.5.2. Figure 3.5.1a and 3.5.1b illustrate grid refinement tests made in the axial direction. Four different arrangements of grid

* The stagnation pressure coefficient is defined as:

$$C_{p_o} = \frac{P + \frac{1}{2}U^2}{\frac{1}{2}U_\infty^2} ;$$

where $U^2 = u^2 + v^2 + w^2$. The values of u , v , w and P are calculated at the near wall grid point.

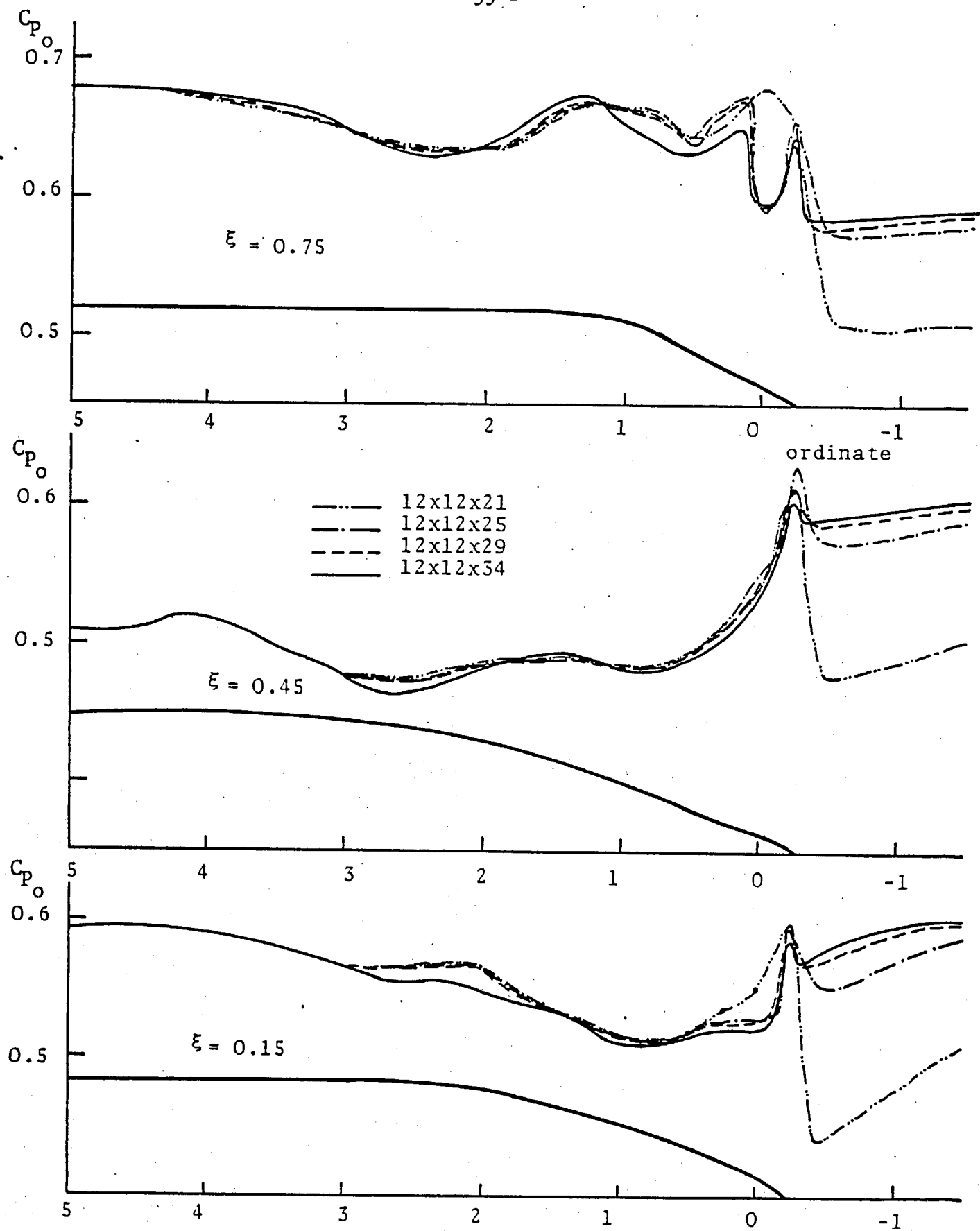


Figure 3.5.1.a Grid-refinement tests (longitudinal direction)

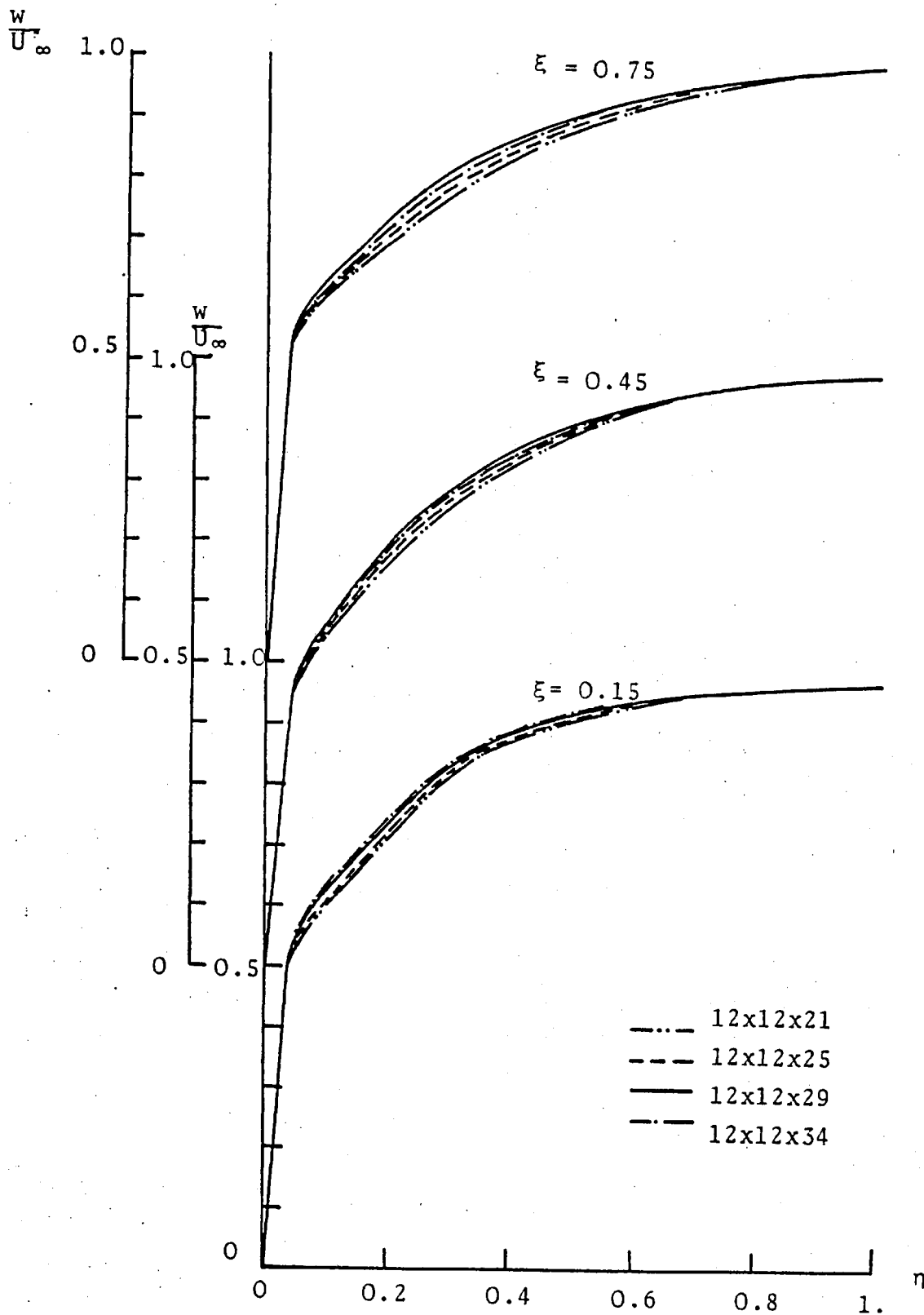


Figure 3.5.1.b Grid-refinement tests (Longitudinal direction) continued.

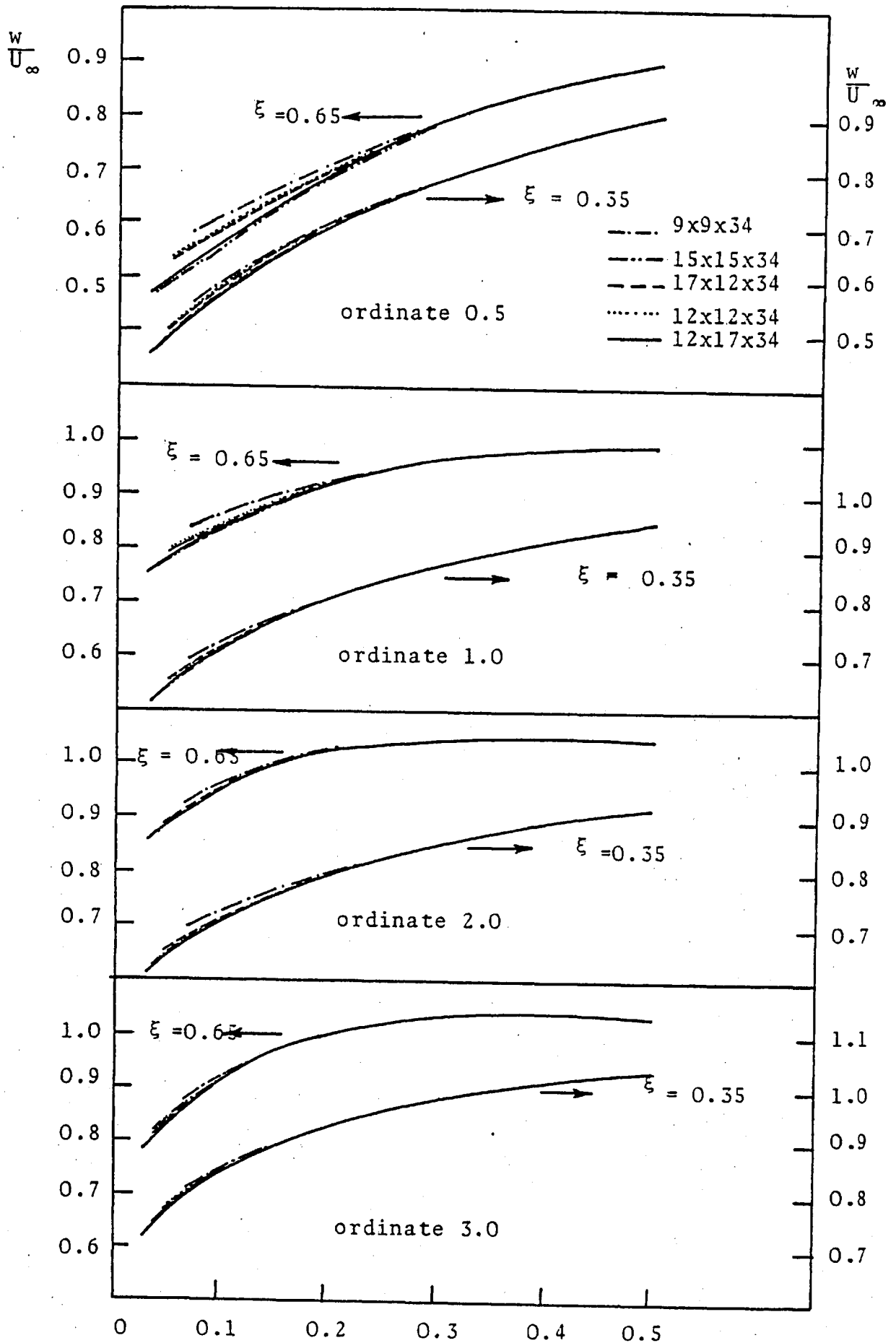


Figure 3.5.2 Grid refinement tests (cross-section)

distribution are compared together; the grid distribution in the cross-stream direction is kept the same in all cases, while the number of nodes in the axial direction were increased from 21 to 34, by adding more nodes near the stern. The test indicates that 34 grid points in the axial direction are enough to ensure grid-independence in that direction. On the other hand, Figure 3.5.2 illustrates, grid refinement tests made in the cross-stream direction. Five different arrangements were made. The tests show that the number of grid points in the circumferential direction has minor influence on grid independence tests, and that the number of grid points in the radial direction is the dominant one. A number of grid points of 17 in the radial direction was found to give grid independence up to ordinate 1.. Downstream of ordinate 1., the grid does not show the same independence. Further refinement of the grid was restricted due to the prohibitive computational cost expected.

It can be concluded from these tests that the results obtained by the grid 12x17x34 is grid-independent up to ordinate 1. only. Downstream of ordinate 1., the results obtained are slightly grid-independent.

3.5.2 Convergence of the procedure

A matter which needs urgent investigation is the rate of convergence of the partially-parabolic procedure.

Despite the fact that the procedure has many advantages for the problems for which it is intended, over a fully elliptic procedure, there is considerable room for improving its economy, by reducing the number of sweeps required.

The convergence of the partially-parabolic method is solely determined by the initial guess for pressure; if this initial guess is the correct one, the momentum and continuity equations are satisfied simultaneously and the solution is obtained in one sweep over the flow domain; on the other hand, if the initial guess for pressure is quite far from the correct one, several sweeps are required to obtain the correct solution. The sweeps were terminated when further sweeps caused a change of the flow variables at monitoring locations of less than 1%. Figure 3.5.3 illustrates typical variation of the value of the axial velocity, at different grid locations, with the number of sweeps; 60 sweeps were found necessary to dampen out such oscillatory behaviour. Similar conclusions can be obtained from Figure 3.5.4, where the variation of the pressure, at the monitoring locations, with the number of sweeps is shown. The calculations presented in the next chapter are those after 70 sweeps.

To promote convergence, local iterations are made at

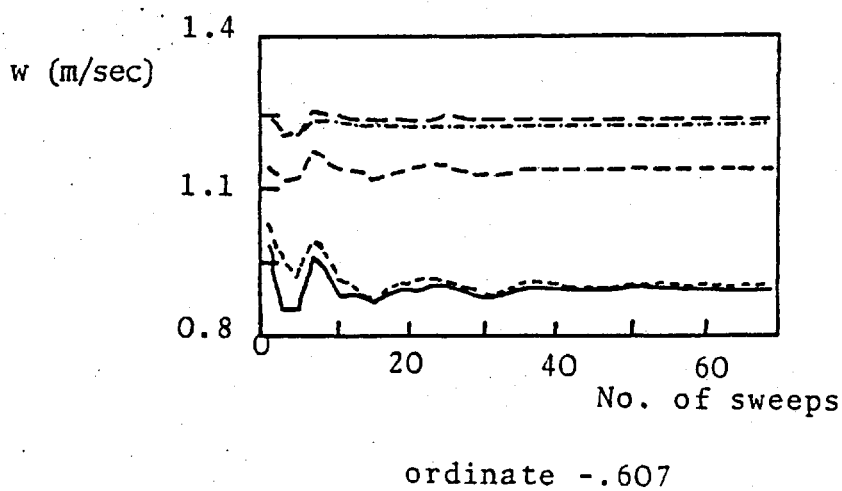
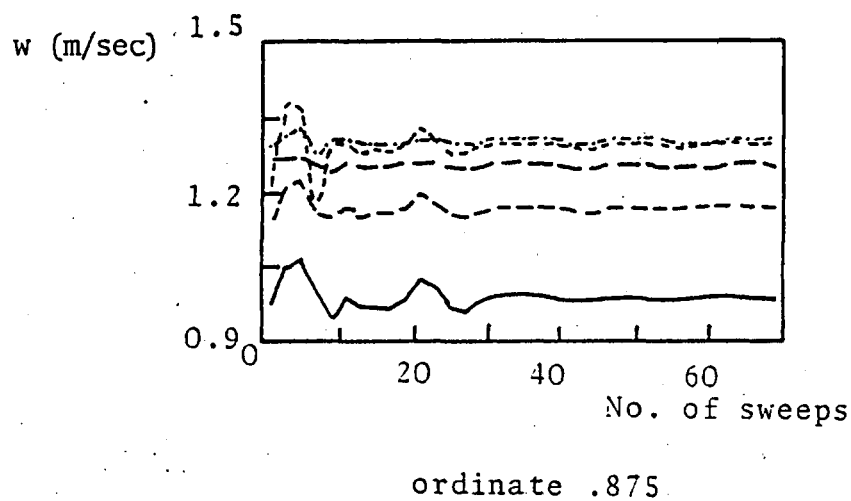
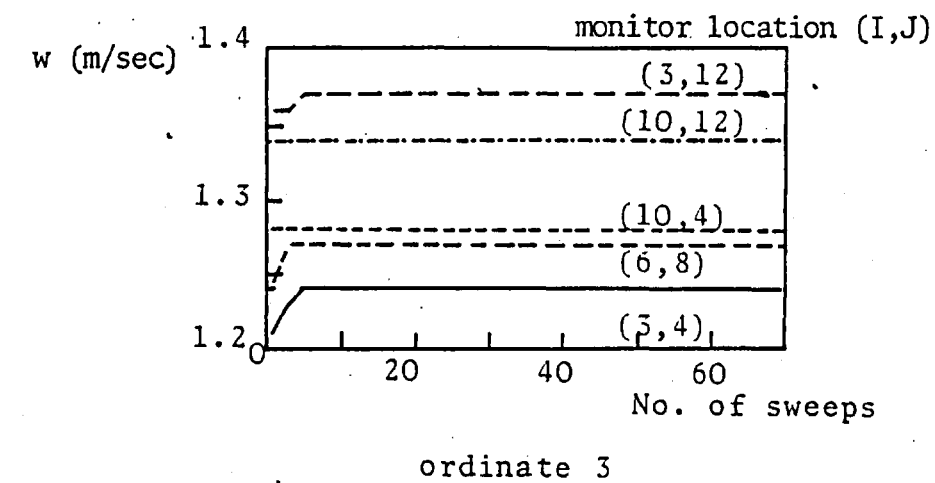


Figure 3.5.3 Variation of longitudinal velocity with sweeps.

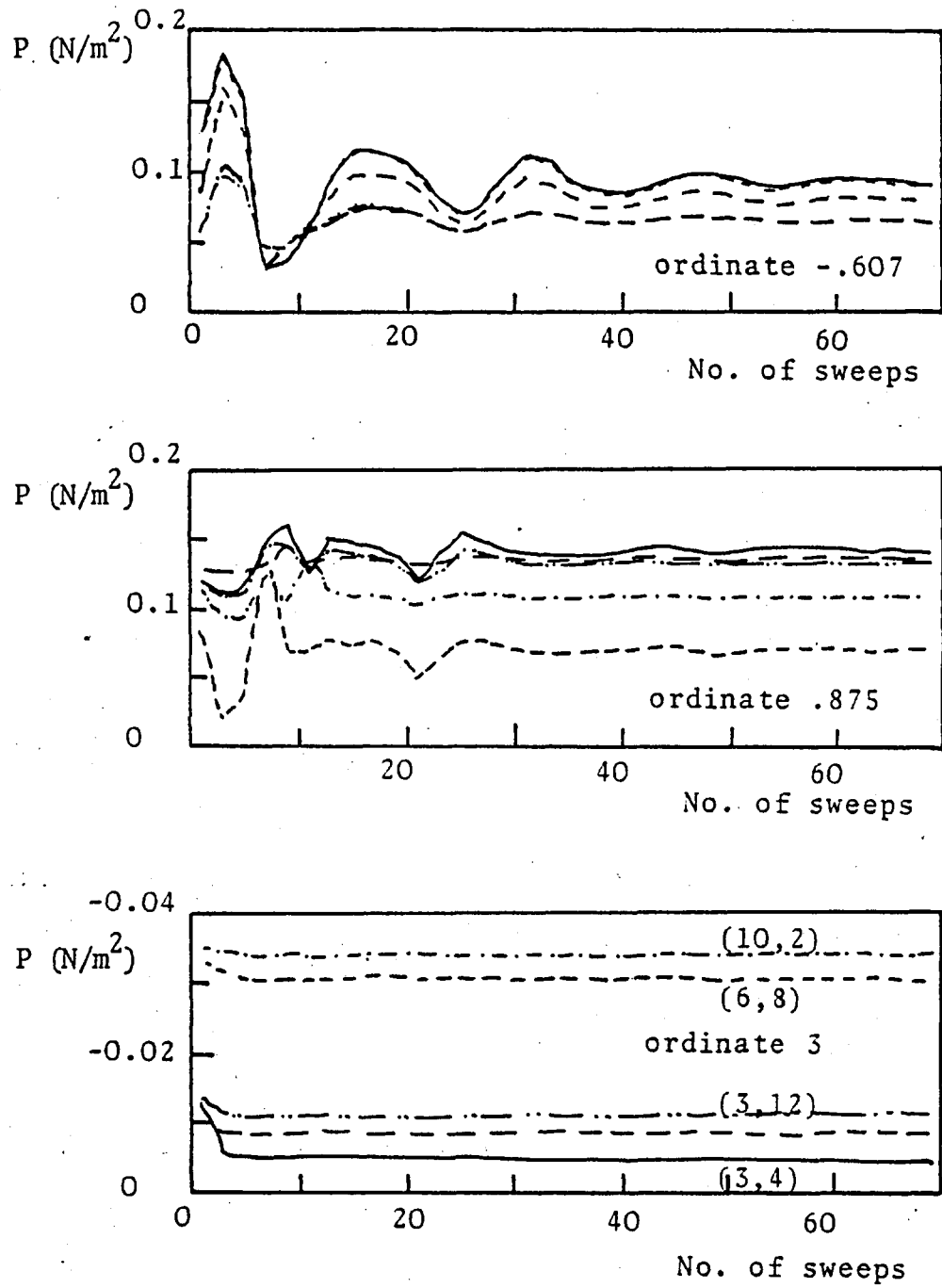


Figure 3.5.4 Variation of pressure with sweeps

each longitudinal station. Two criteria for convergence of local iterations were made as iteration proceeded. Mainly they were the normalised sum of the absolute values of the mass imbalance for all computational cells over the whole section, and the normalised maximum pressure correction. When both criteria decrease and become less than .001, then the local iterations are terminated.

3.5.3 Stability

As we start our calculation with the guessed pressure field, it may happen, under some conditions of grid distribution and initial condition, that implausible velocity fields are found which lead later to instability and divergence.

In the course of iterations for the test case presented in Chapter 5, negative velocities were found to occur; the partially-parabolic procedure is not capable of dealing with negative velocities, and therefore, instability occurred which was followed by divergence. To overcome this instability, a trap was made for the negative axial velocities, and they were replaced by small positive ones.

3.5.4 Computer time

The computational cost is very much dependent on the particular geometry solved for. The computer time on a CDC 174 computer was of the order of 1.5 sec for the sweep per one longitudinal station for a cross-stream grid of 12x17.

3.6 Closure

In this chapter, a novel non-orthogonal computational grid has been described in detail, and the partial-differential equations were integrated over the appropriate grid control volume to yield equivalent finite-difference forms. Furthermore, a general solution scheme for these equations has been described. Remarks on the numerical procedure were also given at the end of this chapter. This completes the mathematical formulation of the problem investigated in this thesis.

The validation of the above mentioned calculation procedure will be presented in Chapter 4; two simple test cases, namely: flow in a pipe, and flow past a Rankine body are used to test the procedure. The application of the calculation procedure will be presented in Chapter 5, where the procedure is used to predict the flow around a real ship, and the results are compared with existing measurements.

CHAPTER 4 - THE APPLICATION OF THE PREDICTION

METHOD TO TWO SIMPLE TEST CASES

4.1 Introduction

In Chapter 3, a numerical procedure for the prediction of turbulent flow around ships' hulls was presented. The procedure employed a non-orthogonal finite-difference grid to fit the flow domain of concern. It is in the present chapter that the procedure is examined and tested for simple flow cases. Two such tests are made, and their results are presented in this chapter. In the first case, predictions were made for flow between two concentric pipes, with the calculation domain made to be irregular. The objective of this test was to examine the different assumptions regarding the non-orthogonality effects in the cross-stream direction. In the second test case, prediction for the flow over a Rankine body was made. The objective of this test was to examine the non-orthogonality effects in the longitudinal direction.

The rest of this chapter is divided into three sections. Section 2 describes the tests made on flow between two concentric pipes; Section 3 presents the results for the flow past a Rankine body. Finally, Section 4 presents the closure for this chapter.

4.2 Test Case (A): Flow In Pipes

4.2.1 Preliminaries

A preliminary test of the equations derived for the non-orthogonal grid was performed by obtaining solution for the velocity field under fully-developed conditions of laminar flow between two straight concentric pipes.

The calculation domain was similar in shape to that used later for ship's hull case, and was located in a quadrant of the pipe as illustrated in Figure 4.2.1. A point on the axis of the pipe was taken as the origin of the coordinate system, and the family of constant η -lines coincide with the radii of the circle about that point. The boundaries of the calculation domain were set so as to coincide mainly with the outer cylinder, and to form a shape similar to a hull shape near the inner cylinder. The well-known fully-developed solution for the velocities, corresponding to a given flow rate (or a pressure drop) and a ratio of outer cylinder diameter to inner cylinder diameter, were prescribed on all boundaries (the outer boundary velocities are zero). The object of the exercise is to compute the values of the velocities at the internal grid nodes, and to compare them with the exact ones; thus enabling us to test the validity of the procedure in a simple way. Incidentally, the flow is a parabolic one.

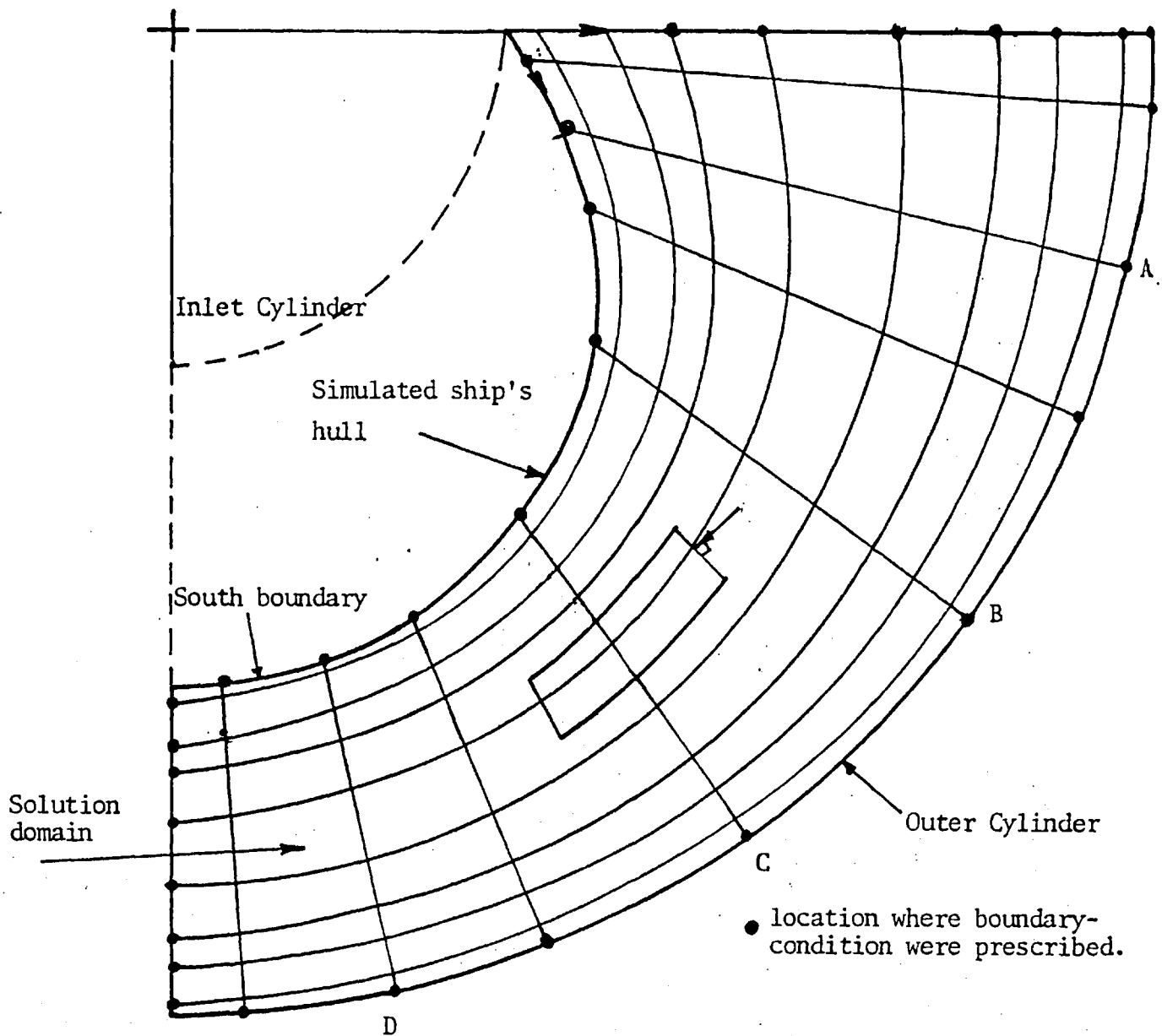


Figure 4.2.1 Calculation domain for Test Case A

The geometry of the inner surface was prescribed as a sinusoidal function, where the radius was given by the equation:

$$R_{in} = r_i + 0.1 \cdot \sin(\theta) \quad , \quad (4.1)$$

where R_{in} is the radius of the south boundary;

r_i is the radius of the inner cylinder (and equal to .1m in the present test); and

θ is the angle measured from the top surface (where $\xi=0$).

The outer cylinder radius, r_o , was 0.3m in the present investigation. Figure 4.2.1 is in fact drawn to a scale of $\frac{1}{2}$ of the actual geometry.

