NPS67-79-004

NAVAL POSTGRADUATE SCHOOL

Monterey, California

STATUS OF CENTRIFUGAL IMPELLER INTERNAL AERODYNAMICS:
EXPERIMENTS AND CALCULATIONS

Dan Adler

February 1979

Approved for public release; distribution unlimited

Prepared for:
Naval Air Systems Command
Washington, DC

NAVAL POSTGRADUATE SCHOOL

Monterey, California

Rear Admiral T. F. Dedman
Superintendent

Jack R. Borsting
Provost

The work reported herein was supported by the Naval Air Systems Command, Washington, DC.

This report was prepared by:

Dan Adler

DAN ADLER, Visiting Professor
of Aeronautics

Reviewed by:

Max F Platzer

M.F. PALTZER, Chairman
Department of Aeronautics

Released by:

William M. Tolles

W.M. Tolles
Dean of Research

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

REPORT DOCUMENTATION PAGE

READ INSTRUCTIONS BEFORE COMPLETING FORM

1. REPORT NUMBER NPS67-79-004	2. GOVT ACCESSION NO.	3. RECIPIENT'S CATALOG NUMBER
4. TITLE (and Subtitle) STATUS OF CENTRIFUGAL IMPELLER INTERNAL AERODYNAMICS: EXPERIMENTS AND CALCULATIONS		5. TYPE OF REPORT & PERIOD COVERED Technical Report August 1978-March 1979 6. PERFORMING ORG. REPORT NUMBER
7. AUTHOR(s) Dan Adler		8. CONTRACT OR GRANT NUMBER(s)
9. PERFORMING ORGANIZATION NAME AND ADDRESS Naval Postgraduate School Monterey, California 93940		10. PROGRAM ELEMENT, PROJECT, TASK AREA & WORK UNIT NUMBERS 61153N; N00019-79-WR-91115
11. CONTROLLING OFFICE NAME AND ADDRESS Naval Air Systems Command Washington, DC 20361		12. REPORT DATE February 1979 13. NUMBER OF PAGES 100
14. MONITORING AGENCY NAME & ADDRESS(if different from Controlling Office)		15. SECURITY CLASS. (of this report) Unclassified 15a. DECLASSIFICATION/DOWNGRADING SCHEDULE

16. DISTRIBUTION STATEMENT (of this Report)

Approved for public relaese, distribution unlimited

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

18. SUPPLEMENTARY NOTES

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

Centrifugal Impellers
Internal Aerodynamics

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

Recent developments in internal aerodynamics of centrifugal impellers and related flows are critically reviewed. The overall picture which emerges provides the reader with a state-of-the-art perspective on the subject. Gaps in understanding are identified to stimulate future research. Topics included in this review are: two-dimensional subsonic and transonic inviscid flows, three-dimensional inviscid flows, the structure of turbulence in curved rotating passages and solution of viscous flow problems in impellers.

DD FORM 1 JAN 73 1473 EDITION OF 1 NOV 65 IS OBSOLETE
S/N 0102-014-6601

UNCLASSIFIED

SECURITY CLASSIFICATION OF THIS PAGE (When Data Entered)

Acknowledgements

The research for this report was partly conducted while the author was a Visiting Consultant to Cummins Engines Company Inc., Columbus, Indiana. The study was completed and the report written while he was NAVAIR Visiting Professor at the Turbopropulsion Laboratory of the Naval Postgraduate School, Monterey, California.

The author is grateful for the support he received from Dr. W. T. Lyn, Vice President and Head of the Research Division at Cummins, for sponsorship by the Naval Air Systems Command under the cognizance of Dr. H. J. Mueller, Code AIR-310, and for the considerable help given by Professor R. Shreeve, Director of the Turbopropulsion Laboratory, Naval Postgraduate School, who read the manuscript and added most valuable comments.

Last, but not least, the author should mention the Department of Mechanical Engineering at the Technion, Israel Institute of Technology, Haifa, to which faculty he belongs, for the opportunity given to him to carry out this work.