4.2.2 Inlet and boundary conditions

$$\text{At } z=0, u=v=0, \text{ everywhere} \quad ; \quad (4.2)$$

$$w=\text{constant}, \text{ everywhere} \quad ; \quad (4.3)$$

$$z>0 \quad u=v=0 \text{ at } \xi=0; \xi=1; \eta=0; \eta=1 \quad ; \quad (4.4)$$

$$w = - \frac{1}{4\mu} \frac{dP}{dz} r_o^2 \left[1 - \left(\frac{r}{r_o} \right)^2 + \left(\frac{1-\kappa^2}{\ln(\frac{1}{\kappa})} \right) \ln \left(\frac{r}{r_o} \right) \right]$$

$$\text{at } \xi=0, \xi=1, \eta=0 \quad ; \quad (4.5)$$

$$w=0 \text{ at } \eta=1 \quad ; \quad (4.6)$$

where $\frac{dP}{dz}$ is the pressure gradient, and κ is the ratio between the inner to outer radii ($= \frac{r_i}{r_o}$).

4.2.3 Computational results

A nearly uniform 10x10 grid was employed in the η - ξ plane, and the integration was performed until the axial velocity w at all internal grid nodes reached their constant final values.

The distribution of η and ξ was:

.0, 0.05, .15, .25, .4, .6, .75, .85, .95, 1.

As the flow is parabolic, one sweep is sufficient to obtain a converged solution.

4.2.4 Results

The predicted velocity distribution within the calculation domain is compared with the exact fully developed velocity profile in Figure 4.2.2.

Figure 4.2.2 present the axial velocity distribution with respect to the non-dimensional coordinate η , at four different angular stations A, B, C and D (see Figure 4.2.1). It is clear that the predictions for the axial velocities well agree with the analytical solution.

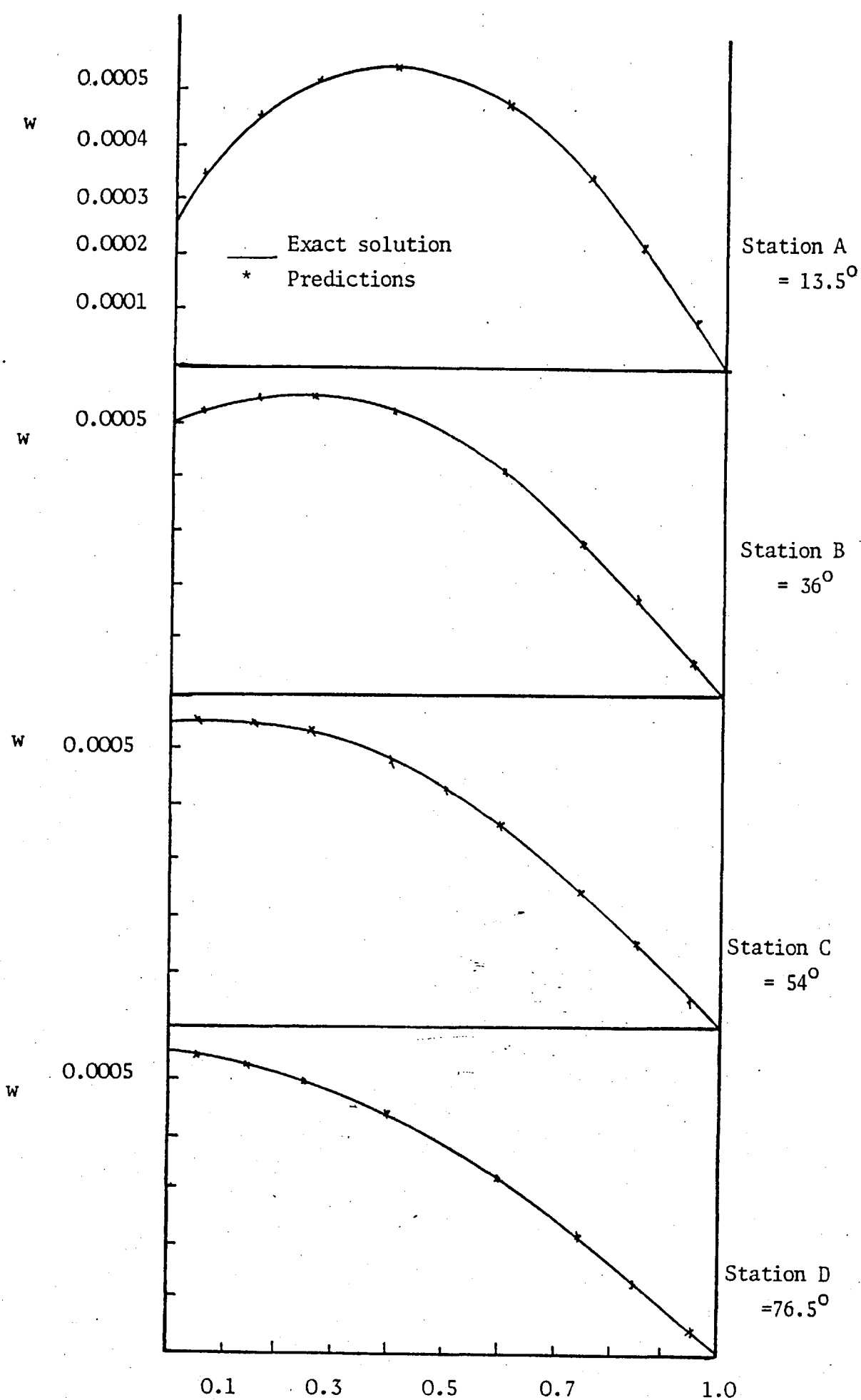


Figure 4.2.2 Comparison of the fully developed axial velocity distribution with the predicted one

4.3 Test Case (B): Flow Over A Rankine Body

4.3.1 Preliminaries

A more rigorous test of the procedure is to predict the inviscid-flow past a Rankine body (see Figure 4.3.1); where the body is having a variable shape in the longitudinal direction, thus giving a similarity to ships' hulls. Comparison of the predicted and exact solutions for the velocity and pressure fields were made.

The Rankine body is, in fact, an imaginary-theoretical-body that results from superimposing a uniform flow on a combination of a source and a sink. When the total strength of the source and sink is zero, and the uniform flow is in a direction parallel to that from source to sink, there will be, in typical cases, a closed loop curve dividing the flow region into two separate ones. The flow outside the loop is that of the uniform stream deflected by the presence of the loop. The flow inside the loop is self-preserved, since the source has the same strength as the sink, all the flow emanating from the source is absorbed by the sink. If, now, the closed loop is replaced by a solid body of the same profile, then the exterior flow will remain the same for the inviscid case considered. Analytical solution of the problem yields to solutions for the velocities and pressures, corresponding to a given

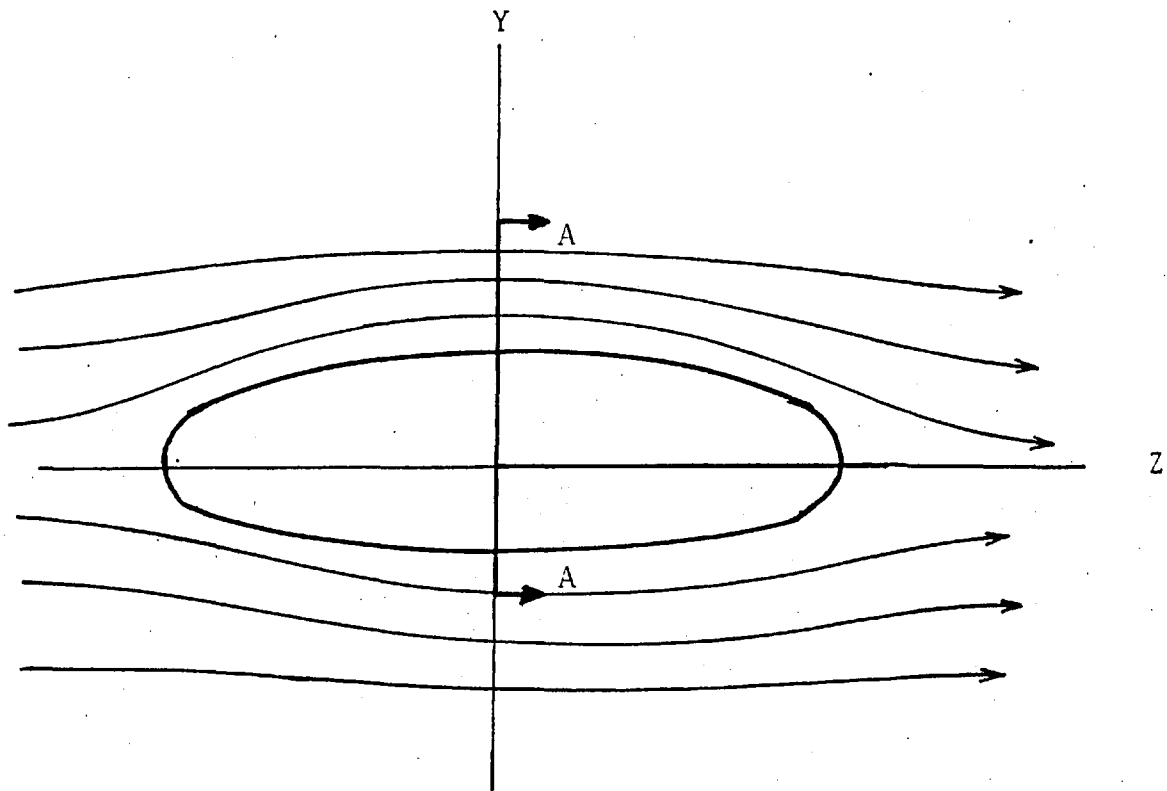


Figure 4.3.1 The Rankine body

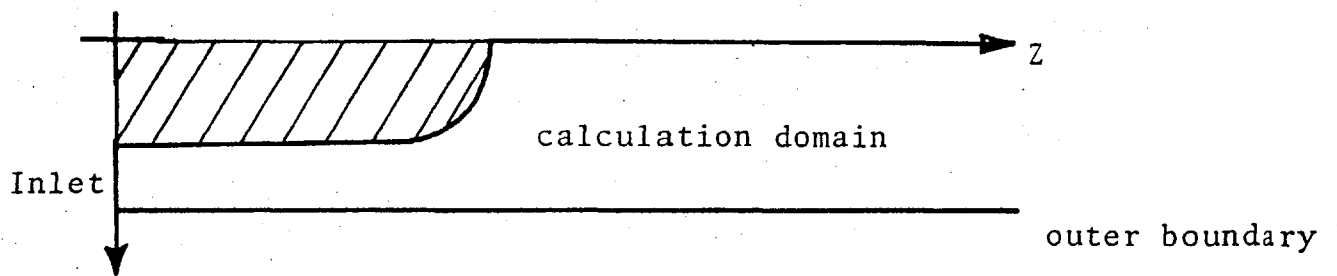


Figure 4.3.2 The calculation domain

shape number*.

The object of the present exercise is to compare the prediction of an inviscid flow over a body identical to a Rankine body having particular shape number, β , ($\beta=.1$ in the present investigation) with the analytical solution of the Rankine body with the same shape number. The calculation domain was made similar in shape to that of a ship's hull as shown in Figure 4.3.2. The larger axis of the body was taken to be the main axis. The boundaries of the calculation domain was set so as to be similar to the ship's hull calculation. The exact analytical solutions for the velocities and pressures corresponding to a given shape number of the body were prescribed on all boundaries; the object of the exercise was to compute the values of velocities and pressures at the internal grid nodes, and to compare them with the exact ones, thus enabling us to test the procedure.

The geometry of the Rankine body was prescribed by the equation:

$$Z = \pm \sqrt{1-Y^2 + 2Y \cot\left(\frac{2Y}{\beta}\right)} \quad (4.7)$$

* The shape number is defined as $\beta = \frac{m}{\pi U_{\infty} h}$ where m is the strength of the source or sink; U_{∞} is the velocity of the uniform flow; and h is half the distance between the source and the sink.

where Y, Z are the normalised vertical and horizontal coordinates respectively; coordinates y, z are normalised to h , which equals half the distance between the source and sink; β is the shape number.

The outer cylinder radius, r_o , was chosen to equal to $.175(=\frac{r_o}{h})^*$. The maximum value of the height of the body, $Y_{\max}$, for the particular shape number of .1 is .14088.

4.3.2 Inlet and boundary conditions

The inlet and boundary conditions are computed from the exact analytical solution:

$$u=0 \quad \text{everywhere} \quad ; \quad (4.8)$$

$$v = -\frac{2}{\beta} U_{\infty} \frac{ZY}{(Z^2+Y^2-1)^2+(2Y)^2} \quad ; \quad (4.9)$$

$$w = U_{\infty} \left\{ 1 - \frac{(Z^2-Y^2-1)^2}{(Z^2+Y^2-1)^2+(2Y)^2} \right\} \quad . \quad (4.10)$$

Inlet conditions

The calculation starts at half the body length. At this plane, the inlet conditions are computed from equations (4.9) and (4.6) by setting $z=0$, which

* The outer cylinder radius has been also used in the calculation in a normalised form.

reduces to:

$$u=v=0$$

$$w = U_{\infty} \left(1 + \frac{\beta}{1+Y^2}\right) \quad (4.11)$$

Free boundary conditions

At the free-boundary of the calculation domain, the flow variable w (axial velocity) and P (static pressure) are given the values obtained from equations (4.9) to (4.10) at the value of Y equal to $\frac{r_o}{h}$. Additionally, the static pressure at the near boundary node is set equal to the pressure at the boundary and the mass flow rate across the boundary is computed from local continuity.

Exit boundary conditions

At the exit of the calculation domain (about half of the body length downstream of the stern), an additional boundary condition is needed for the pressure; the pressure is set to the value specified by the potential flow solution.

4.3.3 Computational details

A parametric study of the grid showed that a grid of 16×31 is fine enough to describe the flow accurately. A non-uniform grid distribution was used so as to

concentrate more nodes near the body in the radial direction, and near the tail in the axial direction. The partially parabolic procedure was used and 30 iterations were found to be necessary to ensure the convergence of the results.

4.3.4 Results

The results presented here are for the prediction of inviscid flow over the tail of a Rankine body whose shape number is .1, and with the free stream velocity of 1 m/sec.

Figure 4.3.3 presents the comparison between the exact and predicted axial velocity profiles along radial lines at various axial stations. The curve is presented in a normalised manner, where the axial velocity is expressed as w/U_∞ , while the radial distance is expressed as the normalised radial distance, η .

Figure 4.3.4 presents the comparison between the value of the predicted wall axial velocity, defined as w at the wall, with the exact one. The predicted value of w_w was taken to be equal to the value of w at the grid node near to the wall.

Figure 4.3.5 shows the comparison between the variation of the predicted wall static pressure coefficient,

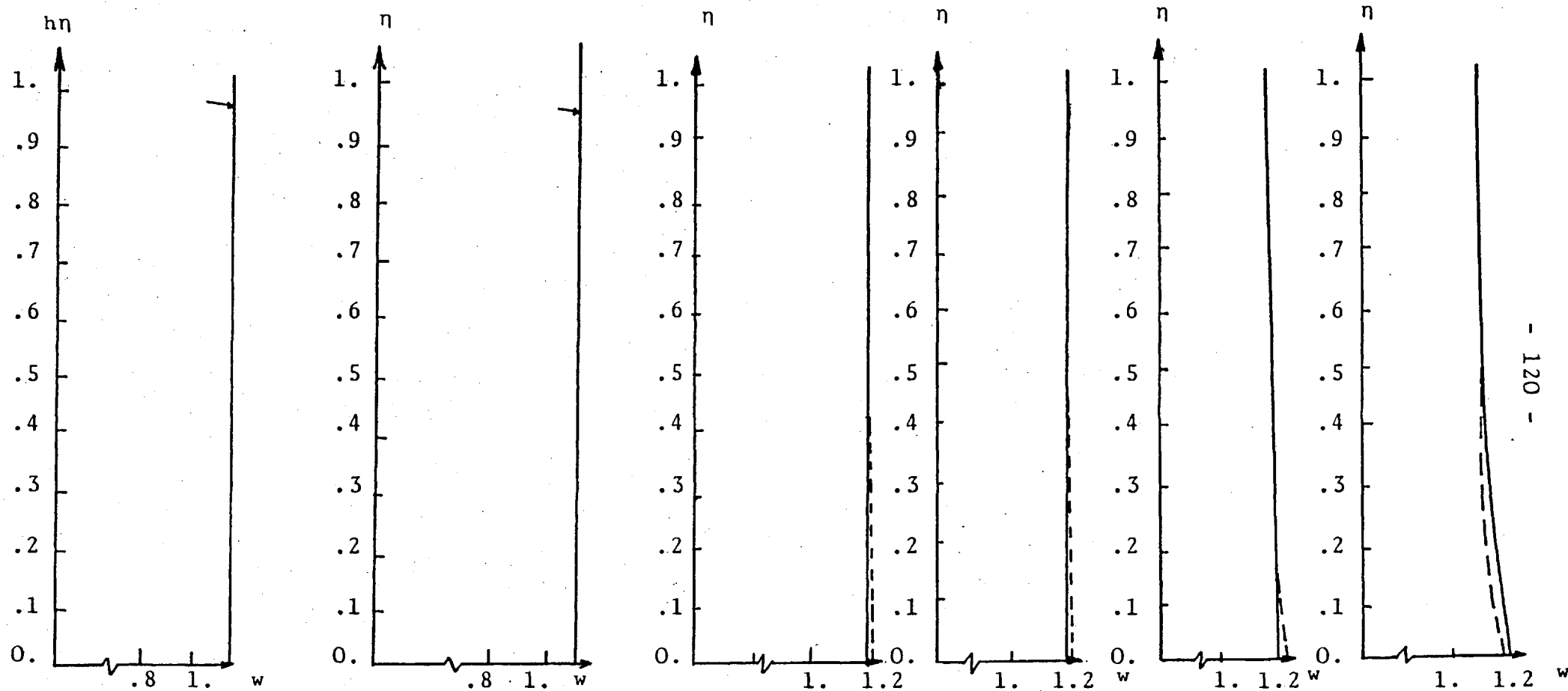


Figure 4.3.3 Comparison of Axial velocity profiles (___ Analytical, - - - Prediction)

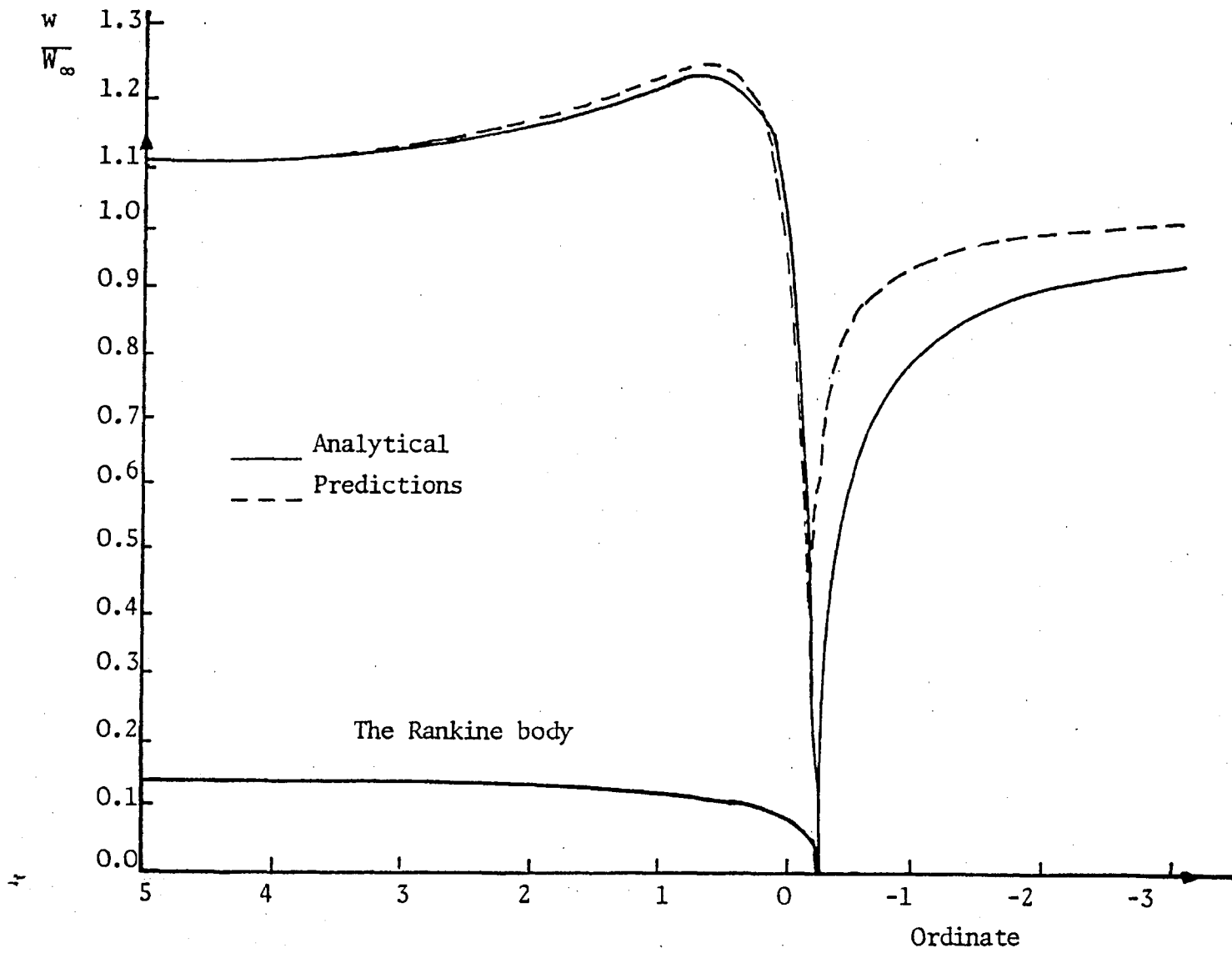


Figure 4.3.4 Comparison of wall axial velocity with analytical solution

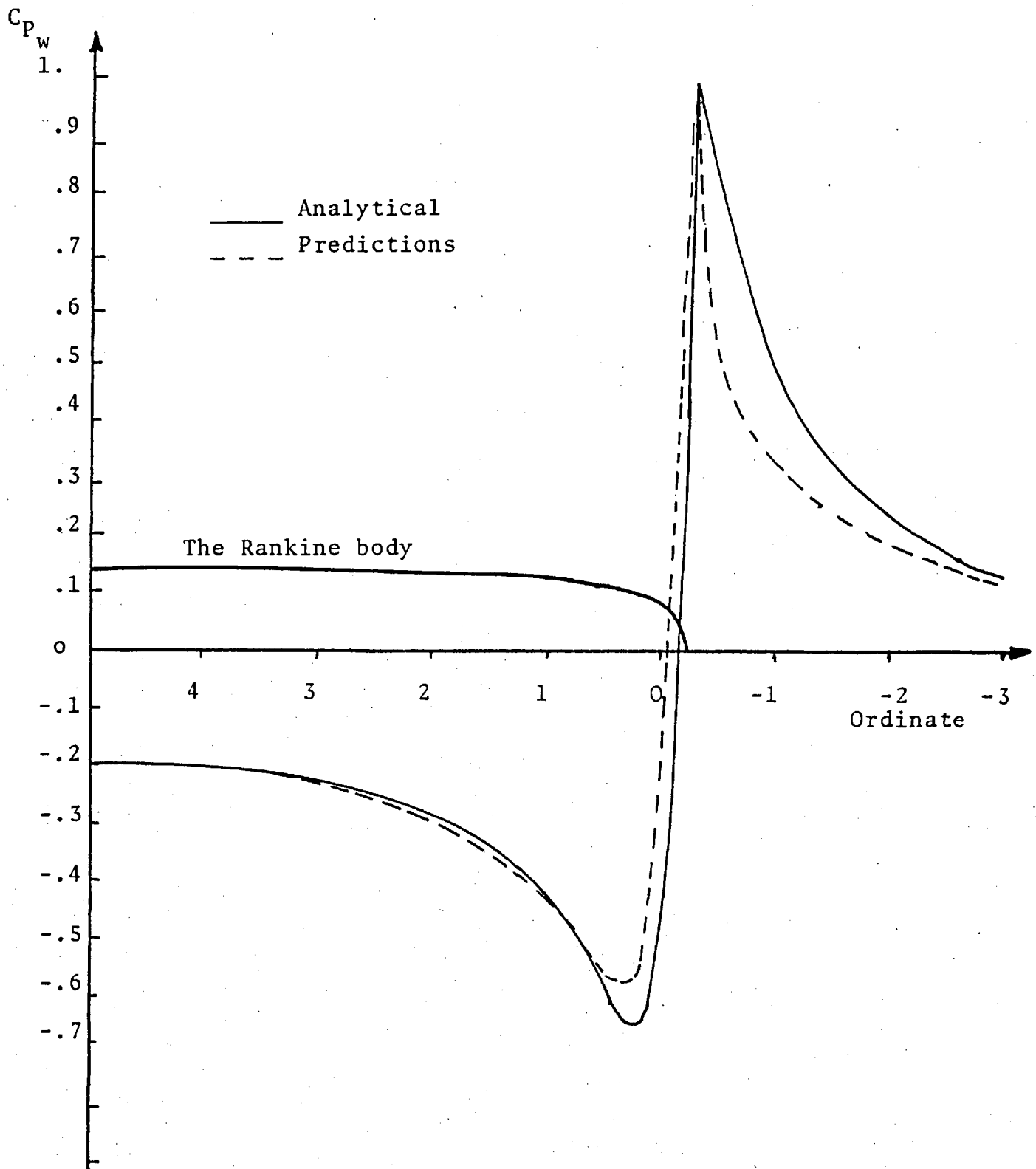


Figure 4.3.5 Comparison of wall static pressure C_{p_w} with analytical solution

defined as, $c_{p_w} = \frac{(P_w - P_\infty)}{\frac{1}{2}U_\infty^2}$, along the length ordinate position, with the exact solution at this boundary. Figures 4.3.3 and 4.3.4 show that the predicted axial velocity is in very good agreement with the exact one over most of the hull. Near the stern, the tail of the body, the agreement is not so good. The same can be seen from Figure 4.3.5, where the predicted wall static pressure is in good agreement with the exact one up to ordinate .5; nearer to the tail, downstream of ordinate .5, the predicted pressure at the wall is larger than the exact one.

Such discrepancies can mainly be attributed to inaccuracies resulting from the existence of the stagnation point at the tail of the body. Stagnation points are difficult to incorporate in finite-difference procedures except with the use of extremely fine grids which is time and cost consuming.

4.4 Closure

In summary, the validation of the prediction procedure presented in Chapter 3 has been done in the present chapter; two simple test cases, namely: flow between two concentric pipes, and flow past a Rankine body were used to validate the procedure.

Further tests were also reported by some of my colleagues who are now using the same numerical procedure. Malin and Markatos (1978) and Markatos et al (1979a, 1979b) reported several tests using a Patel body. Predictions were compared with available measurements, and a fairly good agreement was obtained. Ramachandra (1979) has also reported several tests for elliptic and equilateral-triangular duct flows: his predictions are in very good agreement with the exact solutions.

The application of the calculation procedure to real flow around ships' hulls will be presented in Chapter 5.

CHAPTER 5 - PREDICTION OF FLOW AROUND SHIPS

5.1 Introduction

In Chapter 3 a numerical procedure for the prediction of turbulent flow around ships' hulls was presented. The procedure employed a non-orthogonal finite-difference grid to fit the flow domain of concern. In Chapter 4, a statement about the accuracy of the non-orthogonal finite-difference formulation and the appropriate form of the grid was given. It is in the present chapter that the procedure is being used for the prediction of flow around ships.

Predictions were made for a real ship where measurements are available. The main predictions presented in the present chapter are obtained by the partially-parabolic procedure. Comparison of the predictions with the measurements showed good agreement over 80% of the hull. Near the stern, the comparison was not so good.

The rest of Chapter 5 consists of the following sections: Section 5.2 presents the general definitions and descriptions of the hull; Section 5.3 presents the details of the particular hull under consideration and the measurements along with the computational details. Section 5.4 presents the predictions made using the partially-parabolic procedure; Section 5.5 presents

the discussion of the results. Section 5.6 presents a comparison between the parabolic and partially-parabolic predictions with the measurements, while Section 5.7 closes the chapter with a brief assessment on the results achieved.

5.2 General Definition of the Hull

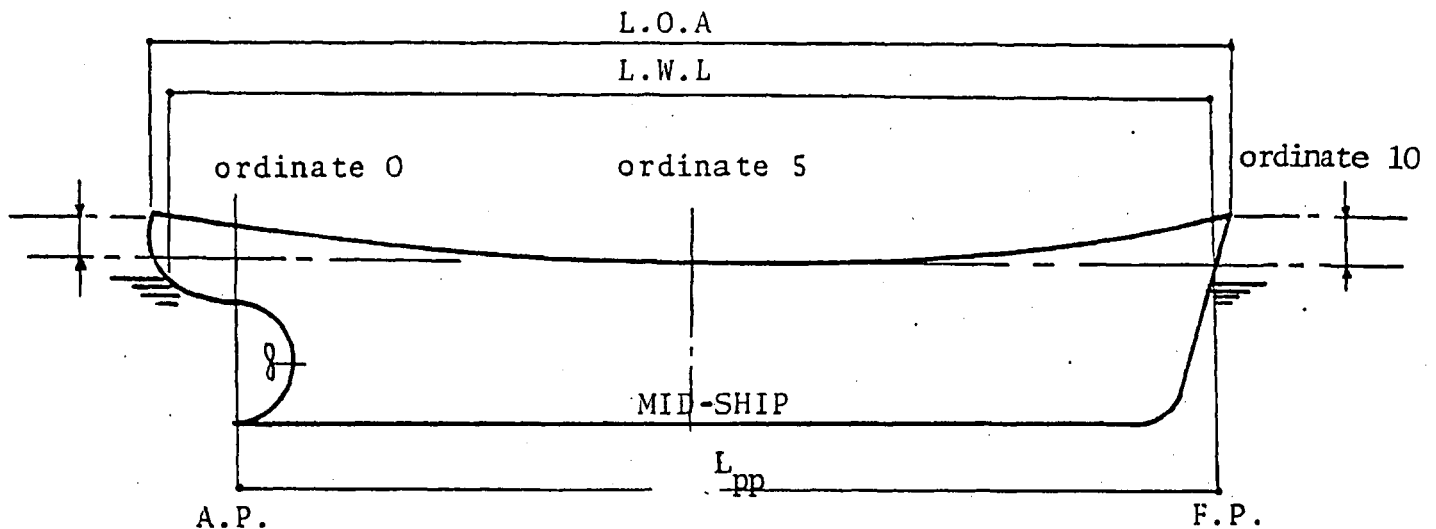
It has been common, in the field of ship hydrodynamics, to define the hull shape with respect to two main coordinates: the water-line and the ordinate. Water-lines are defined as horizontal surfaces that are parallel to the water surface and are equally spaced. Ordinates are defined as vertical surfaces that are perpendicular to the ship axis and are equally spaced.

The ship length is divided into ten intervals of equal length, each station is to be an ordinate, where ordinate 0 is at the A.P. point (see Figure 5.2.1), and ordinate 10 is at the F.P. point.

5.3 The Test Case

5.3.1 Preliminary remarks

As exact solutions of complicated fully three-dimensional turbulent flows are difficult to find, if not impossible, assessment of the accuracy of the present procedure is possible only through the comparison with measurements.



F.P. (Forward perpendicular): This is represented by a line which is $\perp$ to the intersection of the designed load WL with the forward side of the stern.

A.P. (After perpendicular): This is represented by a line which is $\perp$ to the intersection of the after edge of the rudder-post with the designed load WL.

L_{pp} (Length between perpendiculars);

L.W.L. (Length on the designed load WL)

MID-SHIP (Mid-way the F.P and A.P)

Figure 5.3.1 Definitions of common terms used in the thesis.

It is, therefore, the purpose of this section to find an appropriate set of measurements, and to present it.

Measurements of flow around ships are not few. Literature provides many types of measurements. The selection of any of these measurements to be our test case depends mainly on two major points:

- (1) Applicability from the prediction point of view, e.g. to have no regions of recirculation anywhere on the hull; this is one of the limitations of the current procedure.
- (2) Comprehensiveness, i.e. the measurements should be available at many stations, and at many points, so as to help in assessing and understanding the different factors affecting the predictions.

In order to satisfy the above mentioned points, only very few measurements will be of any value to the assessment of the prediction procedures. At the time when the present prediction procedure was being developed, the author knew of two sets of measurements which satisfy the requirements stated above. They are due to Larsson (1974c), and Namimatsu and Muraoka (1974). Since then, three more comprehensive

measurements have been reported, those due to Hauke (1974), Sachdeva and Preston (1976) and Hoffman (1976).

The measurements used in the present investigations are the ones due to Namimatsu and Muraoka (1974).

5.3.2 The test model and the measurements

(a) The hull

Namimatsu and Muraoka measured velocities in the boundary layer, and in the wake near the stern, on a 7-meter ship model. The shape of the hull of this ship model is exactly similar to a 300m long, 200,000 DWT tanker. The purpose of their measurements was to investigate the mechanism and growing process of the bilge vortex, and to find similarities between the actual ship and the model. The model had the following details:

Length (L_{pp})	= 7m
Breadth (b)	= 1.167m
Mean draught (d_m)	= 0.443m
Block coefficient (c_B)	= 0.83

where the length is defined as the length between the perpendiculars (see Figure 5.3.1); the breadth is defined as the width at mid-ship; the mean draught is defined as the depth taken at mid-ship also; and the

block coefficient is defined as the ratio between the volume of the water displaced by the ship to the volume of the rectangular parallel-piped that would have the same dimensions as the maximum dimensions of the ship. The model is made of urethane foam finished with lacquer paint. The shape of the hull is given approximately in Figure 5.3.2 (due to the confidentiality of the hull geometry, Figure 5.3.2 was made slightly out of proportion). The figure shows the shape of the hull at different longitudinal stations denoted by ordinates. The figure shows only half the hull at any of those ordinate stations; the other half is a mirror image of the one shown in the figure. The ordinates shown in the figure are $\frac{1}{2}$, $\frac{3}{4}$, 1, $1\frac{1}{2}$, 2, $2\frac{1}{2}$, 3, $3\frac{1}{2}$. Ordinates 4 to 6 have the same shapes as ordinate $3\frac{1}{2}$.

(b) The measurements

Measurements of velocities in the boundary layer and in the wake near the stern were made, by means of a three-hole pitot tube, at different axial stations. The pitot tube was traversed along perpendiculars to the hull at some nodes* at each station; the total measurements were made at 50 measuring nodes, and at 9 different

* A node is a point on a surface in the space.

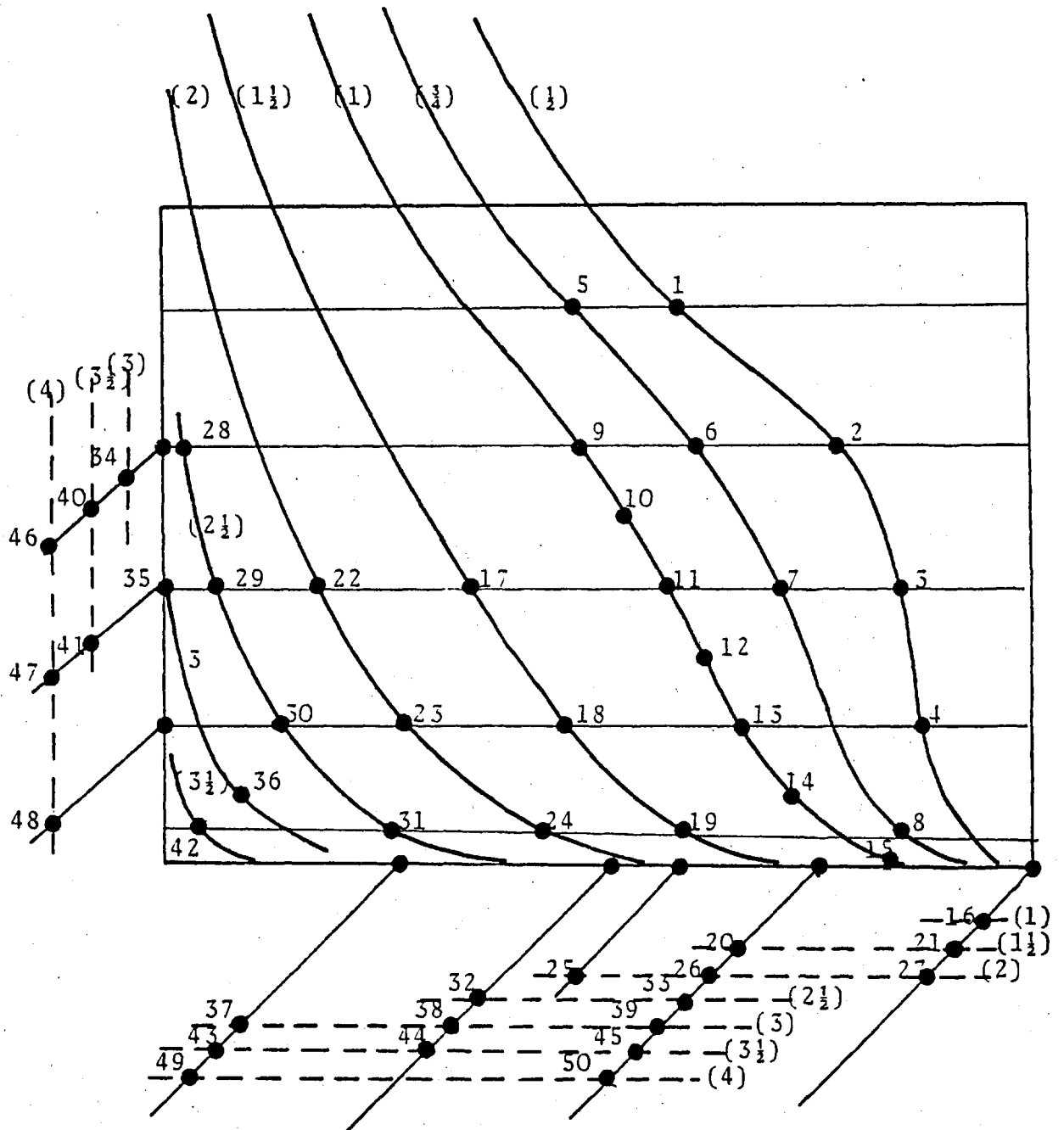


Figure 5.3.2 The ship plan

() ordinate

• measuring nodes

axial stations. Figure 5.3.2 shows the location of those measuring nodes at the respective ordinates. The pitot tube was driven normal to the hull, at the measuring points, by means of a pulse motor whose positioning accuracy was .02mm. The measurements of the vectorial velocities were made on the starboard side.

The authors found that the distribution of velocities and flow direction differed greatly depending on the points measured, but no such flow as separation "backward" was observed. They found that the thickness of the boundary layer around the concave part of the hull was thick near the mid-ship; while at the keel the boundary layer was found to be thin.

5.3.3 Computational details

(a) The hull

As mentioned earlier (Section 5.2) it has been customary, in the field of naval architecture, to define the hull geometry in the form of tabulated values of the width at different water levels (horizontal planes passing through the hull) for different axial stations. To obtain the hull radii at stations where no tabulated values were given, the following procedure was devised and employed:

- (1) Interpolate the tabulated values in the longitudinal direction to obtain the hull width at the required ζ -stations, (a parabolic interpolation technique with respect to z was used).
- (2) Transform the tabulated values of the hull-dimension from the cartesian coordinates to the cylindrical-polar ones.
- (3) Interpolate the hull radii, at any one axial station, in the circumferential direction to obtain the hull radii at the required ξ -stations, (a cubic-spline interpolation technique with respect to θ was used).
- (4) Calculate the non-orthogonal geometric quantities such as G and F functions.

While applying the above procedure to calculate the hull's radii, and the other geometric quantities, the above procedure exhibited a difficulty at ordinate of .25, where the procedure failed to produce realistic values for the hull radii; this is due to a very sharp change of the hull shape at this ordinate. Muraoka (1977) indicated that slight change of the hull shape at this station would not be of any major importance to the development of the flow upstream of this section;

therefore, the hull geometry at ordinate .25 was modified slightly to allow the above procedure to be employed. The validity of such an assumption is based only on the experience of Mr Muraoka.

Furthermore, the hull geometry at the keel of ordinate .25 changes very sharply in the axial direction from the full depth to a very small one. This caused an additional instability in the interpolation process, and was overcome by assuming a discontinuity of the surface gradient at that ordinate.

(b) The grid

In the present investigation, the distribution of grid lines in the ξ - and η - directions is 12 x 17. The distributions in the ξ , η directions are as follows:

ξ -direction: 0, .05, .15, .25, .35, .45,
.55, .65, .75, .85, .95, 1.0.

η -direction: 0, .03, .09, .15, .21, .28, .35,
.42, .5, .58, .65, .72, .79,
.85, .91, .97, 1.0.

It can be noticed from the above distribution that, in the circumferential direction, the grid nodes were made to be evenly distributed; while, in the radial

direction, more grid nodes were concentrated near the hull. By doing so, one ensures that more grid nodes are concentrated where there are steep gradients of properties, thus reducing any error that would result from the discretisation of the finite-difference equations.

In the longitudinal direction, more grid nodes were concentrated in regions of steep hull curvature; a total of 34 grid nodes were taken in the ζ -direction. Other computational details such as grid-independence tests, convergence and accuracy of the procedure have been highlighted earlier in Section 3.5.

(c) The outer boundary condition

At the outer boundary of the calculation domain, i.e. the North boundary, boundary conditions are prescribed from a potential-flow solution. The potential-flow solution at the outer boundary was computed and supplied by IHI as a distribution of pressures and velocities over the cylindrical surface surrounding the hull. The axial velocity and the circumferential one were then supplied as boundary conditions to the boundary-layer calculation. The radial velocity was computed from the local continuity of the outer cells; while the pressure was computed so as to cause the stagnation

pressure at this boundary to equal the free-stream one.

5.4 Presentation of Results

Results presented in this section are divided into four groups:

- (a) Cross-stream plots, where axial-velocity contours and cross-stream velocity vectors are plotted at different axial stations defined by its ordinate.
- (b) Circumferential plots, where velocity vectors in a plane of constant ξ are plotted.
- (c) Comparison with measurements, where vectorial velocities measured along a line normal to the hull at some selected points are compared with predicted ones along the same normal.
- (d) Turbulence characteristics, where predicted profiles of the turbulence kinetic energy and length scale along the same normals as (c) are given.

5.4.1 Cross-stream plots

The results presented in this section are for axial velocity contours and cross-stream velocity vectors.

Axial velocity contours and cross-stream velocity vectors are presented in Figures (5.4.1) to (5.4.4) at different axial stations defined by its ordinate; where ordinate = $.1 L_{pp}$. The length ordinate position is calculated as $(L_p - z)/(.1 L_{pp})$, where L_p is the length from the initial section to the AP point. The initial section is defined in this case as the midship.

The contours of the axial velocity are plotted in the form of contours of $\frac{w}{U_\infty}$, where U_∞ is the free-stream velocity of 1.3m/sec. The strength of the cross-flow can be assessed by its relative magnitude with respect to the free-stream velocity; the velocity vectors are plotted as being $U_{r\theta}$, where $U_{r\theta}$ is the resultant velocity of the r- and θ - components. At the bottom right hand side of each of the plots, an arrow is plotted whose magnitude is U_∞ .

5.4.2 Circumferential plots

The results presented in this section are for the vector plots at different circumferential stations. The results are presented in Figures (5.4.5) and (5.4.6). The vector plots are for U_{rz} , where U_{rz} is the resultant velocity of the U_r and U_z components. The plots are given at different ξ -stations of 0., .15, .25, .35, .45, .55, .65, .75, .85, .95.

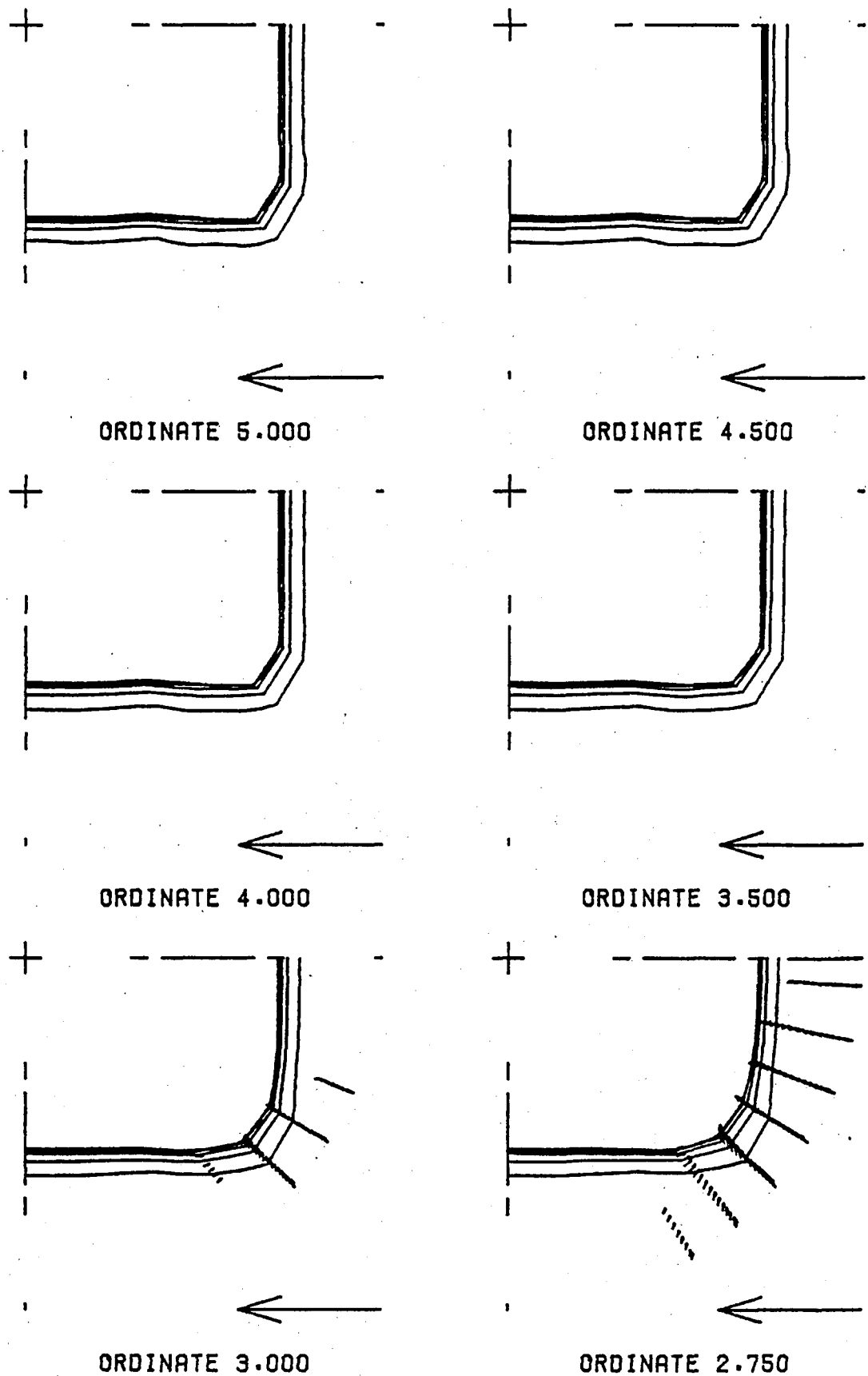


Figure 5.4.1 Cross-stream plots - axial velocity contours and cross-stream velocity vectors (Ordinate 5 - ordinate 2.75)

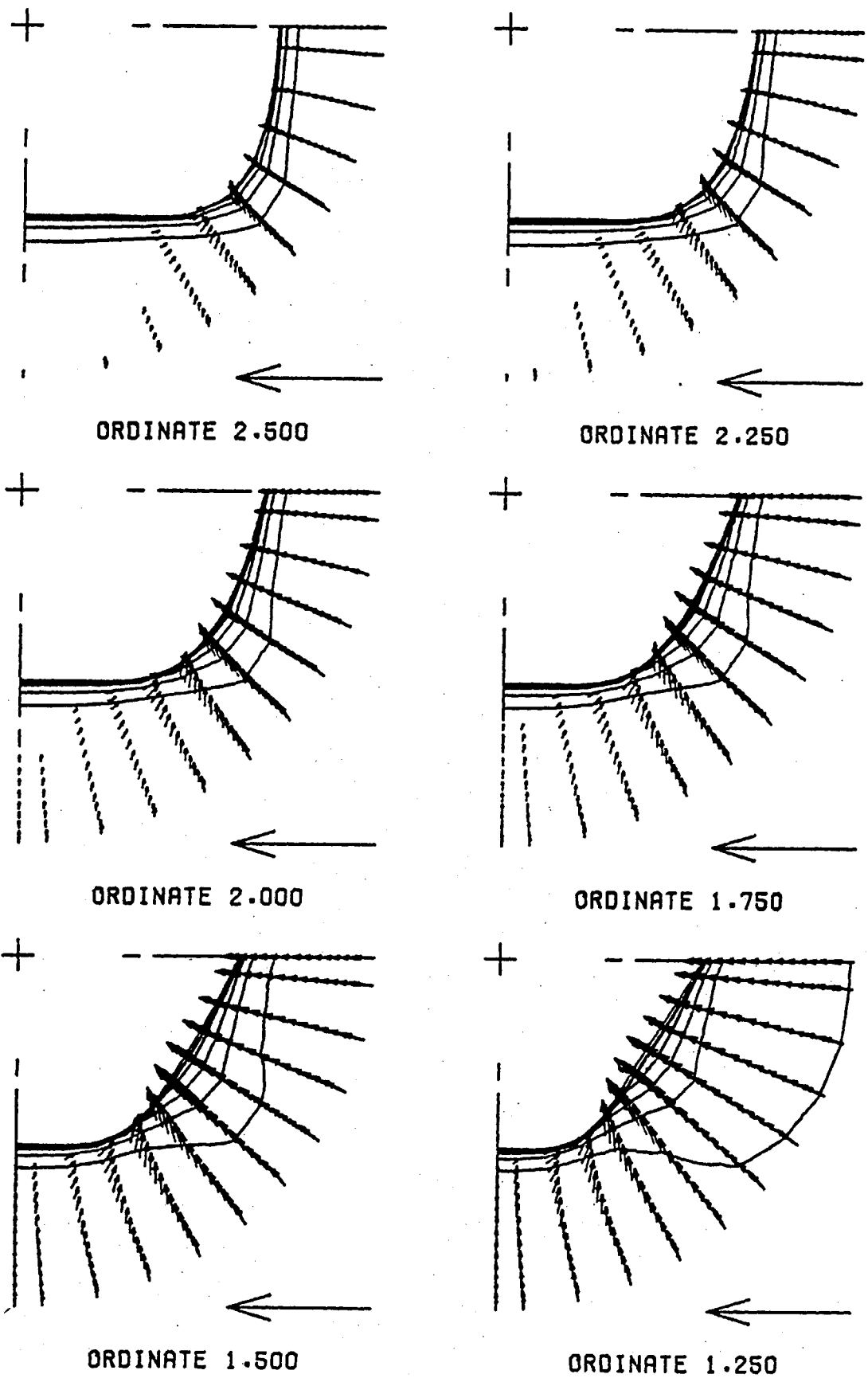


Figure 5.4.2 Cross-stream plots - axial velocity contours and cross-stream velocity vectors (Ordinate 2.5 - ordinate 1.25)

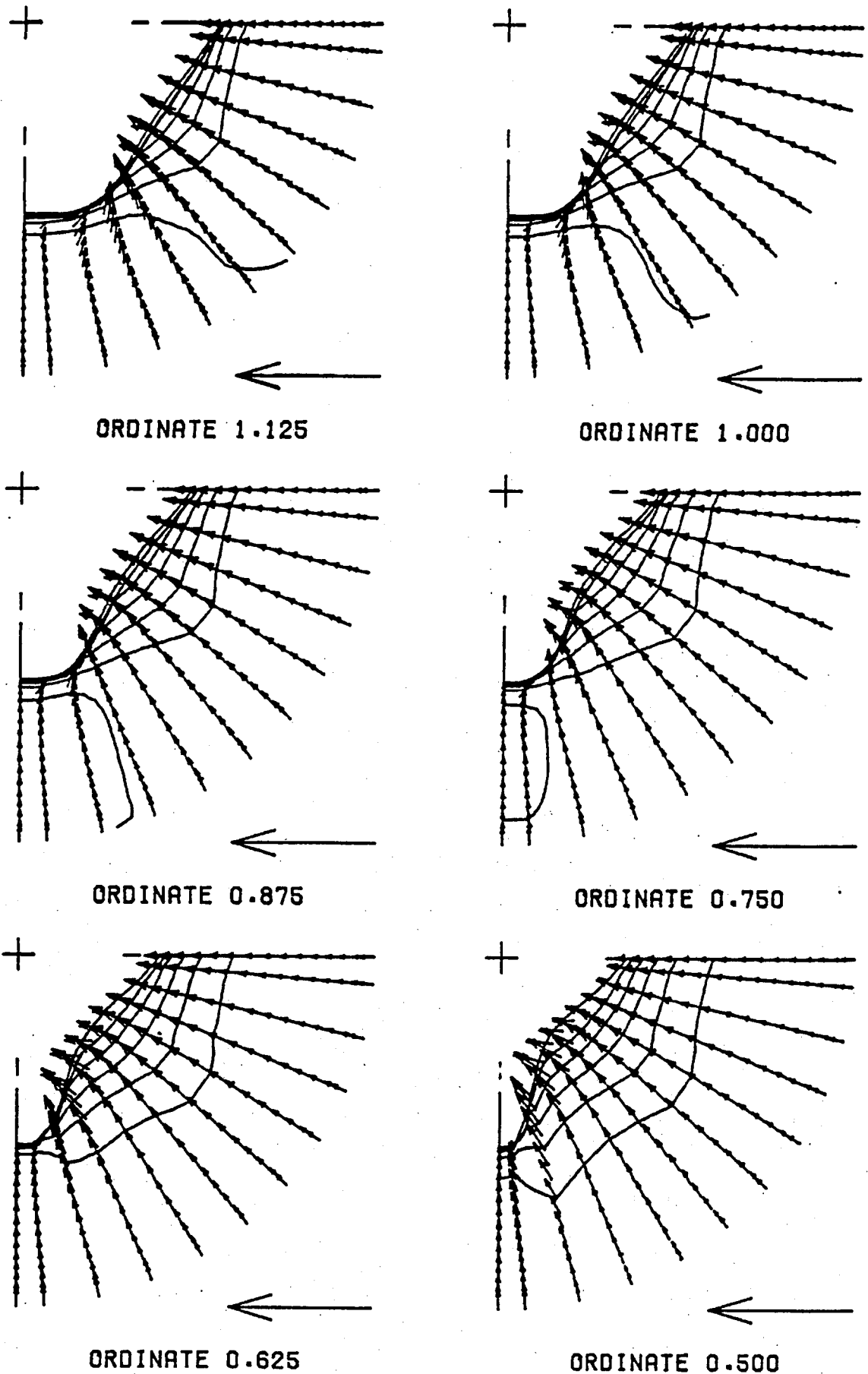


Figure 5.4.3 Cross-stream plots - axial velocity contours and cross-stream velocity vectors (Ordinate 1.125 - ordinate .5)

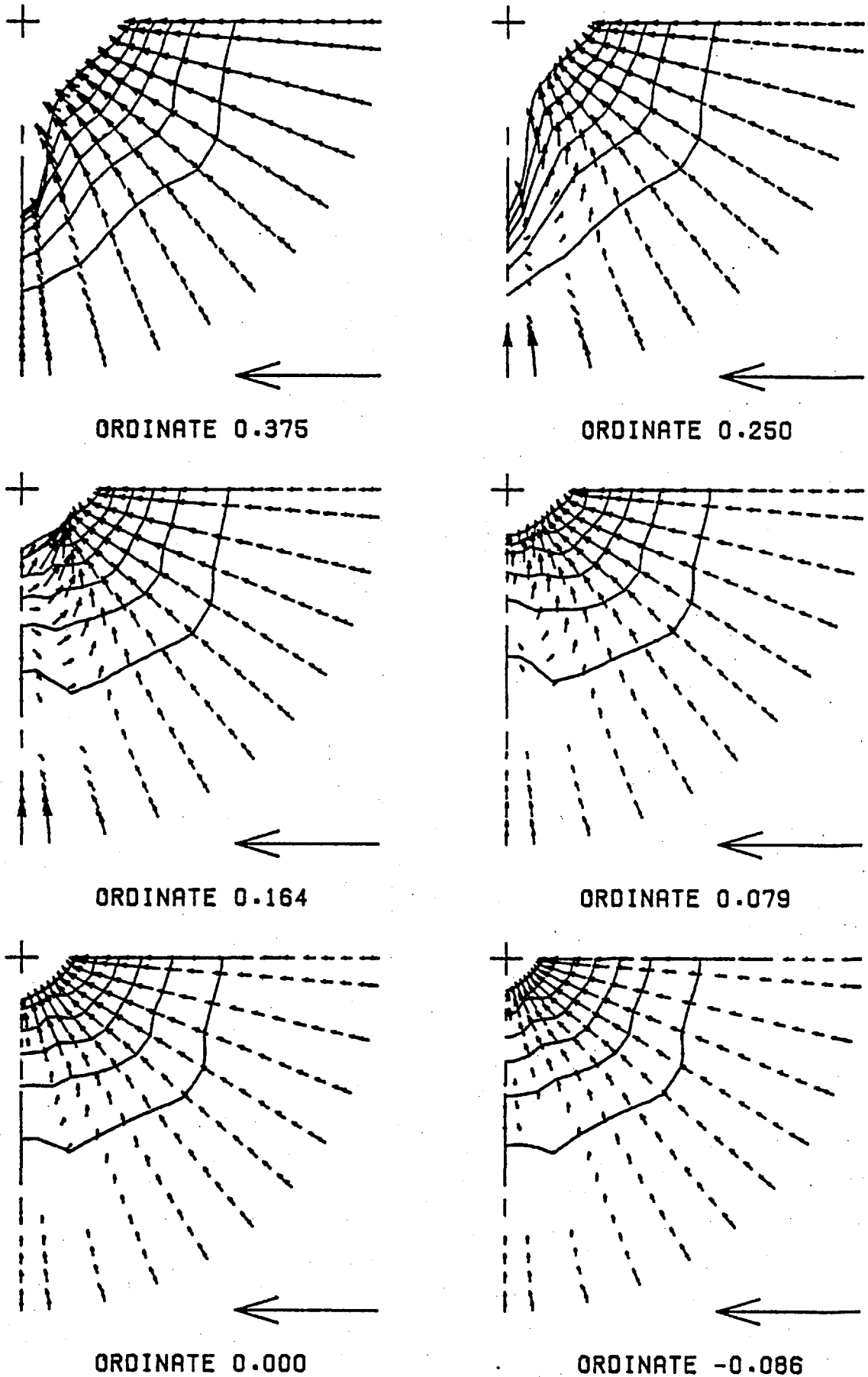


Figure 5.4.4 Cross-stream plots - axial velocity contours and cross-stream velocity vectors (Ordinate .375 - ordinate -.086)