TABLE OF CONTENTS

Section		PAGE
1.	INTRODUCTION	1
2.	EXPERIMENTAL OBSERVATIONS	2
3.	INVISCID FLOW MODELS	11
	3.1 H-S SOLUTIONS	13
	3.2 B-B SOLUTIONS	19
	3.3 TRANSONIC B-B SOLUTIONS	25
	3.4 3-D INVISCID SOLUTIONS	30
4.	THE STRUCTURE OF TURBULENCE AND TURBULENT VISCOSITY IN ROTATING PASSAGES	38
	4.1 CURVATURE EFFECTS	39
	4.2 EFFECTS OF ROTATION	41
	4.3 COMBINED EFFECTS OF SURFACE CURVATURE AND ROTATION AND THEIR APPLICATION	44
5.	INNER OUTER SOLUTION COMBINATIONS (PATCHING TECHNIQUES)	47
6.	FULLY VISCOUS SOLUTIONS	51
7.	COMPARISON OF COMPUTATION TIMES	61
	LIST OF REFERENCES	63
	DISTRIBUTION LIST	93

1. Introduction

The rising fuel costs put continuous pressure on designers to develop prime movers with reduced specific fuel consumption and smaller weight. In the case of small or medium sized gas turbines the demand for low weight focused increased attention on the centrifugal compressor with the requirement for improved efficiencies to achieve the reduction of specific fuel consumption. In the case of reciprocating engines the demand for reduced weight and reduced fuel consumption led to new concepts, like the turbo-compound and the adiabatic turbo-compound [1] as well as to increased interest in turbo-charging. These, in turn, also increased the attention paid to centrifugal compressors.

Both the gas turbine and the reciprocating engine, have to be efficient over a wide spectrum of points of operation. If variable compressor geometry is to be avoided (because of increased costs, weight, reliability and maintenance problems) the compressor must have a wide range of operation in which efficiencies are good, and surge or choke do not occur. This demand contradicts the requirement for high stage pressure ratios.

The evaluation of the optimal compromise between maximum pressure ratio, maximum efficiency at the design point and wide range of operation with fixed geometry, is a difficult task. Only deeper understanding resulting from improved capabilities for experimental observation and better predictive techniques for the internal flow will enable a fulfillment of this task.

It is the objective of this paper to review the recent developments and the present state of the art of centrifugal compressor internal aerodynamics. In view of the great amount of published material selectivity is unavoidable. Therefore the paper is focused on centrifugal impeller flow. Inlet and diffuser flows, as well as flow in axial machines, are mentioned only as much as they affect the centrifugal impeller flow, or are connected to its flow problems.

2. Experimental Observations

For a long time centrifugal impellers were designed with the concept of a full passage flow. The classical approach of infinite number of blades of zero thickness combined with a correcting slip factor is an example of this approach. However, the full passage flow concept was already challenged as early as 1923 by Alfred Carrard [20] who calculated a neutral zone in the impeller passage. He was probably the first to propose a calculation method for this type of flow which about three decades later was called the jet-wake flow.

The full passage concept could survive for a long period only because experimental insight into the impeller flow was hampered by the fact that it is, unfortunately, rotating. The wake in a rotating impeller was visualized or detected by a number of investigators [3, 4, 5, 6] but no detailed velocity mapping was available until 1957, when Acosta and Bowerman [7] measured the flow field inside a centrifugal pump impeller with backswept blades ($\beta = 23.5^{o}$) using manometer tubes which were rotating with the impeller. They did not detect a jet-wake flow, which probably did not occur in the type of strongly backswept impeller which they used. Their results, therefore, did not challenge the full passage flow concept. A year later, in 1958, Fujie [8] published results measured inside a radial exit ($\beta_2 = 90^o$) centrifugal compressor impeller using a low speed "pneumatic slip ring", or pressure transfer cell. Fujie's experiments were carried out at a number of tip speeds ranging from 37 m/sec to 105 m/sec. Although "pockets" of lower relative velocities were detected in the vicinity of the passage suction side, no well developed wake region, like those found by later investigaters, was detected. This is surprising, but furthermore, Fujie's comparison of the measurements to calculated inviscid irrotational and incompressible velocity profiles (Fig. 14 of ref. 8) revealed that viscous effects now recognized and fully appreciated, were smaller in his results than one would have expected.