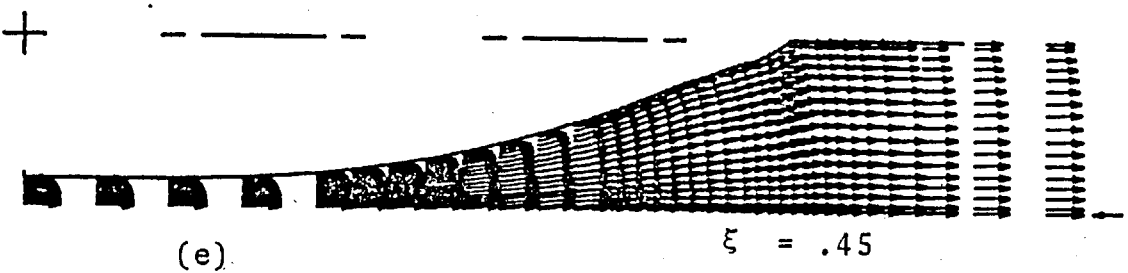
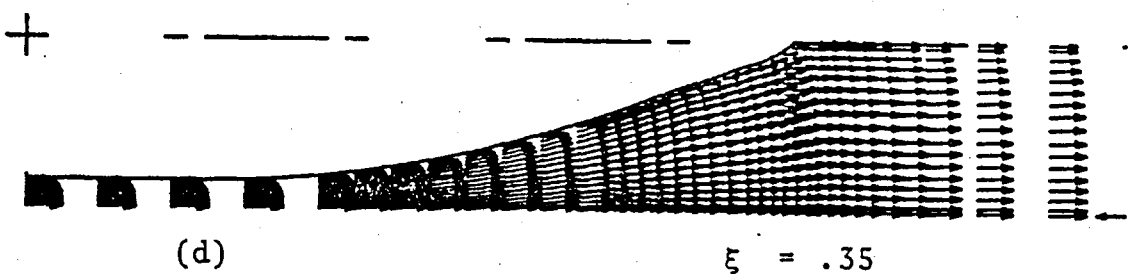
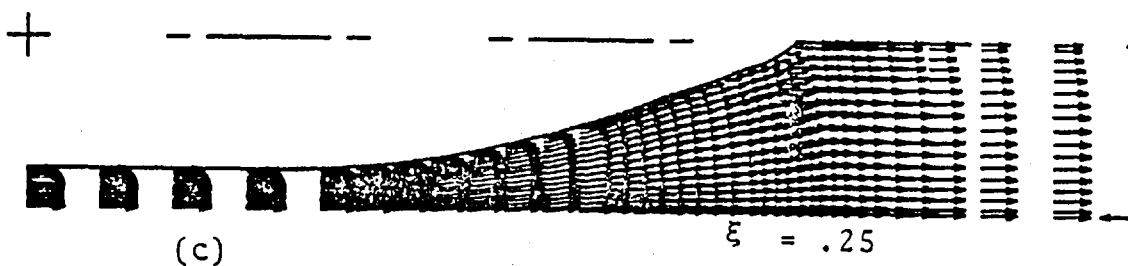
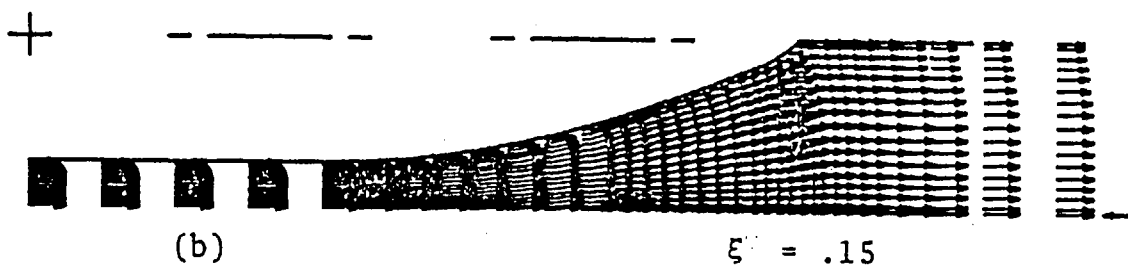
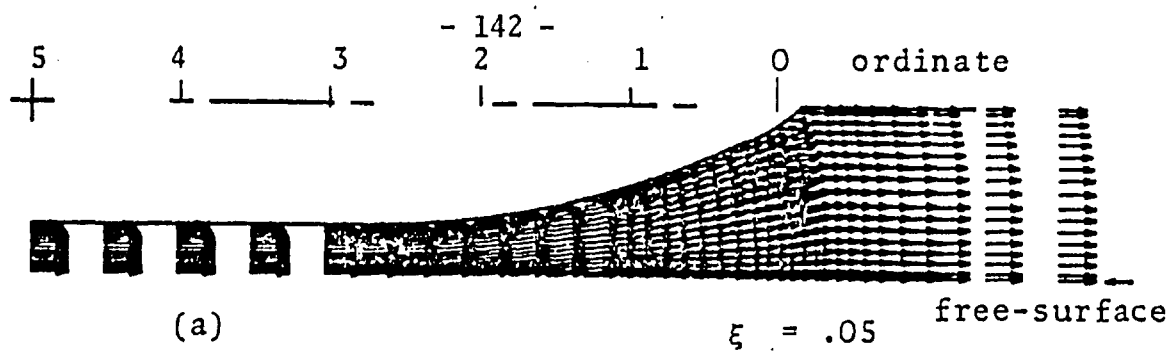


Figure 5.4.5 Circumferential plots - velocity vectors
in constant
stations ($\xi = .05$ to $.45$)

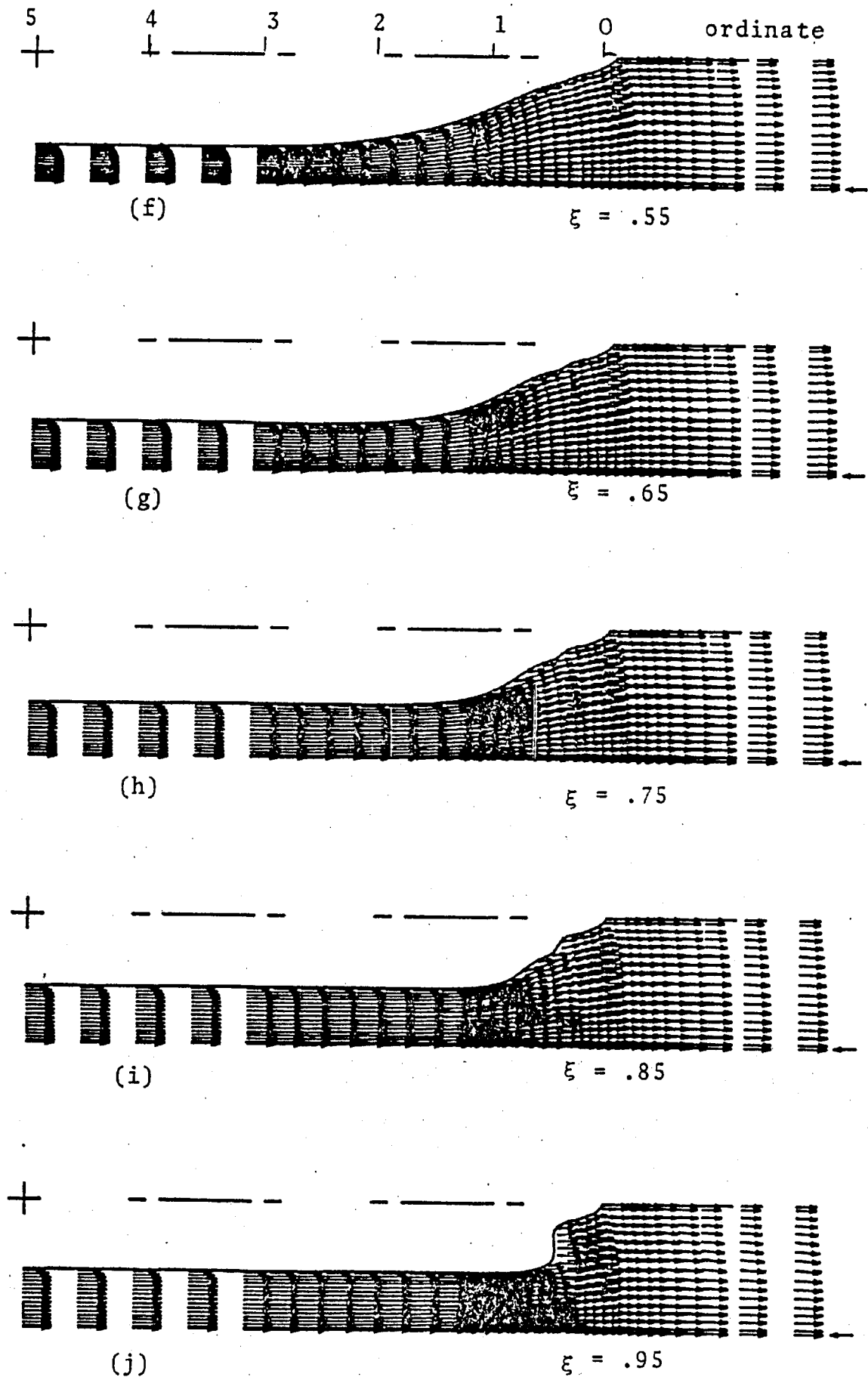


Figure 5.4.6 Circumferential plots - velocity vectors in constant stations ($\xi = .55$ to $.95$)

5.4.3 Comparison with measurements

The third and most important type of plots in this chapter are the 'normal plots', where a comparison of the measured and predicted total velocity, defined as $\sqrt{u^2+v^2+w^2}$, along lines normal* to the hull at some 50 nodes on the hull, is made.

The comparisons of the prediction with the measurements along normals to the hull at different axial stations denoted by its ordinate are presented in Figures 5.4.7 to 5.4.16. The figures show the distribution of w , defined as the total velocity against y which is the distance from the hull along the normals at the measuring nodes. These figures are best understood in connection with Figure 5.3.2, where the figure shows the location of the measuring nodes on the surface.

A comparison between the predicted and estimated (from potential flow solution) pressure distributions at the free-boundary is presented at Figure 5.4.17. The figure presents the pressure distribution along the longitudinal

* As normals to the hull surface do not always fall in planes of constant z , particularly near the stern or along lines of constant ξ , interpolation was necessarily made on the predictions to find the appropriate distribution of the predicted vectorial velocity on the perpendicular to the hull at the surface nodes. The interpolation scheme used was linear.

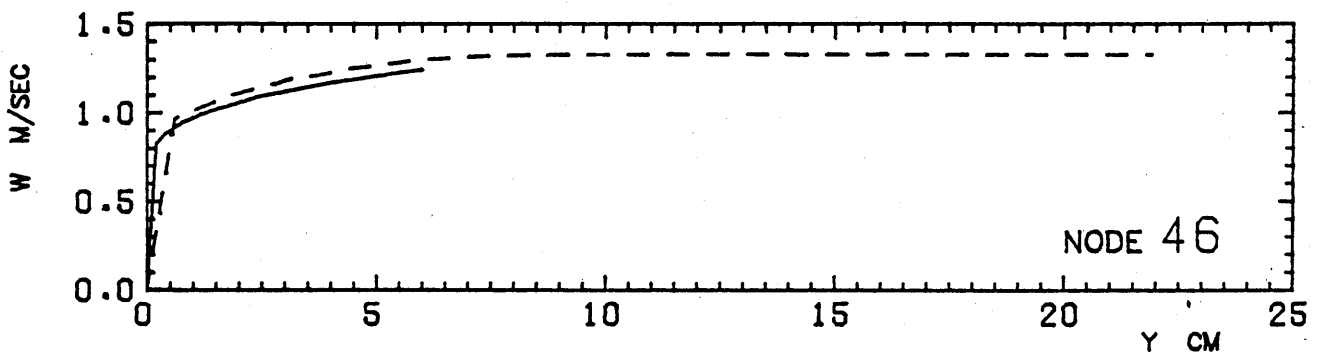
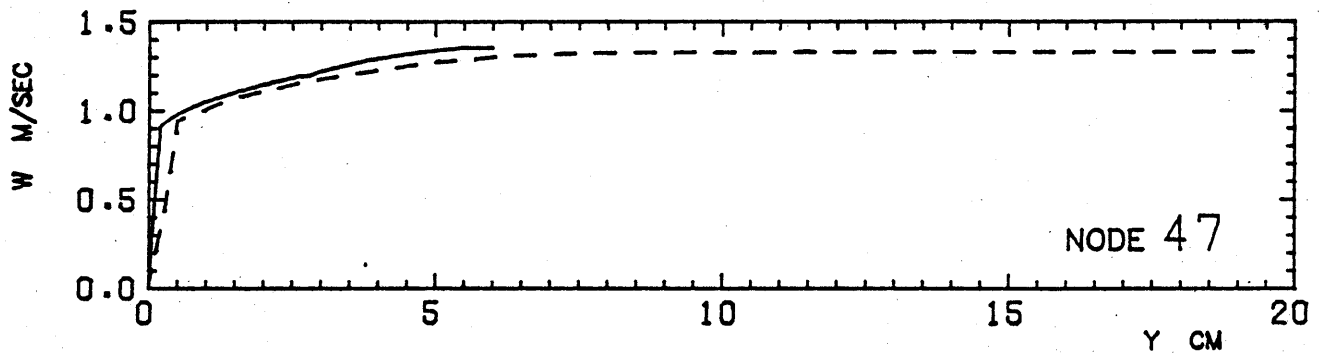
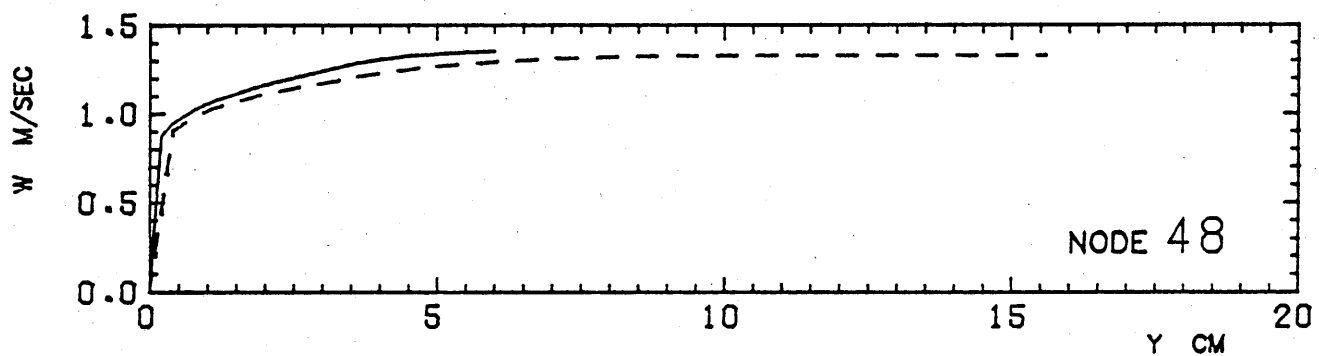
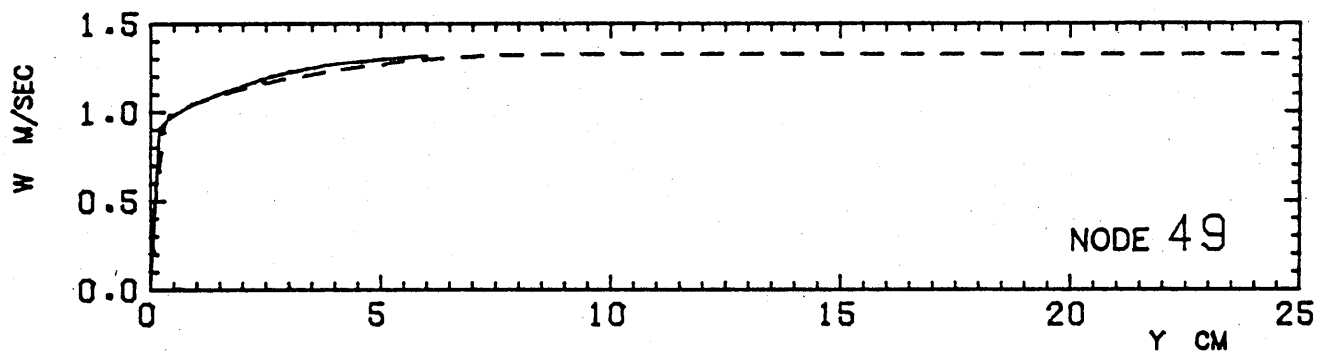
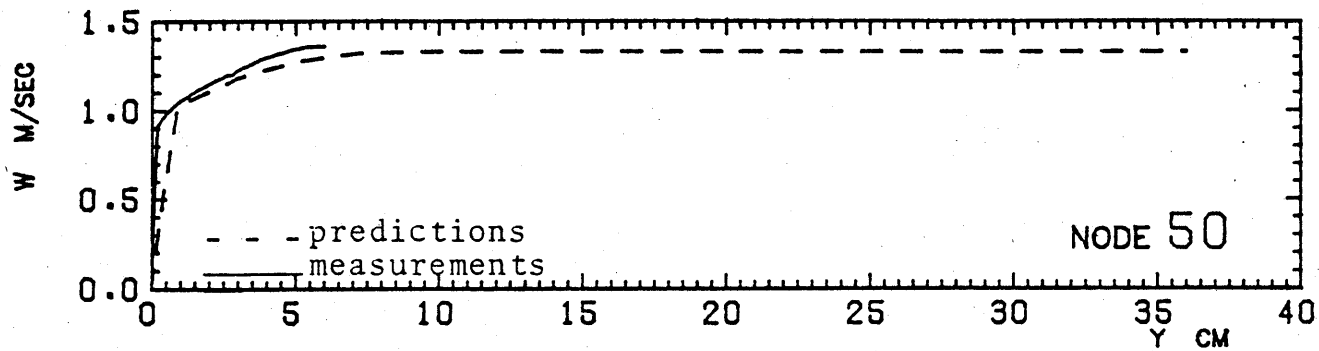


Figure 5.4.7 Comparison of measurements with predictions
(Ordinate 4)

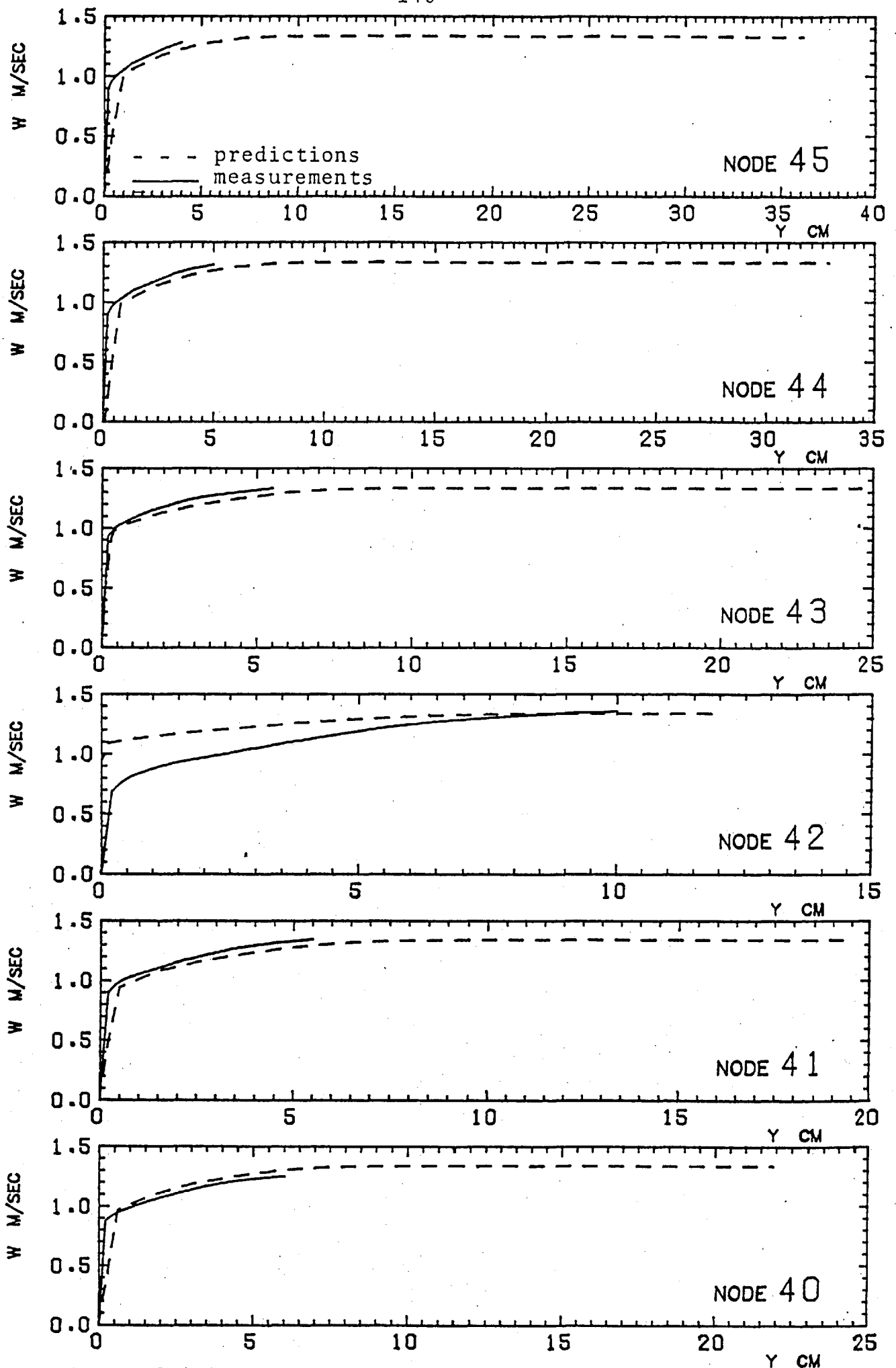


Figure 5.4.8 Comparison of measurements with predictions
(Ordinate 3.5)

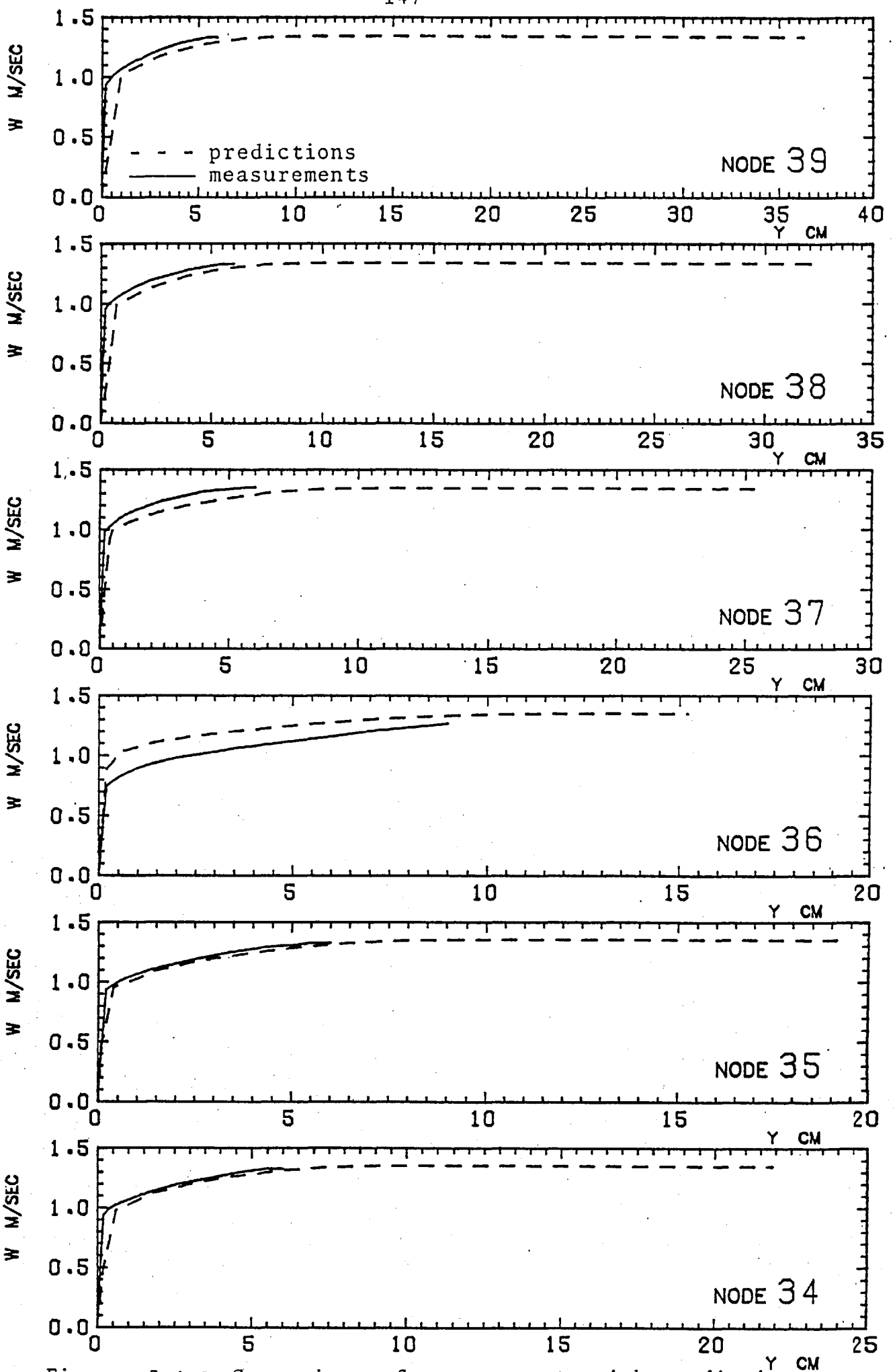


Figure 5.4.9 Comparison of measurements with predictions
(Ordinate 3)

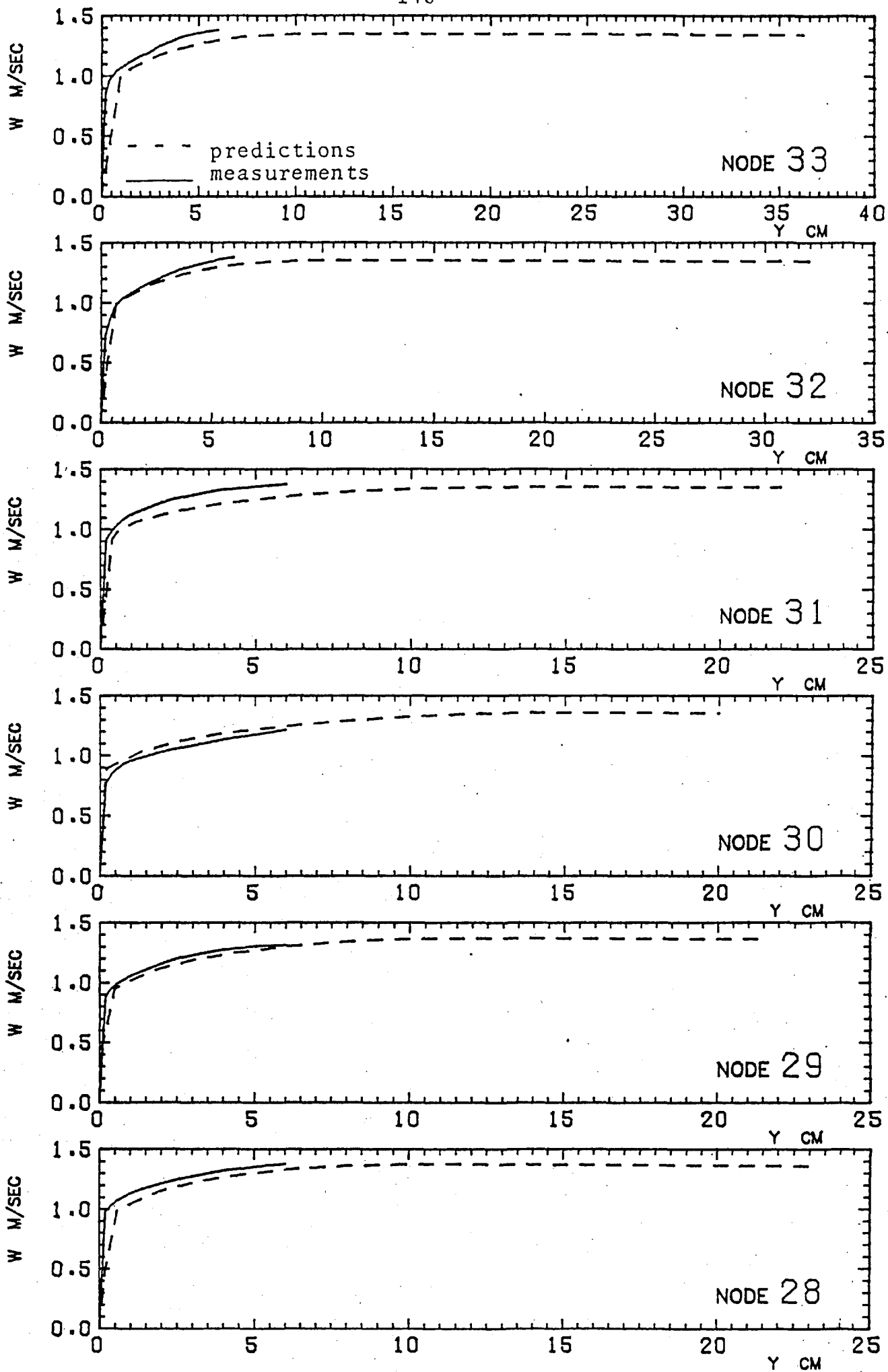


Figure 5.4.10 Comparison of measurements with predictions
(Ordinate 2.5)

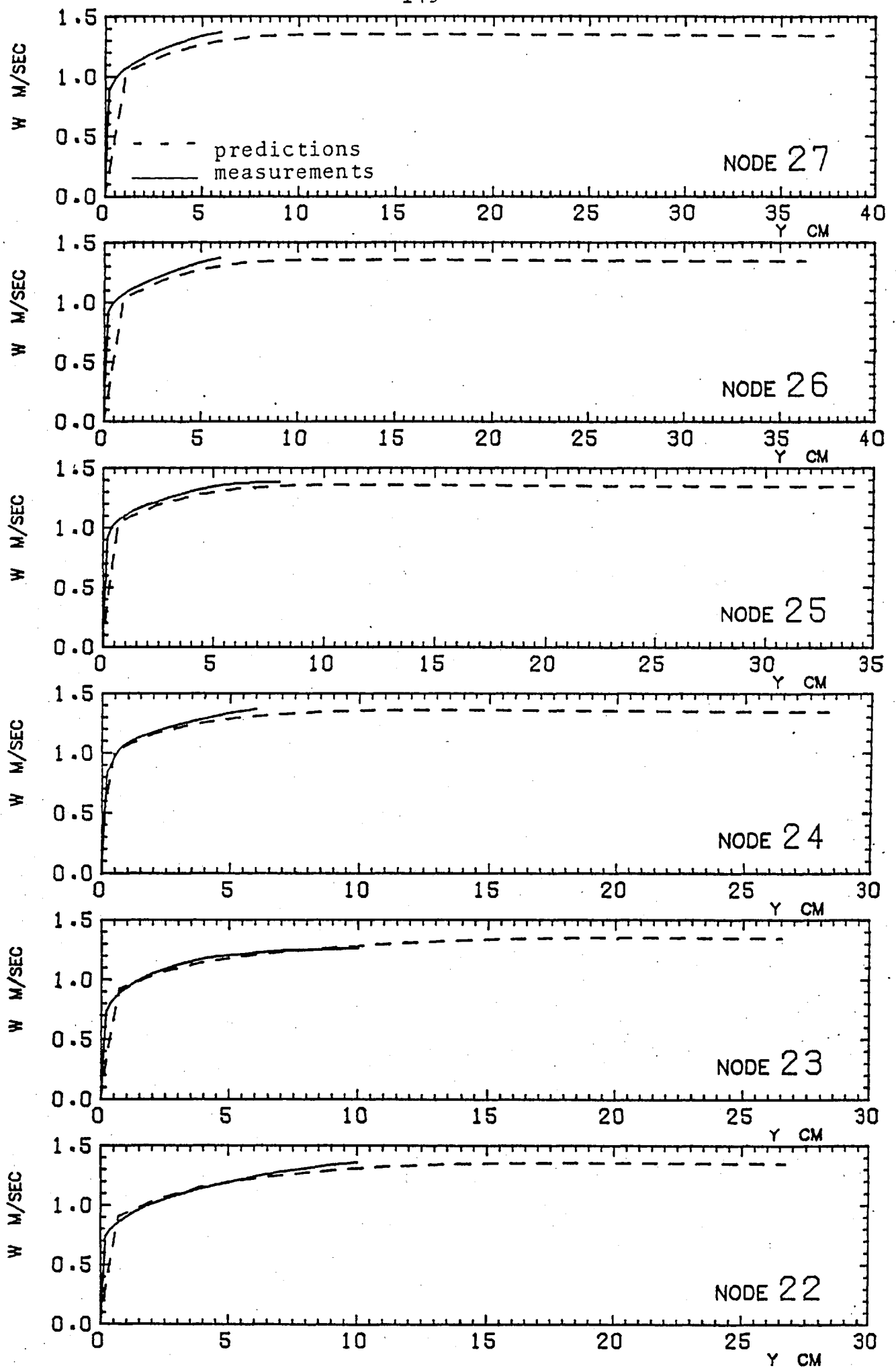


Figure 5.4.11 Comparison of measurements with predictions
(Ordinate 2)

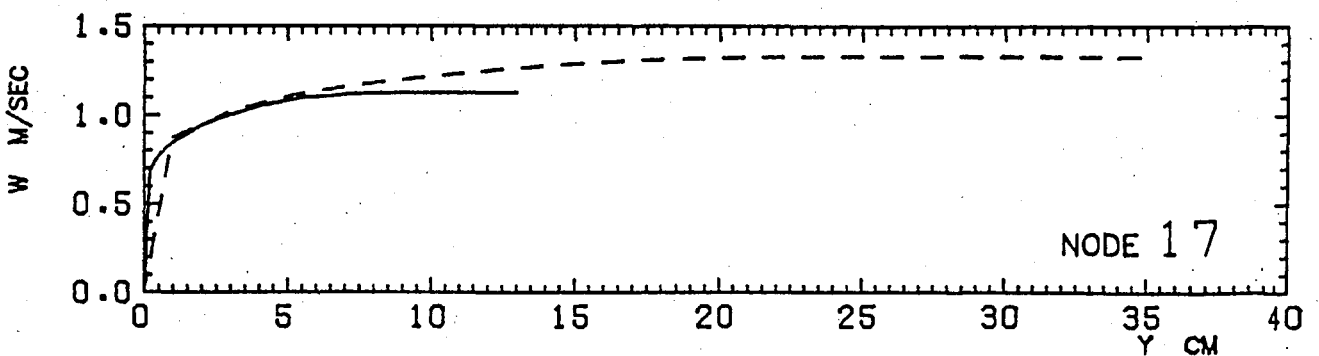
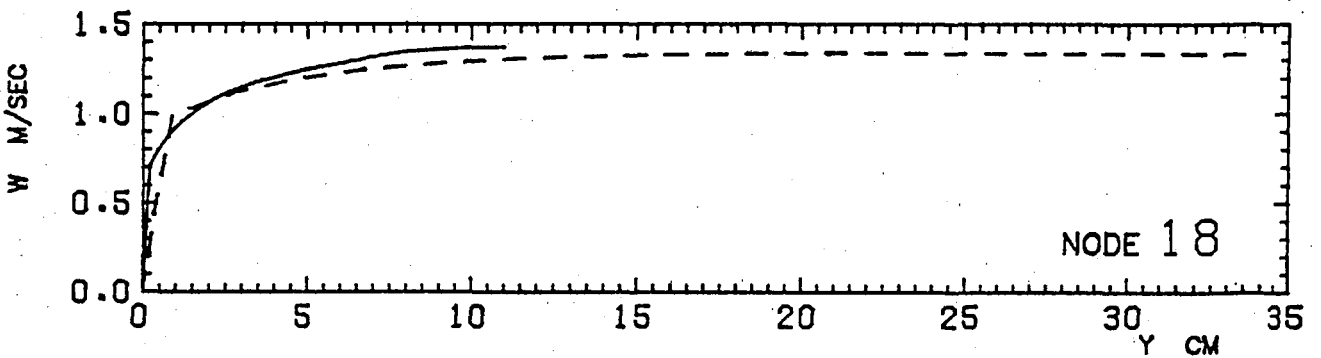
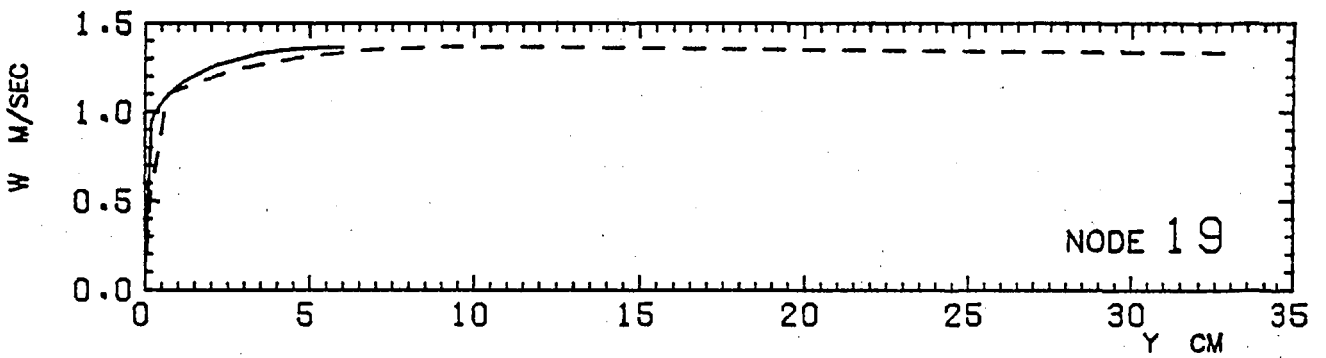
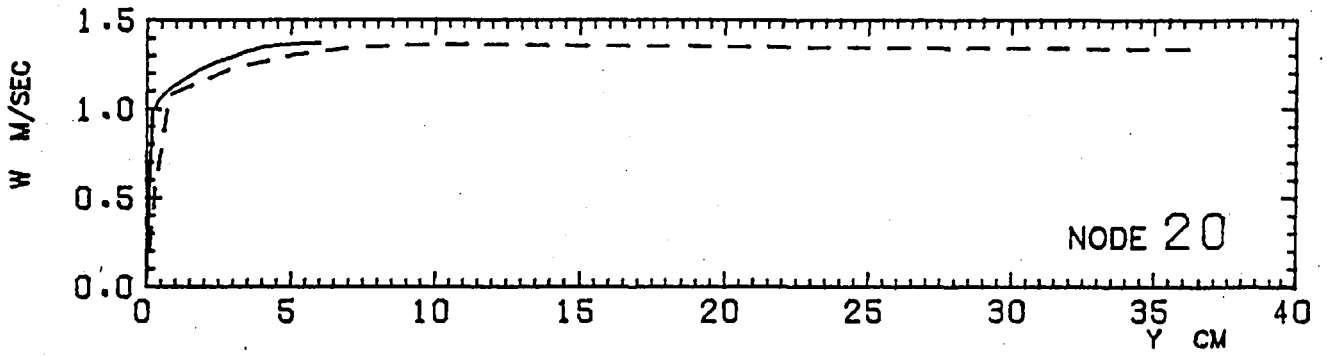
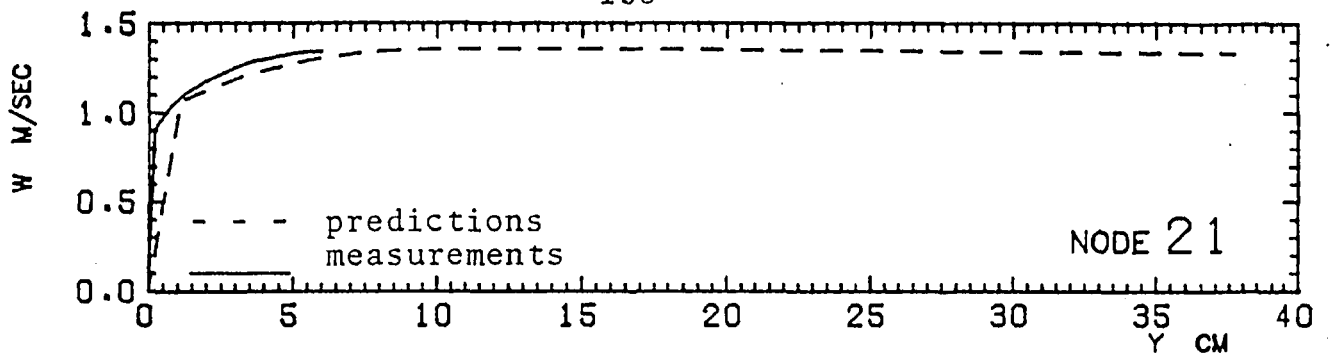


Figure 5.4.12 Comparison of measurements with predictions
(Ordinate 1.5)

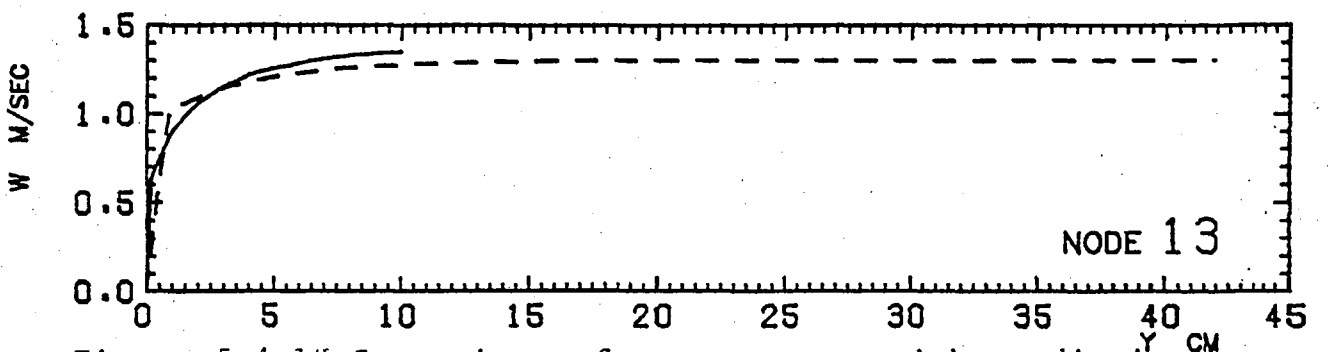
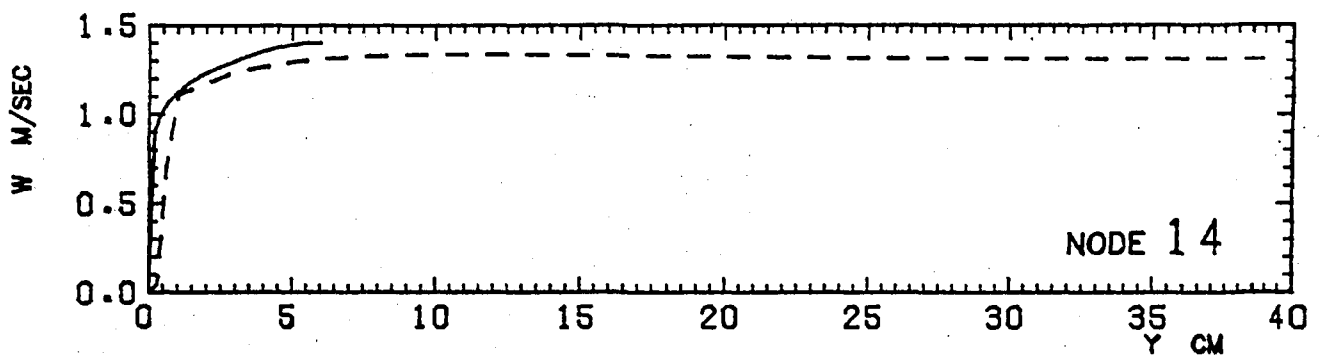
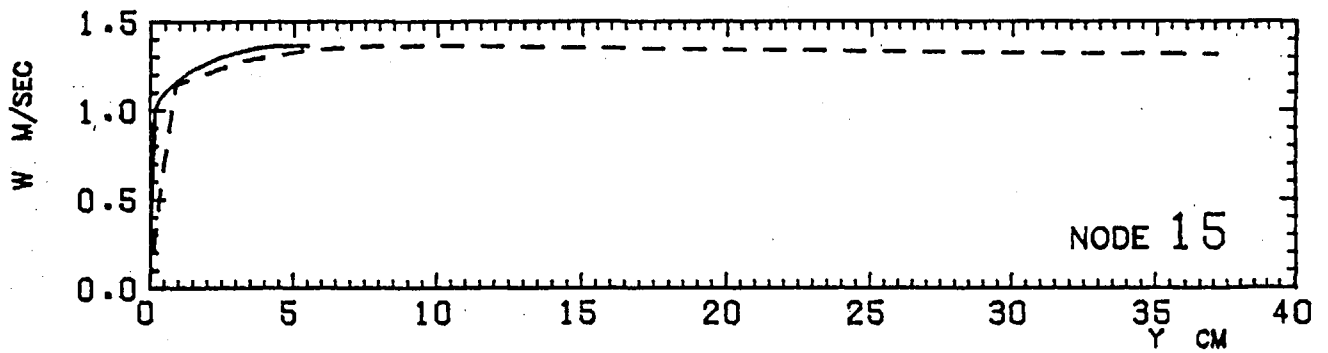
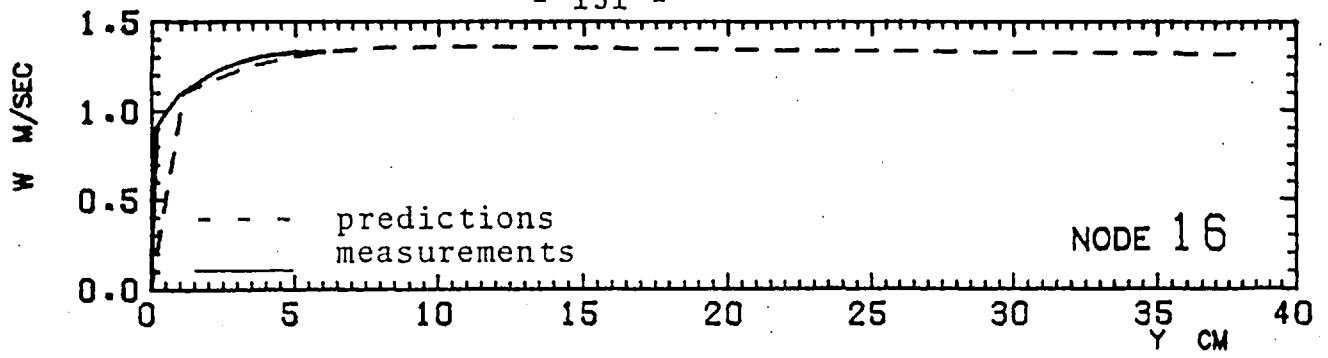


Figure 5.4.13 Comparison of measurements with predictions
(Ordinate 1.5 (Continue))

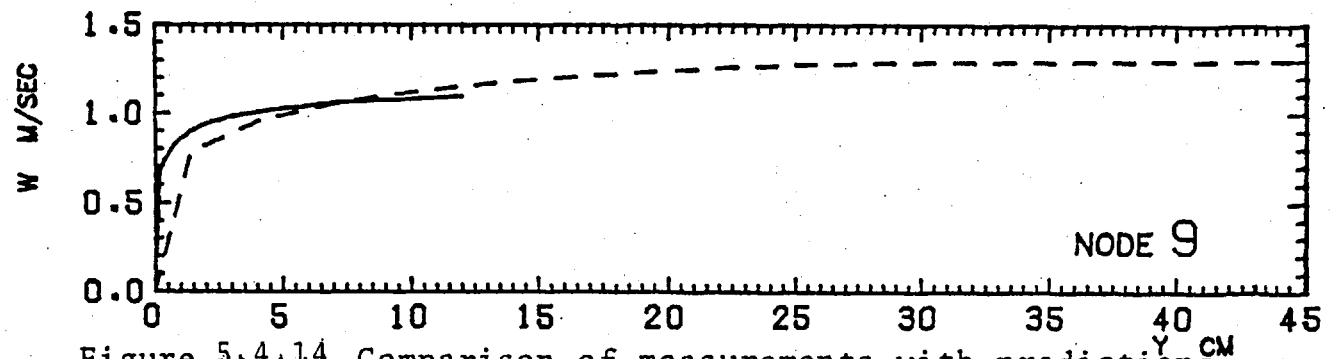
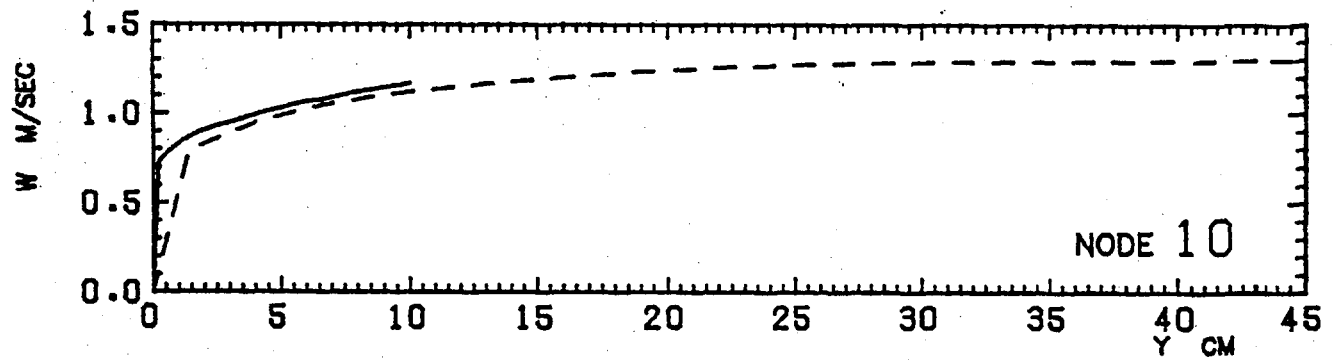
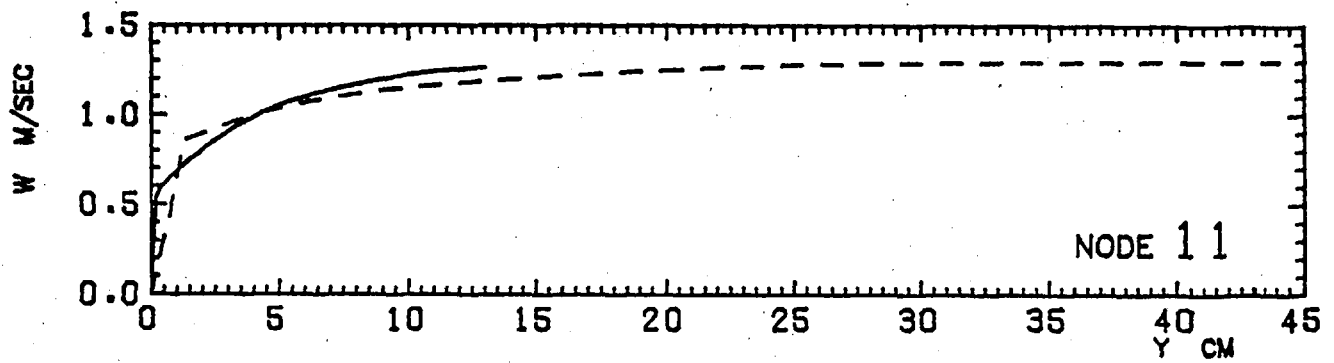
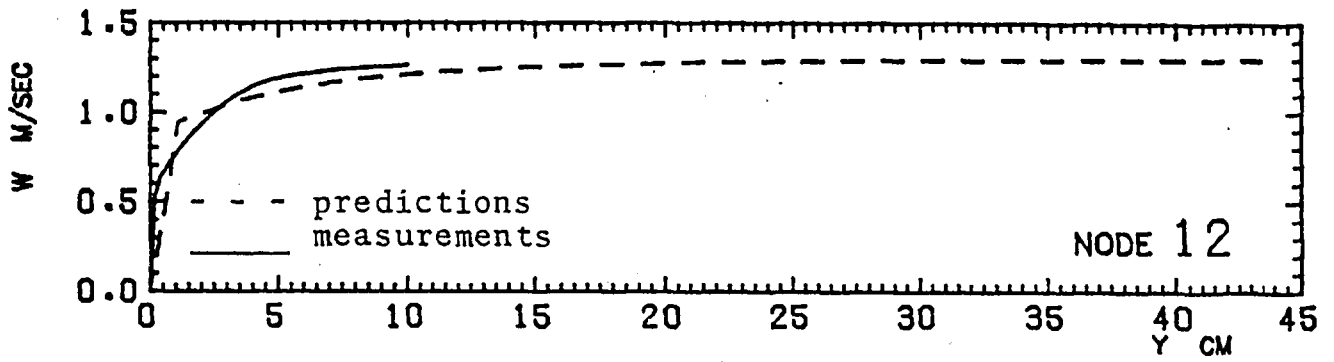


Figure 5.4.14 Comparison of measurements with predictions
(Ordinate 1)

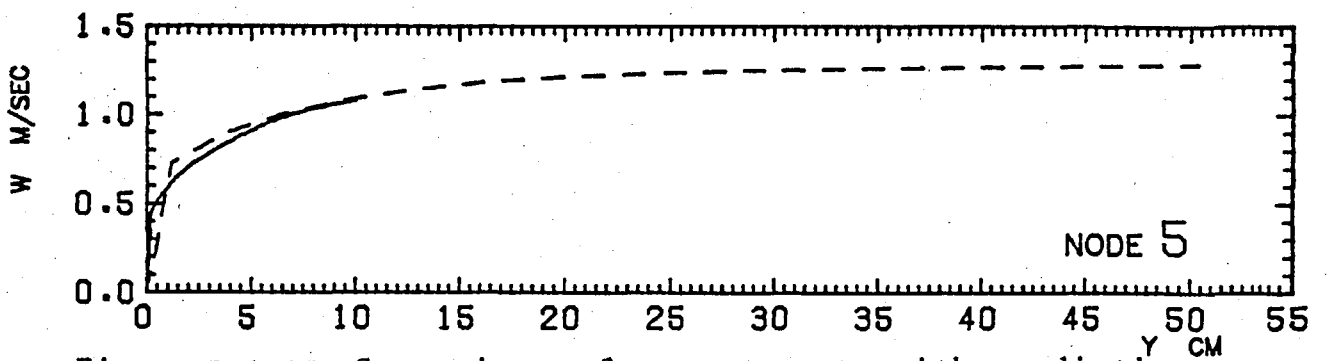
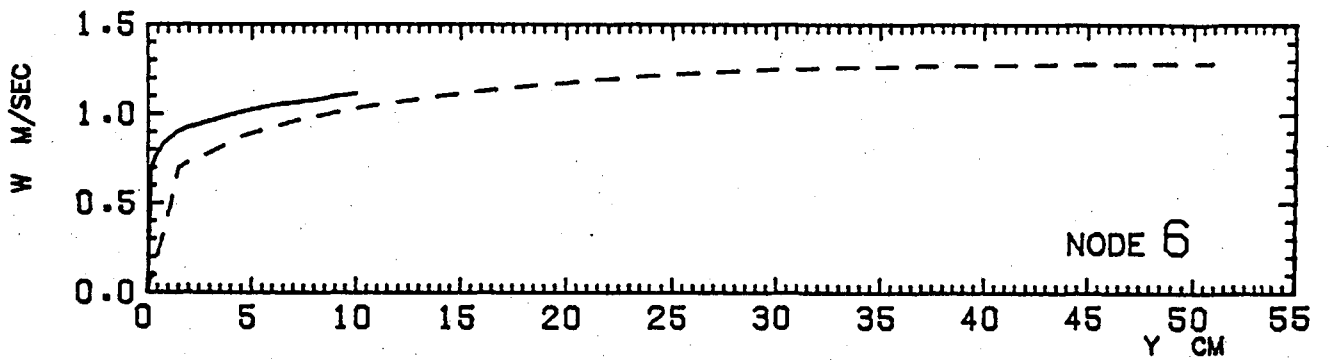
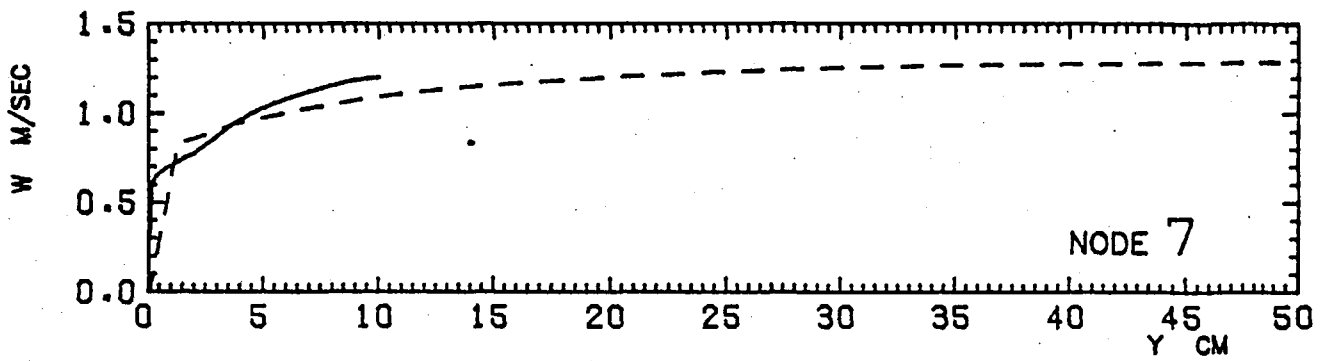
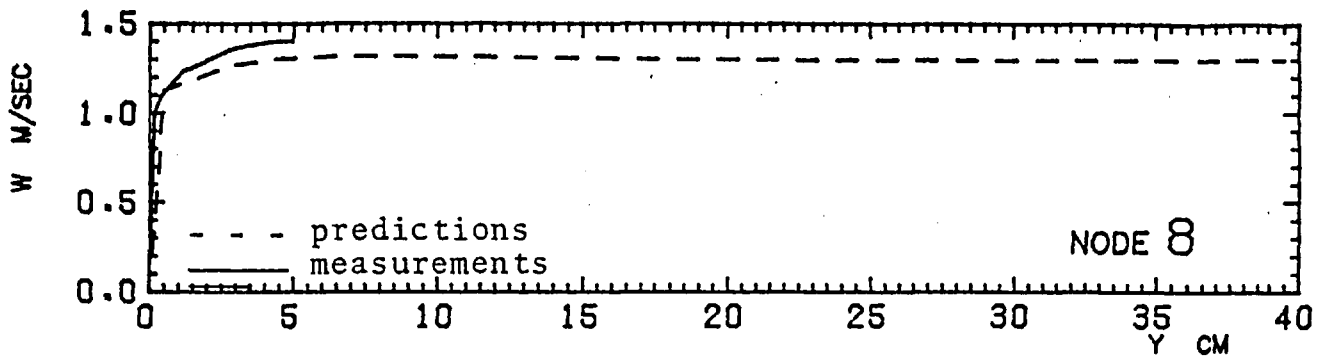


Figure 5.4.15 Comparison of measurements with predictions
(Ordinate .75)