Ten years passed, until in 1968, Fowler [9] published results taken in a radial exit impeller at low speed (30 to 60 r.p.m.). Fowler used hot wire anemometers for his measurements. The impeller had to rotate very slowly because a technician, rotating with the impeller in a special cage, monitored the readings. The tip velocity in this experiment is estimated to have been 2.5 m/sec to 5 m/sec (Fowler did not publish the exact geometry and diameter of his impeller), but despite this low tip speed, Fowler detected a flow having the main features of the jet-wake structure. In fact he measured, in all through-flows from fully open flow to 50%

reduced flow rates, a suction side velocity that was much lower than pressure side velocity, - in contrast to inviscid flow predictions. Fowler did not detect the shear layer which separates the wake from the jet, and in which the velocity gradients are exceptionally large. However, his measurements showed that for radial exit centrifugal impellers, even at very low speeds, the inviscid flow models were not able to describe the real flow with sufficient accuracy.

In 1973 J. Moore [10] published his results obtained in a radial exit rotating single passage. The results were reported in detail, including secondary flows, and were the first published data that describe the shear layer between a jet and a wake in a rotating passage. They were detailed enough to allow a quantitative analysis of a jet wake flow. The jet wake structure observed by J. Moore was well developed despite the small peripheral tip speed of about 20 m/sec and the fact that the fluid was pushed through the passage. Moore's experiments were carried out at a number of flow rates, and it is interesting to point out that in contrast to that of Fowler, Moore's jet-wake structure weakened considerably as the flow rate was reduced until it disappeared completely and wake filled up with healthy flow. Unfortunately this weakening of the jet-wake flow with flow rate was not discussed in detail, as it was not consistent with Fowler's findings and with the observations of later investigators. A possible explanation could be that Moore reduced his flow rate more than others did.

In 1975 Howard and Kittmer presented measurements carried out inside a backswept centrifugal impeller ($\beta_2 = 22^o$) with and without a rotating shroud [11]. Their impeller pumped water, and the velocity field was measured with a miniature hot film probe. Like Acosta [9] and in contrast to Moore [10] and Fowler [9], Howard and Kittmer did not detect a jet-wake flow though all these experiments were carried out under imcompressible low speed flow conditions. The full passage flow in Howard's experiments was present in both the closed and the open impeller versions, an indication that in this case the blade tip clearance flow did not affect the flow stability. Acosta [7] and Howard [11] measured in back swept impellers and they did not detect a jet-wake flow. Fowler [9] and Moore [10] measured in radial exit impellers and they found a jet-wake structure. Is this observation significant, taking into account that all these experiments were low speed and incompressible?

Later, in 1976, Eckardt published his paper [12] on the optical measurement of flow in a high speed open radial exit centrifugal impeller, close to its design point. The tip velocity was about 300 m/sec. Eckardt used Schodl's L2F technique [13] and was able to obtain the internal impeller flow field details. His results showed the development of a well established jet-wake structure from an almost uniform inlet flow field into a highly distorted exit flow field. Unfortunately, the geometry of Eckardt's impeller was not published by him, so that his results could not be used for analysis by other investigators. Luckily the missing information was furnished by J. Moore [14] who reconstructed Eckardt's flow passage using available data.

Also in 1975, Mizuki et al published a very detailed report [15] on measurements in an open radial exit centrifugal impeller at a number of points of operation. The impeller rotated at a tip speed of 60.8 m/sec and readings were taken inside the impeller with 4-hole yaw probes and taken out by means of mechanical seals. Further, static pressures were measured along the stationary shroud with high frequency transducers and gas velocities were measured at the impeller exit with a hot wire anemometer. Their results are interesting because like Fowler [9] and Moore [10] they extend over a relatively wide range of flow coefficient from $\varphi = 0.11$ to $\varphi = 0.33$. Unfortunately, these points are not identified on the compressor map so that their relation to the choke and surge lines is not known. Like previous investigators Mizuki et al identified a jet-wake structure, the wake being located as usual close to the suction side-shroud corner, though a little displaced from the boundary. As in Fowler's experiments, and even more so in Moore's, the intensity of the wake was decreasing with reduced through-flow probably because of a general reduction of intensity of the flow field. This was verified by Mizuki both in his internal measurements at the impeller exit and his stationary measurements in the diffuser immediately after the impeller exit. An important conclusion to be drawn from Mizuki's experiments is that the jet-wake flow is verified to persist over a wide range of operation. Another interesting result from Mizuki's work was that a separation bubble was identified in the impeller inlet near the shroud. The bubble did not exist at the high flow rate ($\varphi = 0.33$) but was initially detected at medium flow rates ($\varphi = 0.22$) and became more intense at low flow rates ($\varphi = 0.11$). The separation bubble was not detected at all by Eckardt probably because it did not exist in his apparatus because of the inlet geometry, or because he operated at a higher flow rate.