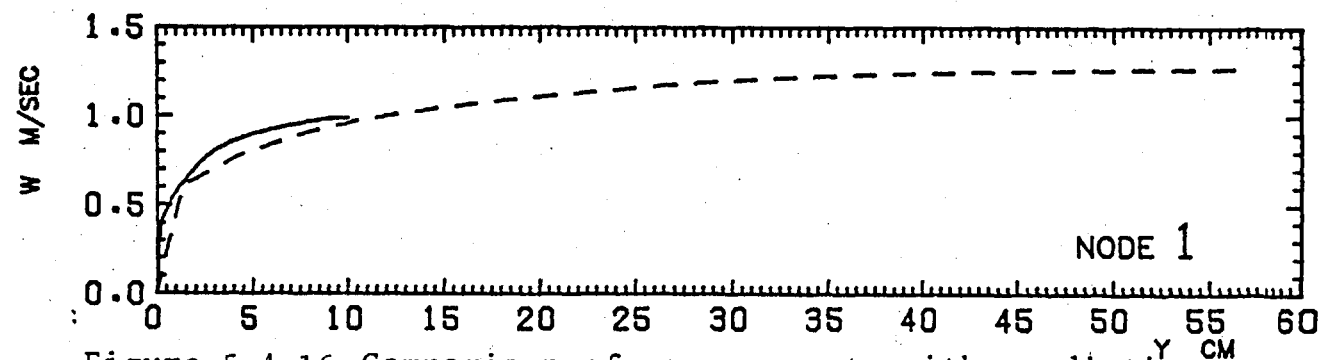
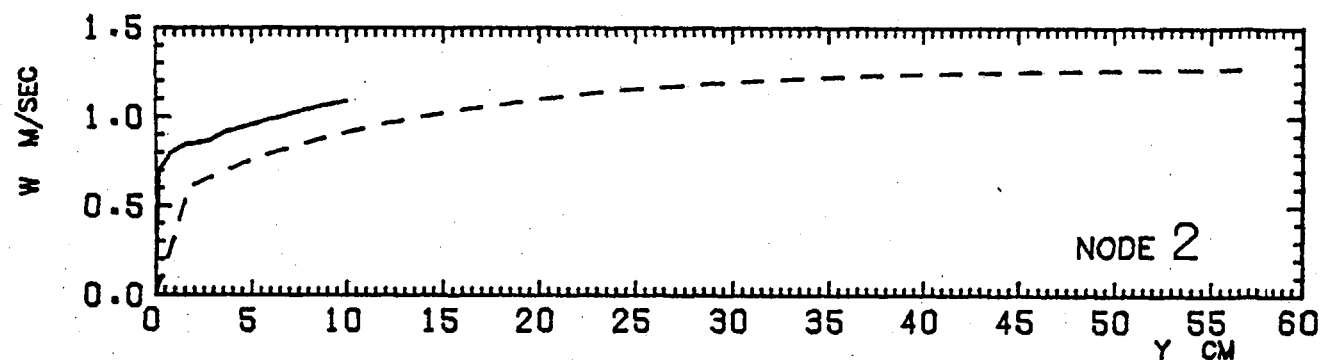
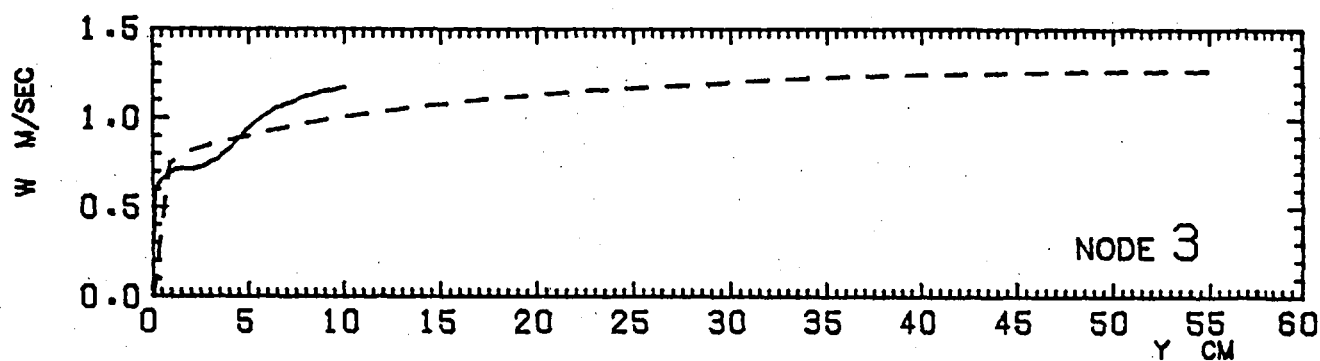
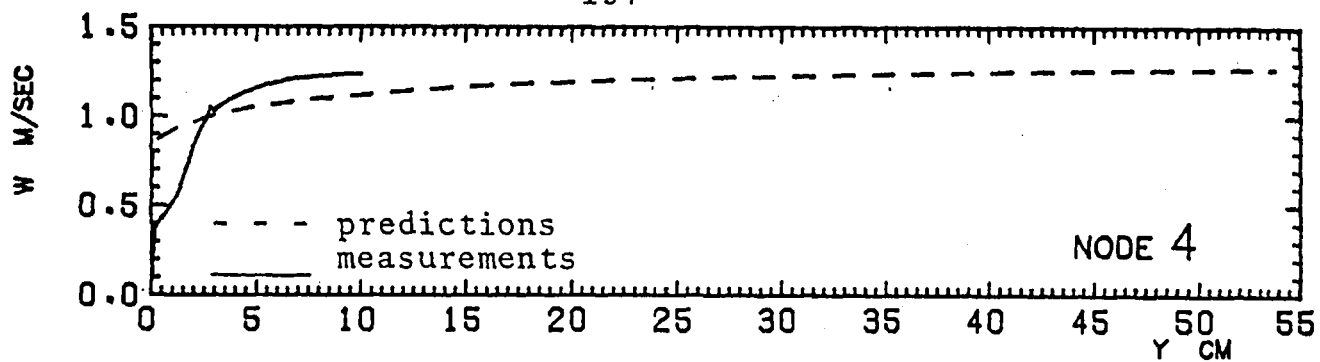


Figure 5.4.16 Comparison of measurements with predictions
(Ordinate .5)

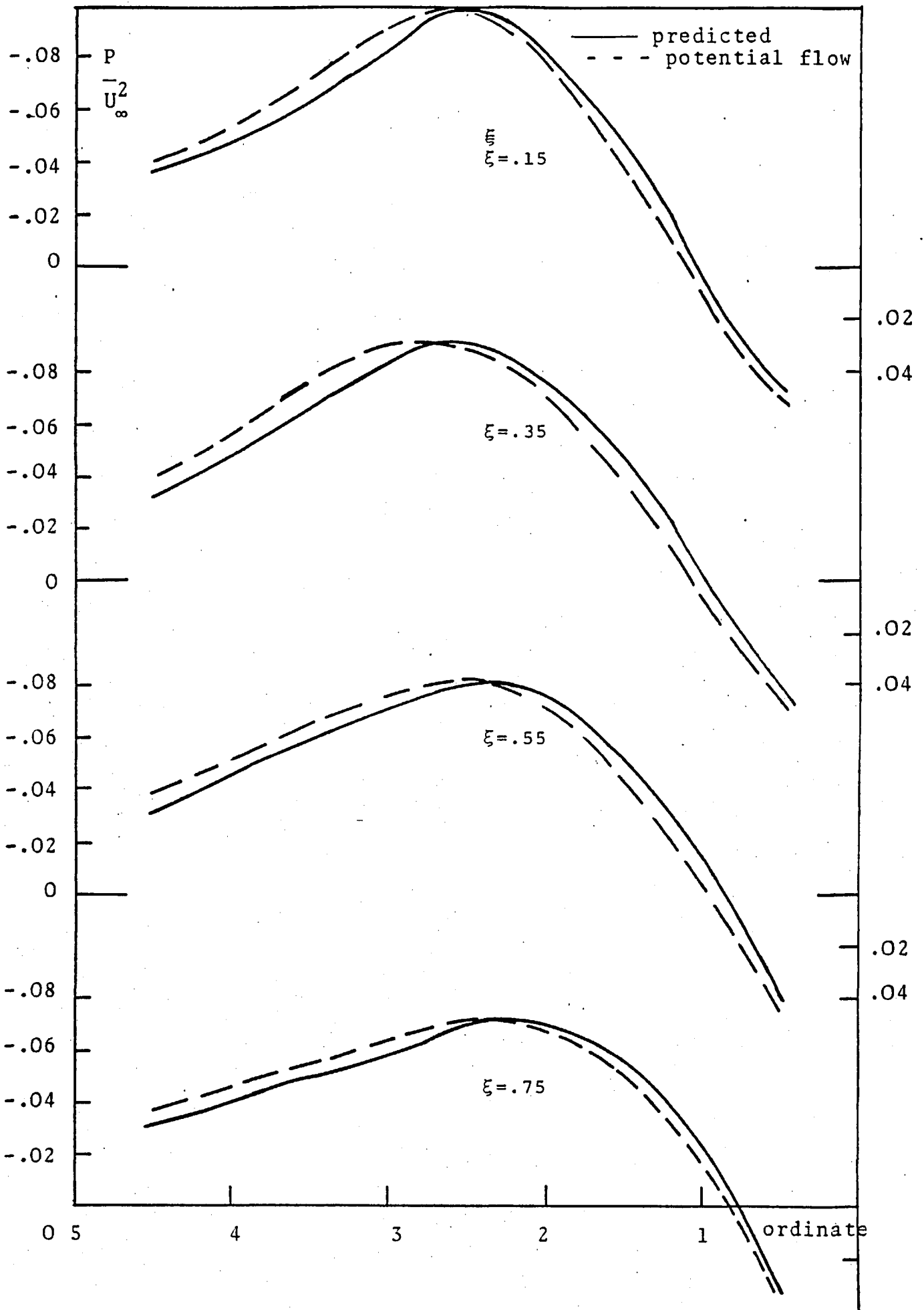


Figure 5.4.17 Comparison between the predicted and potential flow pressure distribution at the free-boundary

direction at constant circumferential stations
($\xi = \text{constant}$).

5.4.4 Turbulence characteristics

Results presented under this title are for the variation of the turbulence kinetic energy level, c_k , and the normalised turbulence length scale, l_n , with distance from the hull at the same fifty nodes reported in the previous section. The various sets of measurements are all very similar, therefore only one set is selected and represented by Figure 5.4.18. The turbulence kinetic energy level, c_k , is defined as $1000k/U_\infty^2$, where k is the kinetic energy of turbulence, and U_∞ is the free-stream velocity. The value of c_k at the inlet section (ordinate 5) was assigned to 4.

The normalised turbulence length scale, l_n , is defined as $1000l/l_{pp}$, where l is the turbulence length scale, and l_{pp} is the ship's length between the perpendiculars.

5.5 Discussion

5.5.1 Cross-stream plots

The cross-stream plots are very good in providing a qualitative look at the growth of the boundary layer. The following points may be observed from Figures 5.4.1 to 5.4.4:

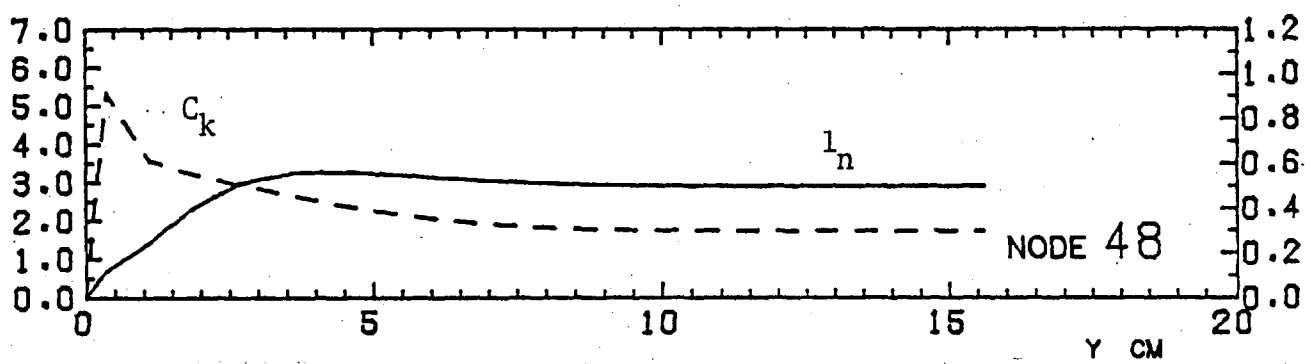


Figure 5.4.18 Turbulence kinetic energy level and length scale profiles (ordinate 4)

- o At the inlet section, ordinate 5, the results show the thin, uniform boundary layer imposed as inlet conditions, and the non-existence of cross-flow as also imposed by the inlet conditions.
- o Downstream of ordinate 5, a fairly uniform boundary-layer growth occurs up to ordinate 3.5. No cross-flow is observed at this station.
- o Immediately downstream, at ordinate 3, the boundary layer in the concave part of the hull starts to thicken, and a pattern of cross-flow starts to appear.
- o At ordinate 2.75 and downstream to ordinate 2, this pattern persists. The boundary layer at the concave part of the hull thickens more and more, and a clear pattern of cross-flow starts to develop. Near the free surface, the flow turns towards the bottom of the ship; at the bottom of the ship, the flow is turning upwards, thus causing the thickening of the boundary layer mentioned above. The low-axial-velocity region in the concave part of the hull surface persists and spreads outwards.
- o Further downstream (i.e. near to the stern region), complex flow patterns develop, the main features

of which are: a high velocity region and thin boundary under the keel; a thick boundary layer and low velocity region near the free surface.

The strength of the cross-flow is at its maximum.

In the observations of Namimatsu and Muraoka (1974) they found that the boundary-layer is thick at the concave part of the hull as early as ordinate 3.5. As the hull is having a constant geometry between ordinates 6 to 3.5, this suggests that such thickening of the boundary layer was already there at midship, and it must have been carried with the upstream flow approaching midship. In any case, such a feature, early thickening of the boundary layer, was not accounted for in the imposed initial condition.

5.5.2 Circumferential plots

The results presented in Section 5.4.2 suggests the following features:-

- o Vector plots are very much similar for all circumferential stations from ordinate 5 down to ordinate 3.
- o Downstream of ordinate 3, the resultant cross-flow velocity vector U_{rz} near the free surface (see Figure 5.4.5(a), (b) and (c)), heads

inward and is nearly parallel to the hull. This is due to the diminishing action of the hull near the surface. Near the keel, the hull geometry remains nearly constant to ordinate 1, and then steeply diminishes (see same Figure (h), (i) and (j)). The flow pattern follows almost the same trend. The flow direction remains unchanged in the longitudinal direction from ordinate 5 down to ordinate 1. The steep change of the hull geometry does affect the flow direction to become not parallel to the hull surface.

5.5.3 Comparison with measurements

The comparison of the predictions with measurements as presented in Section 5.4.3 present the following observations:-

- o The predictions are generally in good agreement with the measurements over most of the hull (ordinate 4 down to ordinate 1), (see Figures 5.4.7 to 5.4.13).
- o The prediction at the stern region, ordinates $\frac{1}{4}$ and $\frac{1}{2}$, are not so good.
- o Measured velocity profiles are much steeper than the predicted ones near the wall. This

indicates that the predicted shear stress at the wall is less than the measured (computed from measurements) one. This in turn suggests either an error in the turbulence characteristics near the walls (possibly in the wall function) or an error in the measurements near the wall. The first measured point near the wall was about 2mm from the wall, while the pitot tube had a diameter of 2mm. The authors who described the measurements however, gave no indication of the accuracy of their measurements near the hull.

- o The comparison of the predictions with the measurements at node number 42, ordinate 3.5, shows that the measured boundary layer is much thicker than the predicted one. The pattern continues to be the same at nodes 36 and 30 at ordinates 3 and 2.5 respectively. Nodes 42, 36 and 30 are characterised by all being on the concave part of the hull, for which a similar observation was given in the previous section.
- o The comparisons made near the free surface (nodes 22, 17, 9, 5 and 1) show that the boundary conditions specified at the free boundary are not consistent with the measured ones. The reason

may be that the potential-flow solution specified at the free-boundary was computed as a function of the true hull and not the thickened one.

Further observations can be obtained from Figure 5.4.17. The comparison of the predicted and potential flow pressure at the outer boundary (a boundary condition of uniform, constant stagnation pressure at the outer boundary was assumed) shows that the predicted pressures are larger than the estimated ones from the inlet section (ordinate 5) down to ordinate 2.75; thereafter, the predicted pressures become smaller than the estimated ones. Such feature exist at all angular stations. The difference between the predicted and potential flow pressures at regions downstream of ordinate 2.75 is of the order of 0.01. Therefore, the longitudinal velocity at the outer boundary based on the potential flow has an error of up to 15% of the true value at this boundary. This suggests that our boundary condition at the outer boundary is not consistent with the boundary layer calculation.

5.5.4 Turbulence characteristics

Figure 5.4.18 presents the distributions of turbulence kinetic energy and dissipation along the perpendicular to the hull. The figure shows that the profiles

of the level of kinetic energy of turbulence, c_k , and the normalised turbulence length scale, ln , exhibit the same pattern elsewhere. The turbulence kinetic energy increases steeply near the hull to a maximum, then decreases and takes a steady value far from the hull; the length scale, on the other hand, increases gradually to a maximum, then it reduces to a steady value slightly less than maximum. Both the curves of c_k and ln go to a steady value as set by the zero-gradient boundary condition of k , and ϵ at the outer boundary. The decay of the turbulence quantities is a known fact and is similar to the decay of turbulence behind a grid.

5.5.5 Summary

Agreement of the predictions with the experimental measurements was achieved, in the present work, only forward of the stern. For the stern region, the agreement was not so good. Many reasons can contribute to the disagreement between predictions and the measurements, some of these are presented below:

- (a) The boundary conditions specified at the free boundary, as supplied by IHI (which were obtained from a potential flow solution using the true hull), appear not to conform with the experimentally determined values of velocities

near the free surface. Such a pattern is recognised also at the stern region.

- (b) Inlet boundary conditions seem to fail to represent the pattern indicated by the measurements in an accurate manner; the imposed inlet distribution of boundary layer thickness overlooks an early thickening at the concave part of the hull.
- (c) The finite-difference grid employed is not fine enough.

Other reasons may also contribute to the poorness of the agreement at some places, they are:

- (d) The turbulence model currently employed incompletely accounts for the complex turbulence structure which occurs in the presence of secondary flows, as in the present case.
- (e) The approximation made at the air-water interface by assuming zero-gradient of property is not well justified.
- (f) Inaccuracy in the measurements.

General errors in the computational procedure may also account for such poor agreement at some places, some of these are:

- (g) Errors in any of the interpolation processes used in the computational procedure.
- (h) Errors due to the modifications made to the hull geometry at ordinate 0.25.

5.6 Comparison Between the Parabolic and Partially-Parabolic Solutions

In Section 1.2, a description of the flow around ships was given. Further understanding from Section 1.4.1 elaborates that near midship the flow is nearly parabolic, while near the stern the flow is either elliptic or partially-parabolic. It is in this section that such flow characteristics is reviewed once again. Comparison between the parabolic and partially-parabolic solutions of the test case, presented in Section 5.4, and the measurements given in Figure 5.6.1. The figure presents the comparisons between the experimental data and the calculations made by both the parabolic and the partially-parabolic procedures. The quantities presented are the longitudinal velocity distributions with distance from the hull at different computational stations characterised by their ordinate numbers.

The following phenomena may be observed from the figures. The partially-parabolic solution is, in principle, superior to the parabolic one because of the downstream

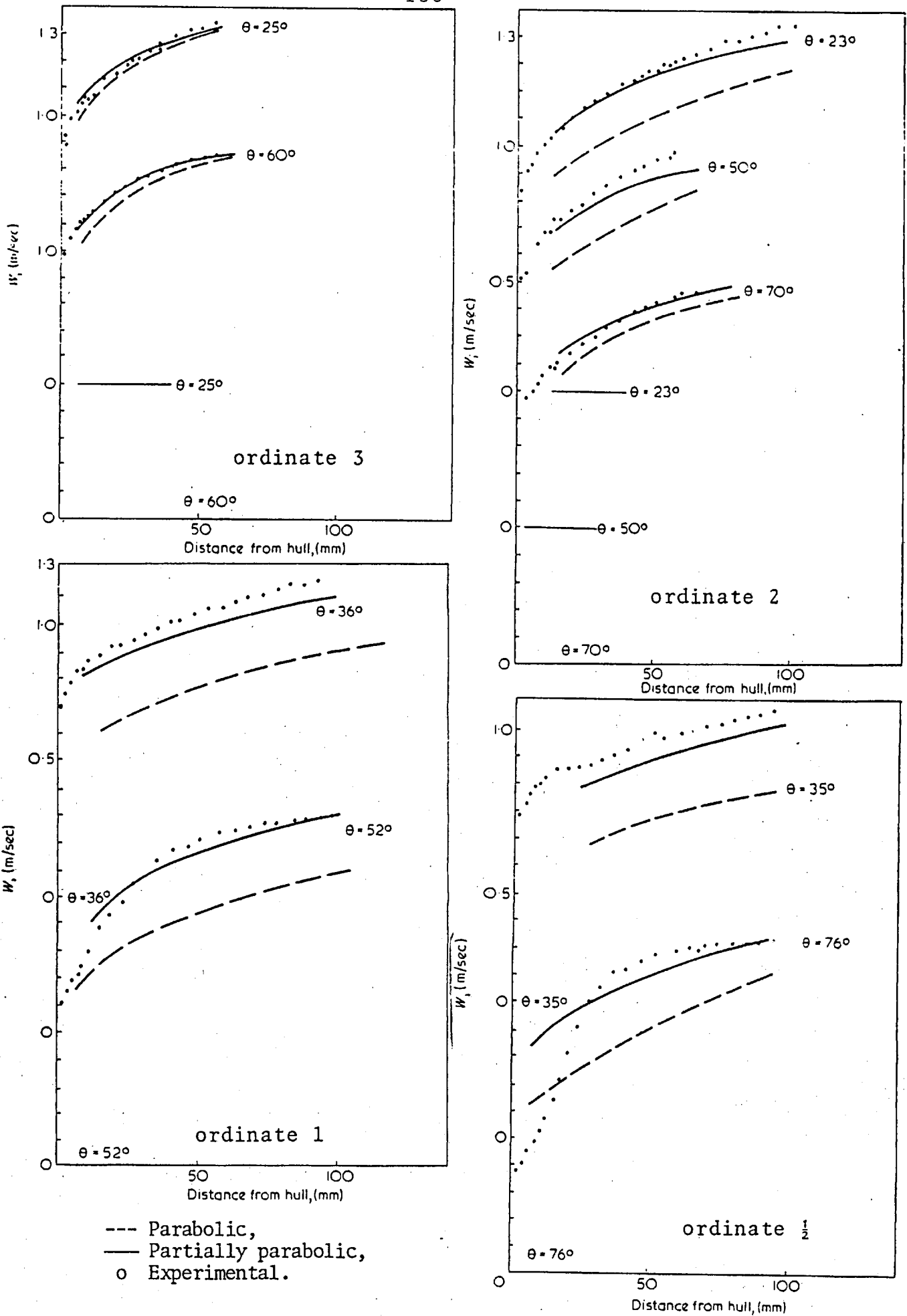


Figure 5.6.1 Comparison of axial velocity profiles

pressure effects propagating upstream from the stern region (which are not taken into account by the parabolic procedure).

At ordinate 3 (i.e. far away from the stern), the partially-parabolic and parabolic solutions are quite close to each other, indicating the weak influence of the stern region so far upstream. Nearer to the stern, however, large discrepancies between the partially-parabolic results and the parabolic ones are observed. The partially-parabolic procedure showed better agreement with the measurements than the parabolic one. This has been expected, as the steep hull gradients near the stern have certainly created partially-parabolic effects.

5.7 Closure

The present numerical scheme, incorporating the two-equation $k-\epsilon$ model of turbulence, has been successfully applied to the prediction of flow around ships' hulls. Detailed comparisons with experimental measurements have been made so as to assess the accuracy of the present procedure. The agreement was good over most of the hull; however, at the stern the agreement was not so good.

Further refinement of the method is needed with regard to handling many of the factors affecting the solution, i.e. pressure interpolation, geometric area and volume calculation, wall function, etc.

CHAPTER 6 - CONCLUDING REMARKS

6.1 Present Achievements

In conclusion, the main achievements of the research described in this thesis may be summarised as follows:

- (a) A non-orthogonal finite-difference formulation for the problem of three-dimensional turbulent boundary layers on arbitrary shaped bodies, but basically cylindrical, has been successfully demonstrated. The major advantage of this approach is that accurate prescription of the conditions at the flow boundaries which generally determine the solution to the governing equations is easily allowed for. This technique has been embodied in a three-dimensional computational procedure for partially-parabolic flows, as devised by Pratap and Spalding, (1975).
- (b) Numerical solutions have been obtained for turbulent flow around ships' hulls. The research project reported in this thesis, and in other published work, represents the first application of such procedure to the predictions of flow around ships' hulls. The advantages and merits of the present method were stated earlier in the review of other method (Section 1.4.2).

- (c) An assessment of the qualities of the present method was achieved by comparing the predictions with available measurements, and several recommendations for improving the performance of the procedure will be given in the next section.

6.2 Suggestions for Future Work

The present investigation suggests a few areas where further research is needed to improve the present capabilities to predict flow around ships, these areas are as follows:

- (1) Firstly, it would be beneficial from the point of view of economy if the convergence rate of the partially-parabolic calculation procedure can be further improved. The use of a three-dimensional solver for the pressure correction equation will certainly speed up the calculations.
- (2) To include an iterative procedure between the boundary layer calculation and the potential-flow one. Thus achieving better boundary conditions at the free-boundary.
- (3) To include a method which can predict the conditions at the air-water interface, such as the GALA method, and to be able to incorporate the effect of surface waves on the calculations.

- (4) In future, it may be advantageous to change the present coordinate system to be based in curved coordinates in the axial direction.

Further study can also be done in the method through:

- (a) Studying the effect of surface roughness on the predictions.
- (b) Comparing the predicted boundary-layer parameters, i.e. momentum thickness, shape factor, etc, with those of experiments.
- (c) Performing a parametric study of the effect of inlet turbulence levels, and inlet velocity profiles on the predictions.
- (d) Examining the effect of the location of the free-boundary on the predictions.
- (e) Obtaining more results with finer grids.
- (f) Predicting flow around other ships, where measurements are available.

REFERENCES

ABDELMEGUID A M, KOOSINLIN M L & MARKATOS N C (1978)

"A Prediction Method for Flow Around Ships' Hulls - The FLASH Computer Code". CHAM Limited, London, Report No. TR/50.

ABDELMEGUID A M, MARKATOS N C, SPALDING D BRIAN & MURAOKA K (1978)

"A Method of Predicting Three-Dimensional Turbulent Flows Around Ships' Hulls". Paper presented at the International Symposium on Ship Viscous Resistance, SSPA, Goteborg.

ABDELMEGUID A M, MARKATOS N C, MURAOKA K & SPALDING D BRIAN (1979)

"A Comparison Between the Parabolic and Partially-Parabolic Solution Procedures for Three-Dimensional Turbulent Flows Around Ships' Hulls". Appl. Math. Modelling, Vol 13, August. Also in NATO Advanced Study Institute Istanbul, 1978.

ABDELMEGUID A M & SPALDING D BRIAN (1979b)

"Turbulent Flow and Heat Transfer in Pipes with Buoyancy Effects". Journal of Fluid Mechanics, Vol 94, Part 2.

EAST L F (1975)

"Computation of Three-Dimensional Turbulent Boundary Layers". Euromech 60, Trondheim 1976. Published as Report FFA TN AE-1211, The Aeronautical Research Institute of Sweden.

ELHADIDY M (1980)

"Application of a Low-Reynolds Number Turbulence Model and Wall Function to Steady and Unsteady Heat Transfer Computations". PhD Thesis, London University.

ESCUDIER M P (1966)

"The Distribution of Mixing-Length in Turbulent Flows Near Walls". Imperial College, London, Heat Transfer Section, TWF/TN/1.

DYNE G (1977)

"A Theoretical Scale Effect Study on the Propulsion Coefficients of a Body of Revolution". DNV Symposium on Hydrodynamics of Ship and Offshore Propulsion Systems, Hovik, Oslo, 1977.

GOSMAN A D, PUN W M, RUNCHAL A K, SPALDING D BRIAN & WOLFSHTEIN M (1969)

"Heat and Mass Transfer in Recirculating Flows". Academic Press, London.

HARLOW F H (1973)

"Turbulence Transport Modelling". AIAA Selected Reprint Series, 15.

HARLOW F H & NAKAYAMA P I (1968)

"Turbulent Transport Equations". Phys. of Fluids, Vol 8, p.2182.

HATANO S & HOTTA J (1977)

"A Second Order Calculation of Three-Dimensional Turbulent Boundary Layers". Trans. West-Japan Society of Naval Architects, No. 53.

HAUKE W (1974)

"Grenzschichtmessungen an Modellschiffen im Windkanal". Schiffbauforschung 13, 1974.

HINZE J O (1959)

"Turbulence: An Introduction to Its Mechanism and Theory". McGraw-Hill Series in Mech Eng.

HOFFMANN H P (1976)

"Untersuchung der 3-Dimensionalen, Turbulenten Grenzschicht an Einem Schiffsdoppelmodell in Windkanal". Institute Fur Schiffbau, Der Universitat Hamburg, Bericht No. 343.

HUMPHREYS D A (1979)

"Comparison of Boundary Layer Calculations for a Wing: The May 1978 Stockholm Workshop Test Case". Stockholm, January 1979. Published as Report FFA TN AE-1522, The Aeronautical Research Institute of Sweden.

JONES W P & LAUNDER B E (1973)

"Predictions of Low-Reynolds Number Phenomena with a Two-Equation Model of Turbulence". Int. J. Heat & Mass Transfer, 16, p.1119.

LARSSON L (1974a)

"Boundary Layers of Ships: PART I - A Literature Survey on Investigations of Three-Dimensional Boundary Layers". The Swedish State Shipbuilding Experimental Tank, SSPA Report No. 44.

LARSSON L (1974b)

"Boundary Layers of Ships: PART II - A Literature Survey on Investigations of Ship and Model Boundary Layer". SSPA Report No. 45.

LARSSON L (1974c)

"Boundary Layers of Ships: PART III - An Experimental Investigation of the Turbulent Boundary Layer on a Ship Model". SSPA Report No. 46.

LARSSON L (1974d)

"Boundary Layers of Ships: PART IV - Calculation of the Turbulent Boundary Layer on a Ship Model". SSPA Report No. 47.

LARSSON L (1976)

"An Experimental Investigation of the Three-Dimensional Boundary Layer on a Ship Model". Paper presented at the 11th Symposium on Naval Hydrodynamics, London.

LARSSON L (1978)

"Ship Viscous Resistance: A Survey". SSPA Report No. 57.

LAUNDER B E, MORSE A P, RODI W & SPALDING D BRIAN (1972)

"The Prediction of Free Shear Flows: A Comparison of Six Turbulence Models". NASA Shear Flows Conference, Virginia NASA Report No. sp-311.

LAUNDER B E & SPALDING D BRIAN (1972)

"Mathematical Models of Turbulence". Academic Press, London.

LAUNDER B E & SPALDING D BRIAN (1973)

"The Numerical Computation of Turbulent Flows". Computer Methods for Applied Mechanics and Engineering, 3, p.269.

MALIN M R & MARKATOS N C (1978)

"The Prediction of Axisymmetric, Turbulent Flow Over the Tail Region of the Patel Body". CHAM Limited, London, Report No. 2030/1.

MARKATOS N C, MALIN M R & SINGH A (1979a)

"Computation of the Flow Over the Tail Region of the Patel Body of Revolution". CHAM Limited, London, Report No. 2030/2.

MARKATOS N C, MALIN M R & SINGH A (1979b)

"Computation of Flow Over Patel's Half and Full Body of Revolution". CHAM Limited, London, Report No. 2030/3.

MCQUIRK J (1975)

"Prediction of Turbulent Buoyant Jets in Co-Flowing Streams". Imperial College, London, Heat Transfer Section, HTS/75/24.

MURAOKA K (1977)

"Private Communications". London, 1977.

NAMIMATSU M & MURAOKA K (1974)

"Wake Distribution of Full Form Ship". IHI Engineering Review, Japan, Vol 7, No. 3.

NASH J F & PATEL V C (1972)

"Three-Dimensional Turbulent Boundary Layers". SBC
Technical Books Inc., Atlanta.

PATANKAR S V, PRATAP V S & SPALDING D BRIAN (1975)

"Prediction of Turbulent Flow in Curved Pipes". Journal
of Fluid Mechanics, 67, 3, p.595.

PATANKAR S V & SPALDING D BRIAN (1972a)

"A Calculation Procedure for Heat, Mass and Momentum
Transfer in Three-Dimensional Parabolic Flows". Int.
J. of Heat and Mass Transfer, 15, p.1787.

PRATAP V S & SPALDING D BRIAN (1975)

"Numerical Computations of Flows in Curved Ducts".
Aero. Quarterly, Vol 26, pp.219-228.

RAMACHANDRA V (1979)

"The Numerical Prediction of Flow and Heat Transfer
in Rod-Bundle Geometries". PhD Thesis, London University.

RUNCHAL A K (1969)

"Transient Processes in Steady Two-Dimensional Separated
Flows". Imperial College, London, Heat Transfer
Section, HTS/69/18.

SACHDEVA R C & PRESTON J H (1976)

"Investigation of Turbulent Boundary Layers on a Ship Model". Schiffstechnik, Vol 23, No. 111, March 1976.

SCHLICHTING (1960)

"Boundary Layer Theory". McGraw-Hill Book Co, Inc.

SERAG-EL-DIN M A S (1977)

"The Numerical Prediction of The Flow and Combustion Processes in a Three-Dimensional Can Combustor". PhD Thesis, London University.

SHARMA D (1974)

"Turbulent Convective Phenomena in Straight, Rectangular-Sectioned Diffusers". PhD Thesis, London University, also HTS/74/26.

SPALDING D BRIAN (1974)

"Numerical Computation of Steady Boundary Layers: A Survey". Invited Lecture at Conference on Computational Methods and Problems in Aeronautical Fluid Dynamics, Manchester, September 1974. Also as HTS/74/27.

SPALDING D BRIAN (1974)

"Prediction and Control of Flow Around Ships' Hulls". CHAM Limited, London, Report No. TR/16.

SPALDING D BRIAN (1975)

"Calculation Procedure for Three-Dimensional Parabolic and Partially-Parabolic Flows". Imperial College, Heat Transfer Section, Report No. HTS/75/5.

TANAKA I & HIMENO Y (1975)

"First Order Approximation to the Three-Dimensional Turbulent Boundary Layer and Its Application to Model-Ship Correlation". Japanese Society of Naval Architects, Vol 138.

TANAKA I ET AL (1977)

"Examples of Stern Flow Field Using Boundary Layer Theory Approach". DNV Symposium on Hydrodynamics of Ship and Offshore Propulsion Systems, Hovik, Oslo.

TATCHELL D G (1975)

"Convective Processes in Confined Three-Dimensional Boundary Layers". PhD Thesis, London University. Also as HTS/75/20.

TENNEKES H & LUMLEY J L (1973)

"A First Course in Turbulence". The MIT Press, UK.

NOMECLATURE

<u>SYMBOL</u>	<u>MEANING</u>
A_i, A'_i	Coefficients in the finite-difference equation.
B_i, B'_i	Coefficients in the finite-difference equation.
$C_1, C_2,$ C_μ	Constants of the k- ϵ turbulence model.
D'	Refers to the downstream station.
$D^u, D^v, D^w,$ D^{vu}, D^{vw}	Coefficients of pressure gradient terms.
e	Location on grid.
E	<u>E</u> ast, grid node east of <u>P</u> .
E	Constant in the logarithmic law of wall.
GE	Generation term for the turbulent kinetic energy, k .
$J_{\phi,i}$	Flux of ϕ along the i th direction.
k	Turbulence kinetic energy.
ℓ	Turbulence length scale.
L^i_J	Convection coefficient in finite-difference equation.
$\dot{m}$	Residual mass source.
n	Location on grid.
N	<u>N</u> orth, grid node north of <u>P</u> .
p	Static pressure.
P	Grid node of interest.
$\bar{p}$	Average pressure field, used in the parabolic procedure.

<u>SYMBOL</u>	<u>MEANING</u>
Pr_ϕ	Prandtl/Schmidt number appropriate to the transport of ϕ .
r	Coordinate in the cylindrical-polar system.
s	Source/sink term.
s	Location on grid.
S	<u>S</u> outh, grid location south of <u>P</u> .
T_J^i	Diffusive coefficient of finite-difference equation.
U	Refers to upstream station.
U	Velocity vector.
u,v,w	Velocity components along the θ, r, z directions, respectively.
w	Location on grid.
W	<u>W</u> est, grid node west of <u>P</u> .

<u>GREEK SYMBOLS</u>	<u>MEANING</u>
ϵ	Dissipation rate of turbulence kinetic energy.
Γ	Diffusion coefficient for ϕ .
ρ	Density of fluid.
η, ζ, ξ	Coordinates in the (η, ζ, ξ) system.
θ	Coordinate in the cylindrical-polar system.
μ	Coefficient of viscosity.
κ	Constant in the logarithmic law for variation of near-wall velocity.

GREEK
SYMBOLS

MEANING

ϕ	General flow variable.
τ	Shear stress.
σ	Stress tensor.

SUBSCRIPTS

eff	Effective value.
l	Laminar.
t	Turbulent.
'	Denotes corrections in Chapter 4.

APPENDIX A

PUBLISHED PAPER

ON

TURBULENT FLOW AND HEAT
TRANSFER IN PIPES WITH
BUOYANCY EFFECTS

Turbulent flow and heat transfer in pipes with buoyancy effects

By A. M. ABDELMEGUID AND D. B. SPALDING

Imperial College of Science and Technology, Mechanical Engineering
Department, Exhibition Road, London SW7 2BX

(Received 4 November 1977 and in revised form 8 December 1978)

A finite-difference procedure is employed to predict the turbulent flow and heat transfer in horizontal, inclined and vertical pipes when influenced by buoyancy. The flow is treated as parabolic; and the turbulence model used involves the solution of two differential equations, one for the kinetic energy of turbulence and the other for its dissipation rate. Results are presented for the velocity and temperature fields, and the associated flow-resistance and heat-transfer coefficients. The predictions for horizontal and vertical pipes have been compared with the available experimental results, and the agreement obtained is good.

CONTENTS

	<i>Page</i>
1. Introduction	
1.1. <i>The problem considered</i>	384
1.2. <i>Previous work</i>	384
1.3. <i>Present contribution</i>	385
2. Prediction procedure	385
2.1. <i>Governing differential equations</i>	385
2.2. <i>Turbulence model</i>	386
2.3. <i>Solution procedure</i>	387
2.4. <i>Computational details</i>	387
3. Results and discussion	389
3.1. <i>Horizontal pipes</i>	389
3.2. <i>Vertical pipes</i>	389
3.3. <i>Inclined pipes</i>	396
4. Conclusions	397

1. Introduction

1.1. The problem considered

The flow and heat-transfer characteristics of turbulent pipe flow are affected by the presence of gravity forces; at moderate Reynolds numbers and high Grashof numbers, there occur considerable distortion of velocity and temperature profiles, and changes in the heat-transfer and hydrodynamic resistances. The character of these changes are affected by the inclination of the pipe. For horizontal pipes, the gravity force is perpendicular to the main flow direction and operates on the cross-stream flow; then the flow is three-dimensional. For vertical pipes, the gravity force is aligned with the pipe axis, so that axial symmetry is preserved. For inclined pipes, the gravity force acts in both the main-flow and the cross-stream-flow directions.

The present paper demonstrates and tests a means of computing the velocity and temperature fields, and the associated flow-resistance and heat-transfer coefficients, in pipe flows with appreciable buoyancy effects. The situation of fully-developed flow is considered for inclined pipes, while both the developing and fully-developed regions are considered for vertical and horizontal pipes.

1.2. Previous work

(a) *Horizontal pipes.* The literature on buoyancy-influenced flow and heat transfer in horizontal pipes is limited. Polyakov (1974) presented an analysis for the development of a secondary free-convection current in forced turbulent flow with weak buoyancy, i.e. at low Grashof numbers. In this analysis, he assumed that buoyancy forces do not affect turbulent-transfer coefficients. Skiadaressis & Spalding (1977) presented a finite-difference procedure for the prediction of such flow phenomena, allowing for the associated changes in the turbulent transfer properties; their predictions were in good agreement with experimental data. Petukhov *et al.* (1974) presented experimental data for the local heat transfer, and for the fields of axial velocity and temperature, for air. More recently, an experimental study, also conducted by Petukhov *et al.* (1976), provided data for the distribution of fluctuation intensities of temperature and velocity.

(b) *Vertical pipes.* All past theoretical studies on vertical pipes have been confined to the situation of fully-developed flow. Ojalov *et al.* (1967) developed a numerical method for solving the corresponding conservation equations, based on the reduction of the basic conservation laws to three coupled, linear, integro-differential equations. Polyakov (1973) presented a theoretical study of the limits of the influence of the buoyancy forces on the velocity distribution, the temperature, and the heat-transfer and friction resistances.

The measurements of Carr, Connor & Buhr (1973) and Buhr, Horsten & Carr (1974) represent a wide range of data for ascending flow in vertical pipes. They observed that, at high Grashof number, a limiting profile shape was approached, with the centre velocity well below the mean and the maximum occurring in the vicinity of the wall. Petukhov (1976) presented a wide range of experimental data, and concluded that the influence of gravitation on turbulent transfer is determined by two distinct effects: on the one hand, it changes the fields of the average values of velocity and temperature; this results in a change in the turbulent transfer characteristics. On the other hand, it has a direct effect on the motion of the turbulent elements of the fluid, thus strength-

ening or weakening the intensity of turbulent mixing. Depending on the conditions, both effects may be commensurable, or one of them may predominate.

(c) *Inclined pipes.* Little research has been done on inclined pipes, and, to the authors' knowledge, none in turbulent flows.

1.3. *Present contribution*

The present paper reports the results of a theoretical analysis of fluid flow and heat transfer in vertical, inclined and horizontal pipes with appreciable influence of buoyancy, and with uniform heat transfer through the walls.

The aims of the present work are to extend and generalize the numerical method used by Skiadaressis & Spalding (1977) to inclined pipes, and so to provide insight into the physical processes involved, while simultaneously demonstrating the capabilities of the calculation procedure as a useful design tool.

The main assumption is that downstream effects are not transmitted upstream; the assumption renders the finite-difference procedure fully parabolic. The numerical algorithm employed is that reported by Patankar & Spalding (1972); the effects of turbulence are represented by a 'two-equation model' (Launder & Spalding 1974), in which differential equations are solved for the turbulence energy and for its dissipation rate. The velocity and temperature fields, as well as the heat-transfer and flow-resistance coefficients, have been predicted both in the developing and the fully-developed regions, and for both vertical and horizontal pipes. Comparisons with experimental data for fully-developed longitudinal-velocity and temperature profiles, are presented for horizontal and vertical pipes. Comparison of heat-transfer coefficients with experimental data are presented also for horizontal pipes.

2. Prediction procedure

2.1. *Governing differential equations*

The equations governing the present problem are those of continuity, momentum and energy. These equations, in cylindrical-polar co-ordinates, and in accordance with figure 1, take the following forms.

Continuity:

$$\frac{1}{r} \frac{\partial}{\partial r} (\rho r v) + \frac{1}{r} \frac{\partial}{\partial \theta} (\rho u) + \frac{\partial}{\partial z} (\rho w) = 0. \quad (2.1)$$

Longitudinal momentum:

$$\begin{aligned} & \frac{1}{r} \frac{\partial}{\partial r} (\rho r v w) + \frac{1}{r} \frac{\partial}{\partial \theta} (\rho u w) + \frac{\partial}{\partial z} (\rho w^2) \\ &= -\frac{\partial \bar{p}}{\partial z} - \rho g \beta (T - T_{\text{ref}}) \sin \alpha + \frac{1}{r} \left[\frac{\partial}{\partial r} (r \tau_{rz}) + \frac{\partial \tau_{\theta z}}{\partial \theta} \right]. \end{aligned} \quad (2.2)$$

Radial momentum:

$$\begin{aligned} & \frac{1}{r} \frac{\partial}{\partial r} (\rho r v^2) + \frac{1}{r} \frac{\partial}{\partial \theta} (\rho v u) + \frac{\partial}{\partial z} (\rho w v) - \frac{\rho u^2}{r} \\ &= -\frac{\partial \bar{p}}{\partial r} - \rho g \beta (T - T_{\text{ref}}) \cos \theta \cos \alpha + \frac{1}{r} \left[\frac{\partial}{\partial r} (r \tau_{rr}) + \frac{\partial}{\partial \theta} (\tau_{r\theta}) \right] - \frac{\tau_{\theta\theta}}{r}. \end{aligned} \quad (2.3)$$

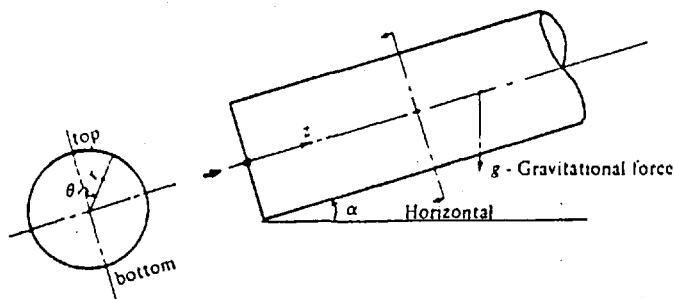


FIGURE 1. The geometry considered.

Angular momentum:

$$\begin{aligned} \frac{1}{r} \frac{\partial}{\partial r} (\rho r v u) + \frac{1}{r} \frac{\partial}{\partial \theta} (\rho u^2) + \frac{\partial}{\partial z} (\rho w u) + \frac{\rho u v}{r} \\ = -\frac{1}{r} \frac{\partial p}{\partial \theta} + \rho g \beta (T - T_{\text{ref}}) \sin \theta \cos \alpha + \frac{1}{r} \left[\frac{1}{r} \frac{\partial}{\partial r} (r^2 \tau_{r\theta}) + \frac{\partial \tau_{\theta\theta}}{\partial \theta} \right]. \end{aligned} \quad (2.4)$$

Thermal energy:

$$\frac{1}{r} \frac{\partial}{\partial r} (\rho r v T) + \frac{1}{r} \frac{\partial}{\partial \theta} (\rho u T) + \frac{\partial}{\partial z} (\rho w T) = \frac{1}{r} \left[\frac{\partial}{\partial r} (r J_r) + \frac{\partial}{\partial \theta} (J_\theta) \right]. \quad (2.5)$$

In these equations u , v and w represent the velocity components in the θ , r and z co-ordinate directions respectively. θ is measured from the top of the pipe. ρ represents the density, p is the pressure and β is the thermal expansion coefficient. The τ 's represent components of the shear-stress tensor and are expressible in terms of the velocity gradients and an effective viscosity, which varies in the field. J 's represent components of heat-flux and are expressible in terms of the temperature gradients and an effective heat diffusivity.

The above equations have been reduced from the general three-dimensional form to that shown by use of the assumptions stated below:

(i) The flow is assumed to be of the boundary-layer type. This implies that neither momentum nor heat is transferred by molecular or turbulent mixing in the longitudinal direction.

(ii) A space-average pressure, $\bar{p}$, is supposed to prevail at each cross-section in the longitudinal momentum equation; whereas the pressure variations with r and θ are allowed to influence the other two momentum equations. This 'pressure uncoupling' is a necessity, legitimate for parabolic flows, if a marching-integration procedure is to be used.

(iii) The viscous dissipation is omitted from the energy equation, because the velocities in question are low.

(iv) Only small variations in density occur, due to temperature differences; and the Boussinesq approximation prevails. In (2.5), T_{ref} represents a reference temperature in the field.

2.2. Turbulence model

The effective viscosity, μ_{eff} , is calculated from a two-equation turbulence model (Launder & Spalding 1974). The governing differential equations for turbulent kinetic energy k and its dissipation rate ϵ are:

Kinetic energy of turbulence,

$$\frac{1}{r} \frac{\partial}{\partial r} (\rho r v k) + \frac{1}{r} \frac{\partial}{\partial \theta} (\rho u k) + \frac{\partial}{\partial z} (\rho w k) = \frac{1}{r} \frac{\partial}{\partial r} \left[r \Gamma_k \frac{\partial k}{\partial r} \right] + \frac{1}{r^2} \frac{\partial}{\partial \theta} \left[\Gamma_k \frac{\partial k}{\partial \theta} \right] + G - \rho \epsilon; \quad (2.6)$$

Dissipation rate,

$$\frac{1}{r} \frac{\partial}{\partial r} (\rho r v \epsilon) + \frac{1}{r} \frac{\partial}{\partial \theta} (\rho u \epsilon) + \frac{\partial}{\partial z} (\rho w \epsilon) = \frac{1}{r} \frac{\partial}{\partial r} \left[r \Gamma_\epsilon \frac{\partial \epsilon}{\partial r} \right] + \frac{1}{r^2} \frac{\partial}{\partial \theta} \left[\Gamma_\epsilon \frac{\partial \epsilon}{\partial \theta} \right] + C_1 \frac{G \epsilon}{k} - C_2 \frac{\rho \epsilon^2}{k}; \quad (2.7)$$

where

$$G \equiv \mu_t \left[2 \left\{ \left(\frac{1}{r} \frac{\partial u}{\partial \theta} + \frac{v}{r} \right)^2 + \left(\frac{\partial v}{\partial r} \right)^2 \right\} + \left\{ r \frac{\partial}{\partial r} \left(\frac{u}{r} \right) + \frac{1}{r} \frac{\partial v}{\partial \theta} \right\}^2 + \left(\frac{\partial w}{\partial r} \right)^2 + \left(\frac{1}{r} \frac{\partial w}{\partial \theta} \right)^2 \right]. \quad (2.8)$$

The expression for μ_{eff} is

$$\mu_{\text{eff}} = \mu_{\text{laminar}} + C_\mu \rho k^2 / \epsilon. \quad (2.9)$$

The turbulent viscosity μ_t is calculated from the relationship

$$\mu_t = C_\mu \rho k^2 / \epsilon. \quad (2.10)$$

The model contains five empirical constants which are assigned the following commonly used values:

$$C_\mu = 0.09, \quad C_1 = 1.44, \quad C_2 = 1.92, \quad Pr_k = 1.0, \quad Pr_\epsilon = 1.3.$$

A special but conventional treatment is adopted for the near-wall regions, since the variations of the flow properties are much steeper in this region; the practices adopted here are the same as described by Launder & Spalding (1974), with appropriate extensions to account for the three-dimensional nature of the flow. No special terms are included in the equations for k and ϵ to account for the interaction of buoyancy and turbulence, despite the conclusion of Petukhov (1976) mentioned above. Computational tests showed that the inclusion of these terms has no significant effect on flow or heat transfer.

2.3. Solution procedure

A finite-difference forward-marching procedure is applied to solve the equations governing the flow and heat transfer. The finite-difference equations are solved by marching from an upstream station, where the flow conditions are known, to successive stations downstream. The procedure thus computes the flow section by section along the pipe length. A full account of the procedure is given by Patankar & Spalding (1972), its main feature is that the downstream pressure field is first guessed, so as to lead to a preliminary velocity field, satisfying the momentum equations but not the continuity equation, then corrections are made in a systematic manner until all the equations are satisfied. No modifications were needed or made to the solution procedure.

2.4. Computational details

(a) *The computer program.* The differential equations described in § 2.1, together with the Patankar-Spalding (1972) method, were embodied in the computer code STABLER (*Steady Three-dimensional Analysis of Boundary-Layer Equations Revised*).

(b) *The grid.* Two different types of grid were used. For inclined and horizontal pipes, for which the flow is symmetrical about a vertical plane, computation was confined to half of the pipe. A grid of 15 nodes in the radial direction and 11 nodes in the angular

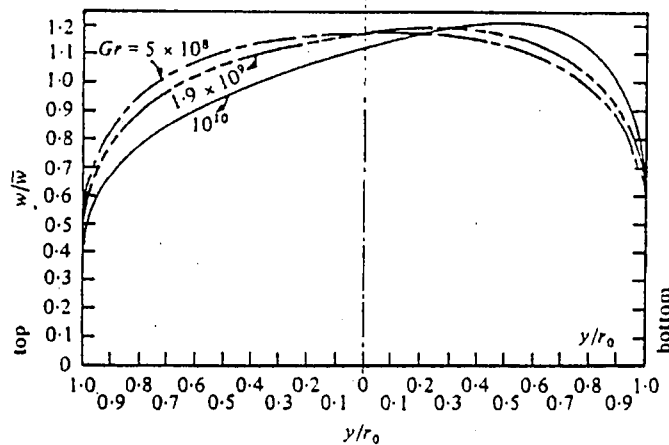


FIGURE 2. The variation of the fully developed velocity profile with heat input in horizontal pipes (view in vertical-diametral plane; $Re = 49\,500$, r_0 is the pipe radius).

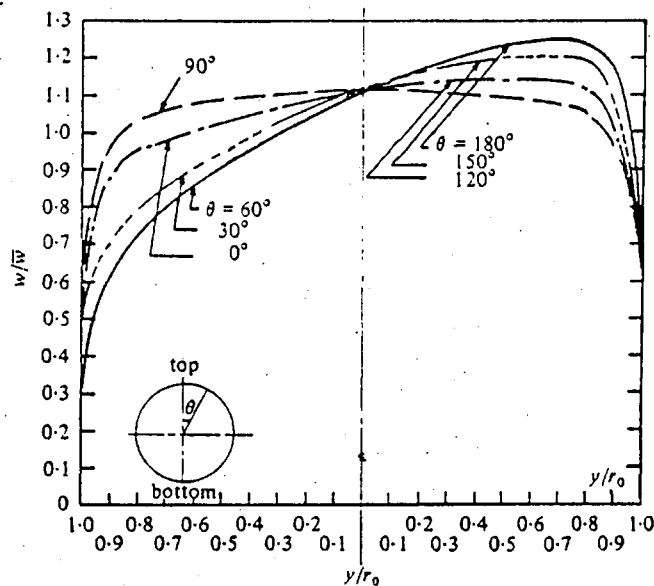


FIGURE 3. Fully developed velocity profiles at different angular stations in horizontal pipes under considerable thermo-gravitational forces ($Re = 25\,000$, $Gr = 10^{10}$).

direction was employed. In the angular direction, a nearly uniform grid distribution was chosen; in the radial direction, more grid nodes were concentrated in the region close to the wall.

For vertical pipes, the flow was axisymmetric and fewer grid nodes were needed in the circumferential direction. A 15×6 grid was employed, but, strictly speaking, a 15×1 grid was all that was needed.

(c) *Computational cost.* An initial isothermal section of 20 diameters preceded the test section in all the predictions.

Approximately 400 sections were visited by the marching-integration process, in the passage from entry to full development. The computer time on a CDC 6600 computer was of the order of 0.5 seconds per forward step.

3. Results and discussion

3.1. Horizontal pipes

Velocity profiles. Figure 2 presents fully developed longitudinal-velocity profiles, at a fixed angular position and at various Grashof numbers Gr . Figure 3 shows such profiles at various angular positions, for a fixed Gr .

The development of the longitudinal-velocity contours with distance from the entry, at a typical Gr , is shown in figure 4.

Figure 5 presents a comparison of the predicted velocity profiles in the vertical and horizontal diametral planes, with experimental data by Petukhov *et al.* (1974), and Petukhov (1976).

Temperature profiles. The temperature distributions in the cross-stream plane, are presented in figure 6. The figure also presents, for comparison, experimental data by Petukhov *et al.* (1974) and Petukhov (1976).

In figure 7, the development of Nusselt number, Nu , is plotted with distance from the entry, at Gr equal 1.9×10^9 at the top and bottom pipe generators. The figure also presents a comparison with the experimental data by Petukhov *et al.* (1974).

Discussion. It can be seen from figures 2 and 6 that the longitudinal-velocity and temperature maximums are shifted from the centre-line, because of the increase in Gr . Figures 5, 6 and 7 show that the agreement between measurements and predictions is good.

3.2. Vertical pipes

(a) *Ascending flow.* Figure 8 presents the development of the longitudinal-velocity profile with distance from the entry, at a fixed Grashof number. Figures 9 and 10 present the fully developed longitudinal-velocity and turbulent-kinetic-energy profiles, for various Grashof numbers. Figure 11 presents the variation of friction factor, with Gr for constant Re ; f_t represents the friction factor for turbulent forced flow.

Figure 12 presents the fully-developed temperature profiles for various Gr . In figure 13, the variation of Nu at various Gr is presented.

It can be seen from figures 9 and 10 that the mean flow and the turbulent transfer are much affected by the value of Gr : for Gr less than a critical value, Gr_c (here 10^9), the effect of buoyancy on turbulence characteristics is modest, and its effect on the averaged flow can be neglected; for Gr greater than Gr_c , buoyancy affects both the averaged flow and the turbulent transfer.

Figure 9 shows also that the maximum velocity occurs in the vicinity of the wall; and, with increase of Gr , the maximum velocity near the wall increases in value and is located nearer to the wall. This is the same trend as is described by Carr *et al.* (1973).

It can also be seen from figure 10 that the turbulent kinetic energy decreases with increase of Gr , until Gr_c is reached, and then increases; and it can be seen from

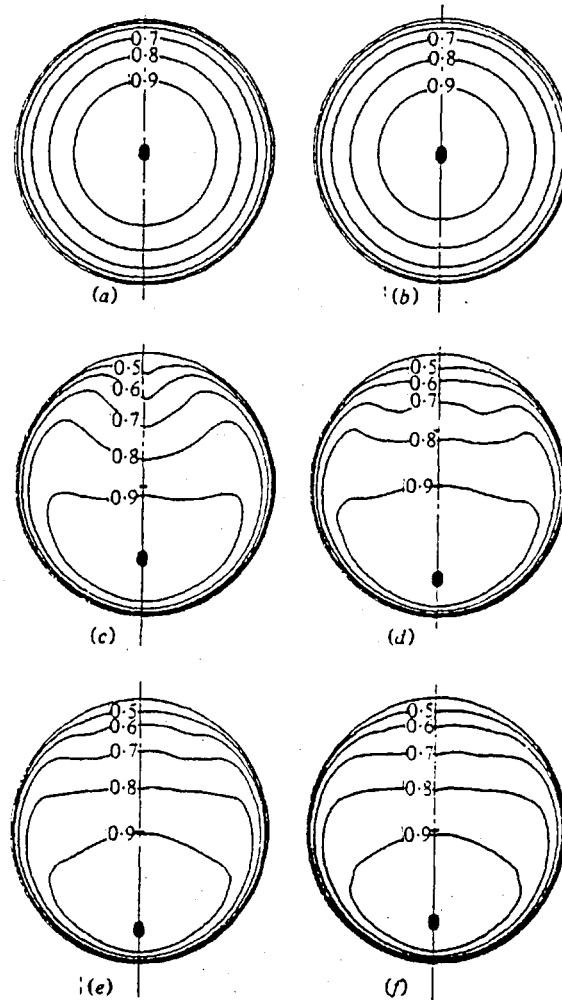


FIGURE 4. Axial-velocity contours in the thermal entry length in a horizontal pipe under considerable thermo-gravitational forces ($Re = 25000$, $Gr = 10^{10}$; contours are for w/w_{max}). (a) At the start of heating. (b) $z/D = 2.4$. (c) $z/D = 13.75$. (d) $z/D = 25.74$. (e) $z/D = 37.75$. (f) Fully developed, $z/D = 49.75$.

figure 13, that, at first, the Nusselt number decreases when the heat input is increased, and then rises. The initial decrease in Nu accords with the initial decrease in the turbulent kinetic energy; the subsequent increase in Nu is due to the buoyancy forces becoming dominant on both the averaged flow and the turbulent transfer.

Figures 14 and 15 compare the predictions with the experimental data of Buhr *et al.* (1974), with mercury as a working fluid. In the predictions, special attention had to be given to the so-called 'wall function'; the treatment adopted was that of El Hadidy & Spalding (1978, unpublished work). Agreement between measurements and predictions is satisfactory.

(b) *Descending flow.* Figures 16 and 17 present the fully developed longitudinal-velocity and turbulent-kinetic-energy profiles for various Grashof numbers. Figure

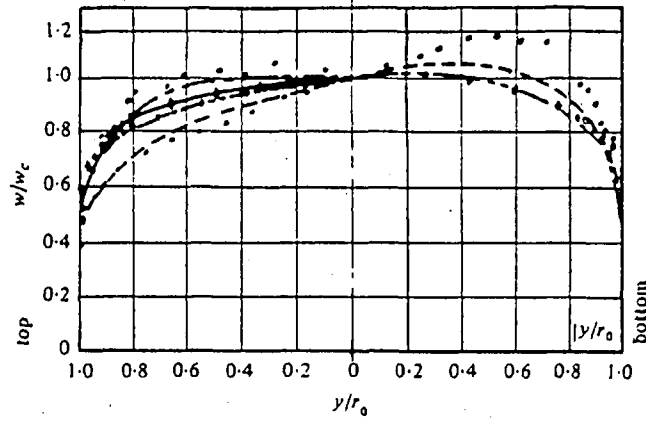


FIGURE 5. Velocity profiles compared with Petukhov *et al.* (1974) and Petukhov (1976). (Predicted, experimental) results for the vertical-diametral plane: (---, ◇), $Re = 52000$, $Gr = 10^9$; (---, ○), $Re = 26000$, $Gr = 7.7 \times 10^8$. Results for the horizontal-diametral plane: (—, ◆), $Re = 52000$, $Gr = 10^9$; (—, ●), $Re = 26000$, $Gr = 7.7 \times 10^8$.

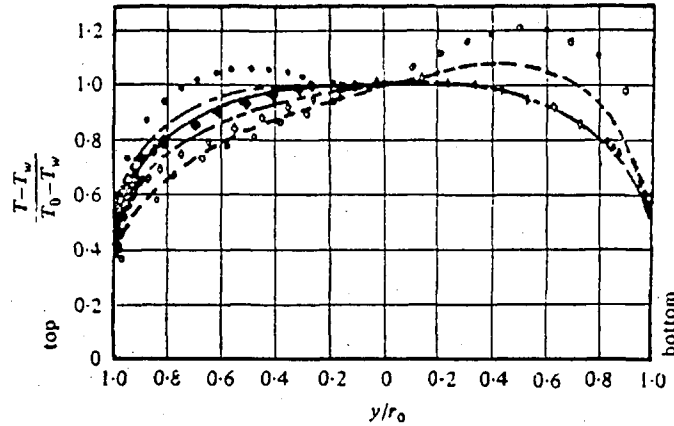


FIGURE 6. Temperature profiles compared with the data of Petukhov *et al.* (1974). Symbols as in figure 5.

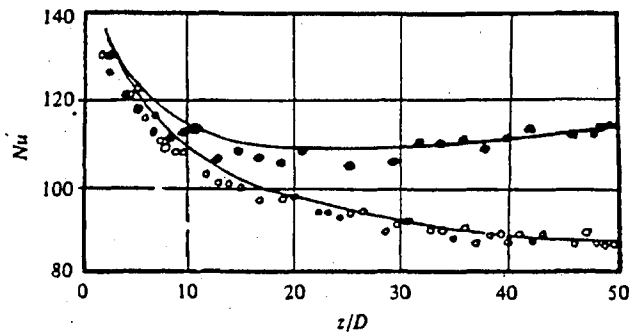


FIGURE 7. Variation of Nusselt number in the thermal entry length compared with the data of Petukhov *et al.* (1974). ($Re = 25000$, $Gr = 10^{10}$.) Experimental: ○, $\theta = 0^\circ$; ●, $\theta = 180^\circ$. Predicted: —.

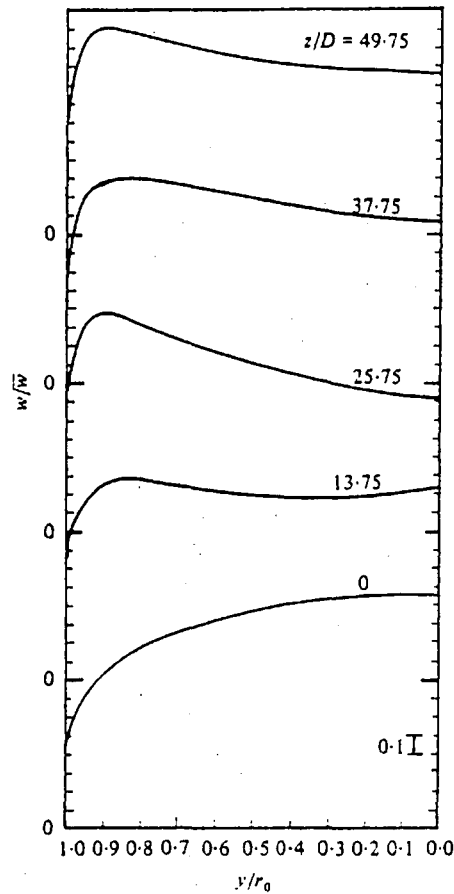


FIGURE 8. Axial-velocity profiles in the thermal entry length for an ascending flow of air in a vertical pipe ($Re = 25000$, $Gr = 10^{10}$).

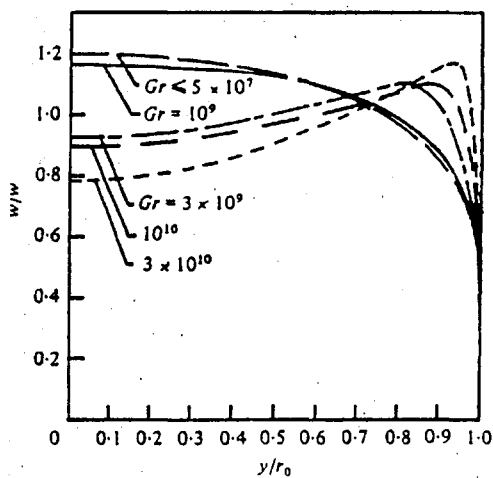


FIGURE 9

FIGURE 9. Variation of axial-velocity profile with the heat input for an ascending air in a vertical pipe ($Re = 25000$).

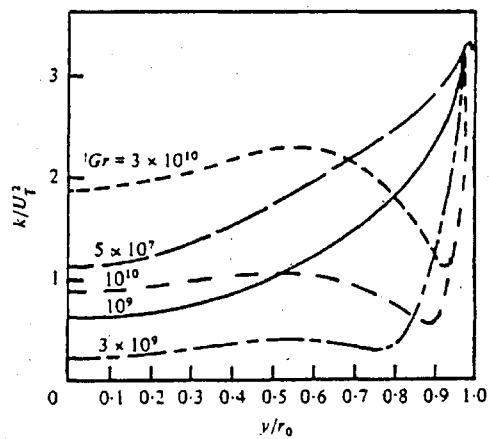


FIGURE 10

FIGURE 10. Variation of the kinetic energy of turbulence with the heat input ($Re = 25000$).

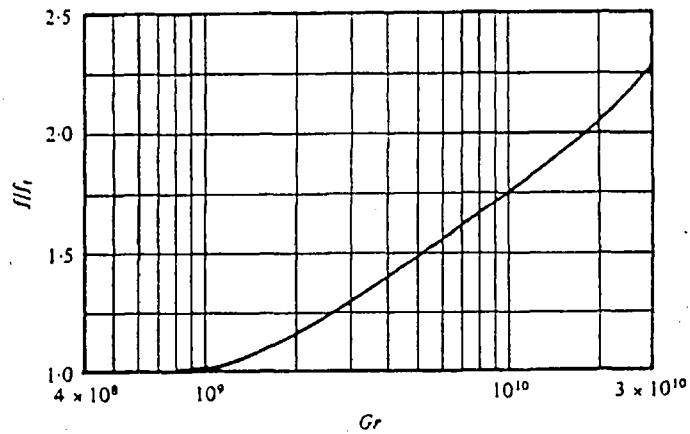


FIGURE 11. Friction factor variation with heat input (ascending flow).

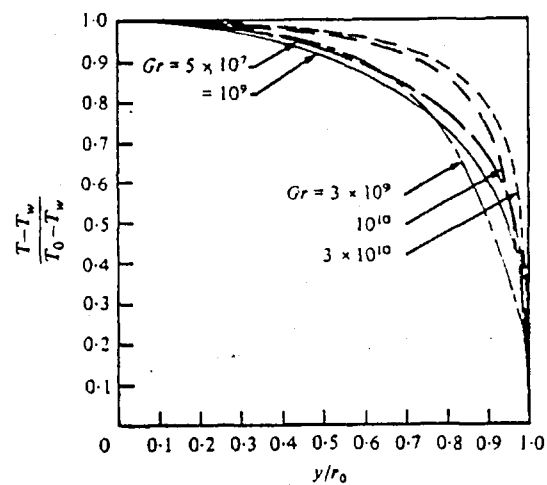


FIGURE 12. Temperature profiles ($Re = 25000$).

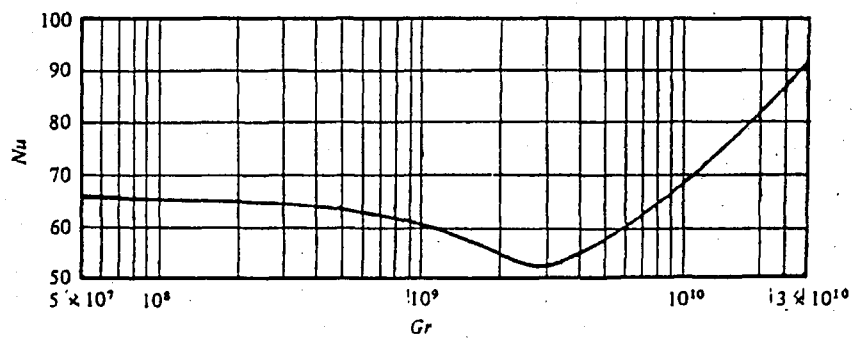


FIGURE 13. Variation of Nusselt number with heat input ($Re = 25000$, ascending flow).

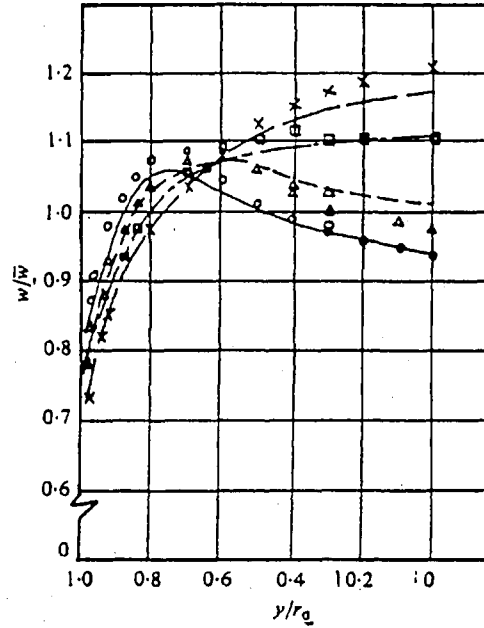


FIGURE 14. Comparison with the data of Buhr *et al.* (1974) of velocity profiles. (Experimental, predicted) results: ($\times$, ---), $Re = 60\,600$, $Gr = 1.19 \times 10^5$; ($\square$, -.-), $Re = 62\,200$, $Gr = 5.88 \times 10^5$; ($\triangle$, ----), $Re = 65\,400$, $Gr = 8.87 \times 10^5$; ($\circ$, —), $Re = 63\,200$, $Gr = 1.61 \times 10^5$.

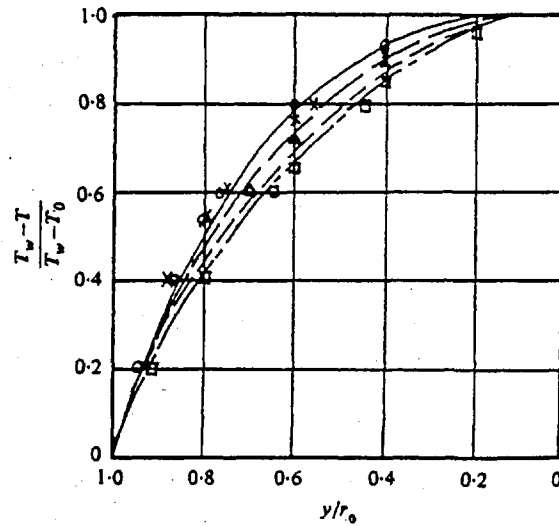


FIGURE 15. Comparison with the data of Buhr *et al.* (1974) of temperature profiles. (Experimental, predicted) results: ($\times$, ---), $Re = 61\,000$, $Gr = 1.11 \times 10^5$; ($\square$, -.-), $Re = 61\,600$, $Gr = 5.88 \times 10^5$; ($\triangle$, ----), $Re = 62\,400$, $Gr = 8.76 \times 10^5$; ($\circ$, —), $Re = 65\,000$, $Gr = 1.63 \times 10^5$.

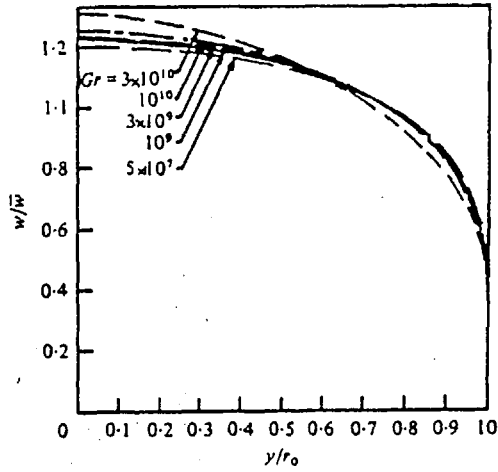


FIGURE 16

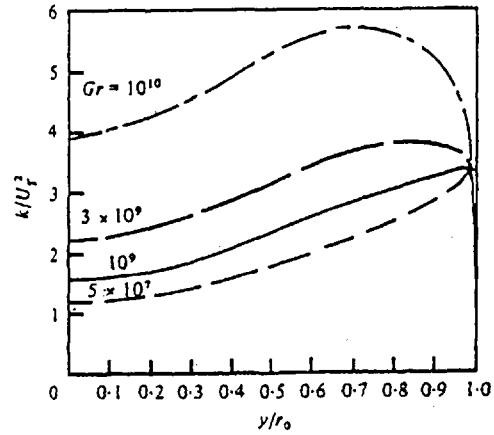


FIGURE 17

FIGURE 16. Variation of axial-velocity profile with heat input for a descending air in a vertical pipe.

FIGURE 17. Variation of the kinetic energy of turbulence with heat input.

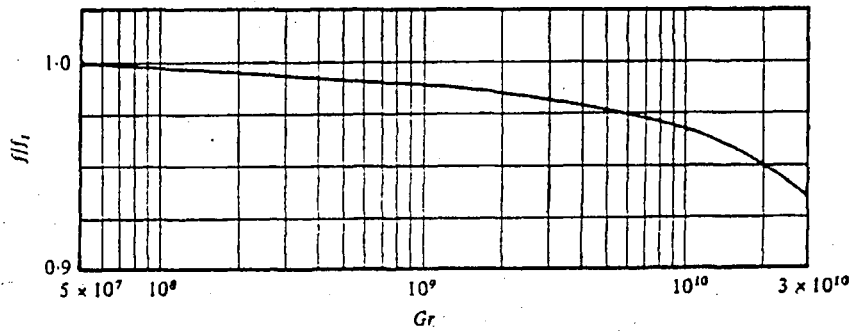


FIGURE 18. Variation of friction with heat input in a descending flow of air ($Re = 25000$).

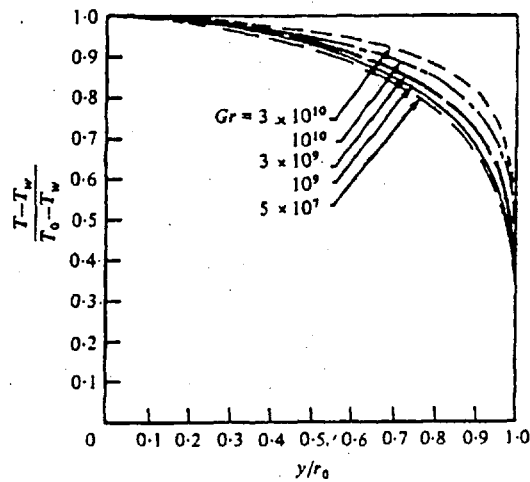


FIGURE 19. Temperature profiles ($Re = 25000$).

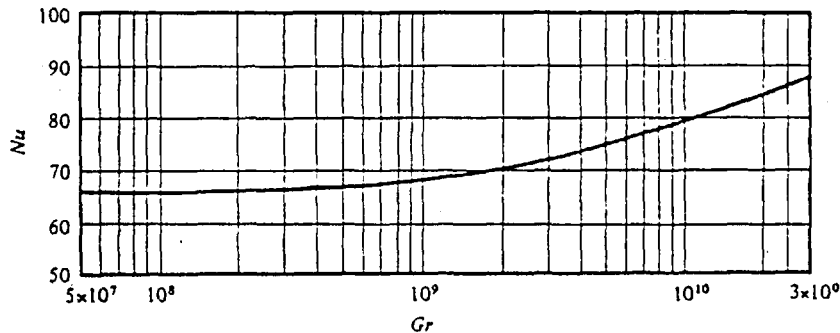


FIGURE 20. Variation of Nusselt number with heat input in a descending flow of air ($Re = 25000$).

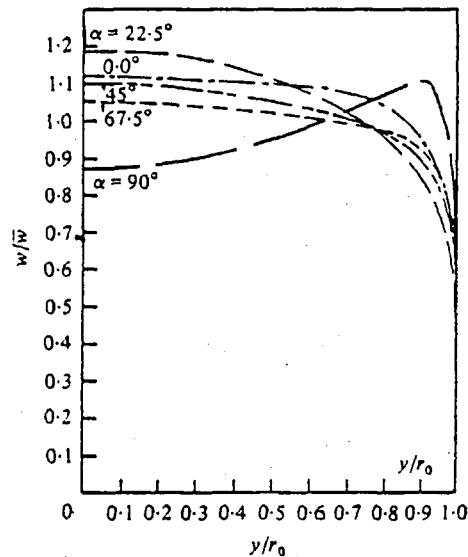


FIGURE 21. Variation of axial-velocity profile with the tube inclination angle (view in the inclined diametral plane; $Re = 25000$, $Gr = 10^{10}$).

18 presents the variation of friction factor, with Gr , for constant Re . Figure 19 shows the fully-developed temperature profiles at various Gr ; figure 20 presents the variation of Nu with Gr . It can be seen from figure 16 that the maximum velocity increases with the increase in Gr , and the velocity profile becomes fuller. It can also be seen that the influence of buoyancy on the turbulent transfer considerably exceeds its effect on the averaged flow. No experimental data are available for verification of these predictions.

3.3. Inclined pipes

The predictions made for inclined pipes are for a fully developed, ascending flow at $Re = 25000$, $Gr = 10^{10}$. No experimental data are available for comparison.

Figure 21 presents the variation of the longitudinal-velocity profiles, in the inclined diametral plane, with pipe-inclination angle. Figure 22 presents the corresponding longitudinal-velocity profiles in the vertical diametral plane. Figure 23 shows the

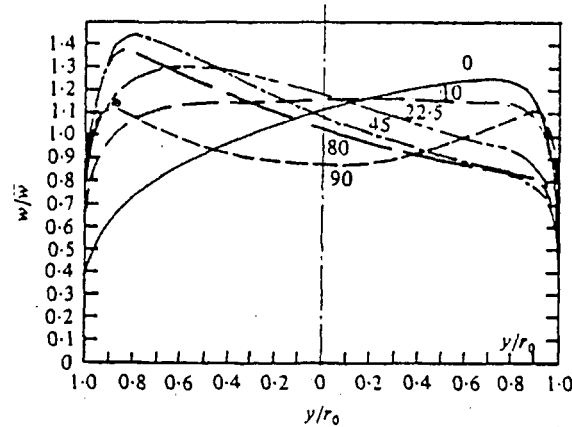


FIGURE 22. Variation of the axial-velocity profiles with the tube inclination angles (view in the vertical-diametral plane; $Gr = 10^{10}$, $Re = 25000$).

variation of the corresponding longitudinal-velocity contours. Figure 24 presents the distributions of Nu on the pipe periphery, at various tube-inclination angles. Figure 25 presents the variation of the average Nu number, based on the average wall temperature, at various tube angles. It can be seen that there exists an inclination angle for which the averaged Nu number has a maximum value.

4. Conclusions

Numerical predictions of flow and heat transfer in horizontal, inclined and vertical pipes with appreciable buoyancy influences, and with uniform heat flux at the wall, have been performed by means of a finite-difference forward-marching procedure. The following were found to be true.

(a) *For horizontal pipes:*

(i) The longitudinal velocity maximum shifts towards the pipe bottom to an extent which increases with Grashof number.

(ii) The results are in agreement with available measurements.

(b) *For vertical pipes:*

(1) *Ascending flow*

(i) For Gr less than some critical value, Gr_c , the effect of buoyancy on turbulence is modest; and its effect on the averaged flow can be neglected.

(ii) For Gr greater than Gr_c , the buoyancy forces affect both the averaged flow and the turbulent transfer.

(iii) For Gr greater than Gr_c , a minimum velocity occurs at the centre-line of the pipe, and a maximum velocity occurs somewhere between the wall and the pipe axis; with increase of Gr the minimum velocity located at the centre-line decreases, and the maximum velocity near the wall increases and approaches the wall.

(iv) The Nusselt number decreases first with an increase in Gr until Gr_c is reached; then it starts to increase monotonically.

(v) The results are in good agreement with available measurements.

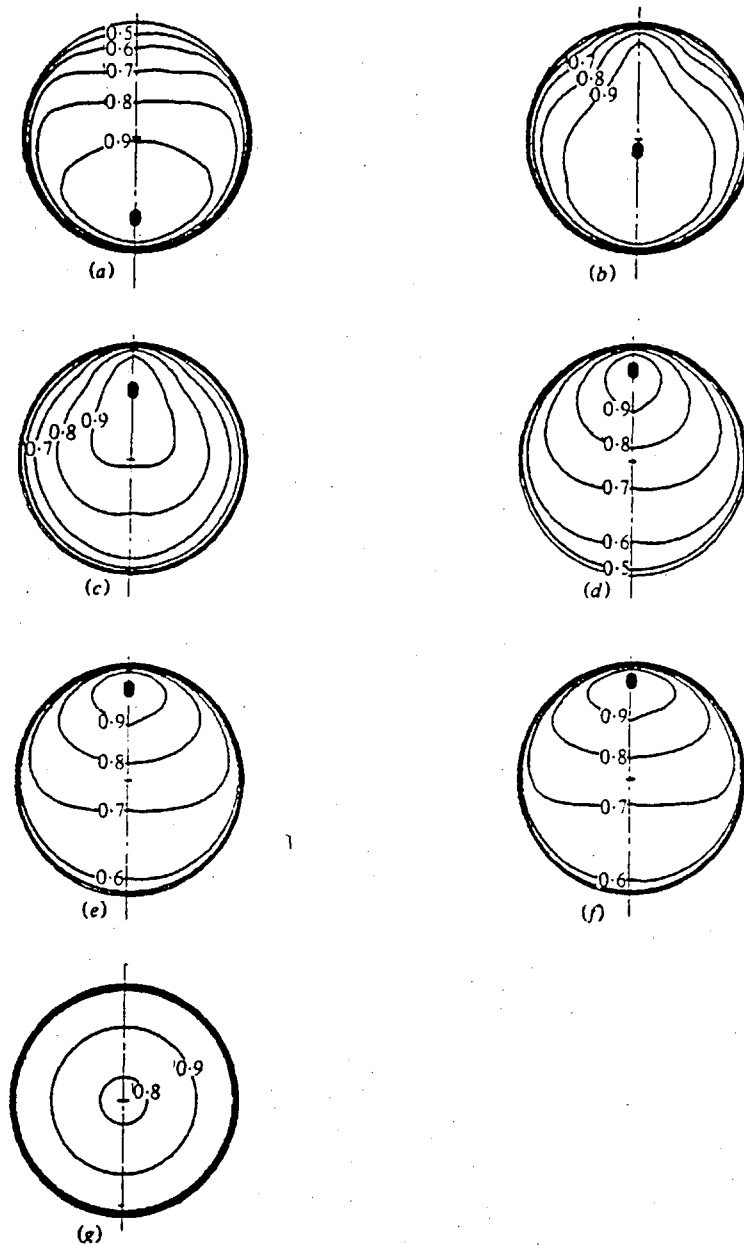


FIGURE 23. The variation of axial-velocity contours with pipe inclination (ascending flow; $Gr = 10^{10}$, $Re = 25000$). (a) $\alpha = 0^\circ$, horizontal. (b) $\alpha = 10^\circ$. (c) $\alpha = 22.5^\circ$. (d) $\alpha = 45^\circ$. (e) $\alpha = 67.5^\circ$. (f) $\alpha = 80^\circ$. (g) $\alpha = 90^\circ$, vertical.

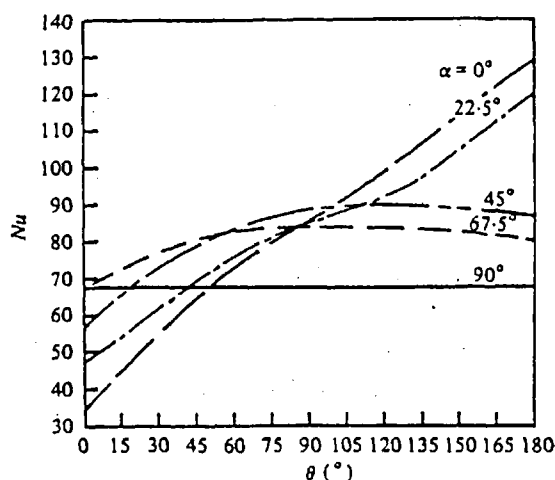


FIGURE 24. Nusselt number distribution on the pipe periphery for different pipe inclination angles ($Re = 25\,000$, $Gr = 10^{10}$).

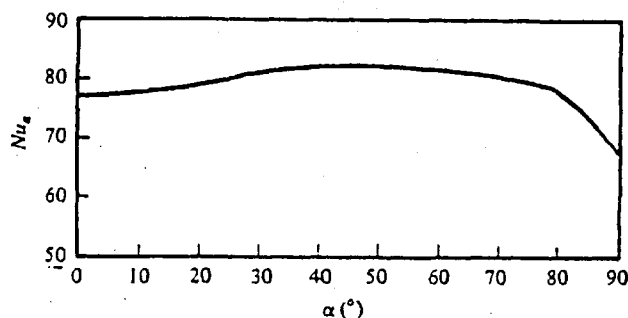


FIGURE 25. Variation of the averaged Nusselt number with the pipe inclination angle ($Re = 25\,000$, $Gr = 10^{10}$).

(2) *Descending flow*

(i) The maximum velocity increases with the increase in Gr , and the velocity profile becomes more full.

(ii) The influence of buoyancy on the turbulence energy considerably exceeds its effect on the averaged flow.

(3) *Inclined pipes*

(i) At least at the Gr and Re values investigated, there exists an inclination angle for which the averaged Nu has a maximum value.

This work forms a part of the research sponsored by the Science Research Council under grant no. GR/A1 01780. The computer program was loaned by Concentration, Heat and Momentum Limited.

REFERENCES

- BUHR, H. O., HORSTEN, E. A. & CARR, A. D. 1974 The distortion of turbulent velocity and temperature profiles on heating for mercury in a vertical pipe. *Trans. A.S.M.E. J. Heat Transfer* 96, 152.
- CARR, A. D., CONNOR, M. A. & BUHR, H. O. 1973 Velocity, temperature and turbulence measurements in air for pipe flow with combined free and forced convection. *Trans. A.S.M.E., J. Heat Transfer* 95, 445.
- LAUNDER, B. E. & SPALDING, D. B. 1974 The numerical computation of turbulent flows. *Comp. Meth. for App. Mech. & Engng* 3.
- OJALOV, M. S., ANAND, D. K. & DUNBAR, R. P. 1967 Combined forced and free turbulent convection in a vertical circular tube with volume heat source and constant wall heat addition. *Trans. A.S.M.E., J. Heat Transfer* 89, 323.
- PATANKAR, S. V. & 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.
- PETUKHOV, B. S. 1976 Turbulent flow and heat transfer in pipes under considerable effect of thermogravitation forces. (Report presented at seminar on Heat Transfer at Turbulent Free Convection.) *Int. Centre Heat & Mass Transfer, Dubrovnik*.
- PETUKHOV, B. S., POLYAKOV, A. F., KULESHOV, V. A. & SHEKTER, YU. L. 1974 Turbulent flow and heat transfer in horizontal tubes with substantial influence of thermogravitational forces. *4th Int. Heat Transfer Conf. Tokyo*.
- PETUKHOV, B. S., POLYAKOV, A. F., SHEKTER, YU. L. & KULESHOV, V. A. 1976 Experimental study of the effect of thermogravitation upon turbulent flow and heat transfer in horizontal pipes. (Report presented at seminar on Heat Transfer at Turbulent Free Convection.) *International Centre for Heat & Mass Transfer, Dubrovnik*.
- POLYAKOV, A. F. 1973 Transient effects due to thermogravity in turbulent and heat transfer. *High Temp.* 11, 1.
- POLYAKOV, A. F. 1974 Development of secondary free-convection currents in forced turbulent flow in horizontal tubes. *J. Appl. Mech. Tech. Phys., PMTF*, 5.
- SEIADARESSIS, D. & SPALDING, D. B. 1977 Prediction of combined free and forced convection in turbulent flow through horizontal pipes. *Lett. Heat & Mass Transfer* 4, 35.