The passages which were mapped by Fowler [9], J. Moore [10], Eckardt [12] and Mizuki [15] were all radial at their exits. The jet-wake structures detected in these passages were likely the result of a complicated non linear combination of the effects of impeller speed, exit angle of the blades, curvature of the blades, number of blades, existence of a shroud cover, rate of flow deceleration through the impeller and flow rate. The effect of each of these parameters certainly can not be explored in a few experiments. However, a comparison of Fowler [9], Moore [10], Eckardt [12] and Mizuki [15] to Acosta [7] and Howard [11] suggested that the jet-wake flow was most dominantly affected by the sweep-back of the blades. This argument prompted Adler and Levy [16] to measure inside a closed centrifugal impeller with swept back straight blades, in which a full passage flow was anticipated to occur at the design point. To show this was the case (and to explore the feasibility of optical measurements through a rotating window) they carried out laser doppler measurement in a closed backswept centrifugal impeller. They did not find a jet-wake structure. The impeller displayed, as anticipated, a full passage flow. This result led them to hypothesize that the jet-wake flow is weakened up to total disappearance as both impeller speed and blade sweep-back angle are reduced. This view is represented graphically in Fig. (1). As the boundary between the full passage flow and the jet-wake flow is dominantly though not solely influenced by the tip speed and the sweepback angle a border region, rather than a border line is shown in Fig. 1.

Adler and Levy's experiments were the first in which laser doppler measurements were carried out through a rotating window. They showed that a rotating window is not a severe obstacle for laser doppler velocimetry though special attention must be given to window glass quality, flatness and cleanliness.

All the experiments described above concentrated on the measurement of the flow field inside the rotating impeller. Some insight into the flow inside the impeller can also be gained, though to a much more limited extent, from measurements at the impeller exit; and in the case of open impellers, from measurements along the stationary shroud. Examples of this approach, which results in a somewhat simpler experimental set-up can be found in the measurements of Eckardt [17] and Senoo et al [18]. Eckardt [17] carried out measurements at three points of operation of the stage investigated in [12]; one on the choke line, the second at peak efficiency and the third about half way between peak efficiency and surge. He used high frequency pressure transducers on the open stationary shroud as well as a high frequency directionally insensitive impact tube and a hot wire directional probe after the impeller exit. His measurements showed again that the jet-wake flow at the impeller exit persisted over the entire operation range investigated, though it was somewhat reduced with reducing flow rate. Further, a possible separation immediately behind the inlet near the suction side was detected. But in contrast to Mizuki [15] this separation bubble appeared not at low flow rates, but rather at high flow rates. These two results are contradictory and require additional investigation. A third, interesting result of this work was the mapping of total pressure fluctuations at the impeller exit. Inside the wake the fluctuations reached a relative value of 14% to 18% while in the jet they were less intense, having a value of 10% to 12%. This may throw some light on the distribution of losses and turbulent viscosity inside a rotating passage.

A recent work was published by Senoo et al [18]. Using stationary shroud taps to analyze the flow in a supersonic impeller, they were able to establish the shock wave pattern inside the impeller. In addition, they discovered a new longitudinal slip band occurring in the subsonic flow regime. This slip band was located in the middle between the blades, and its thickness increased with flow rate reduction. It became thinner as flow rate was increased until it disappeared altogether. Senoo did not give an explanation for this flow behavior, nor did he comment on its effect on compressor performance. Evidently this newly discovered slip band should be further explored in more detail using optical techniques, and correlated with impeller characteristics.

The experimental observations reviewed here lead to a number of imported conclusions:

1. The larger the rotor exit flow angle (β_2) and the higher the rotor speed, the more likely is the development of a jet-wake flow towards the impeller exit.
2. The jet-wake flow when present at the design point, exists also over a wide range of operation points.
3. The jet-wake flow departs so much from inviscid flow predictions that inviscid flow calculation are representative only for small β_2 impellers, or in case of large β_2 impellers, for inlet and mid passage regions only.
4. Small β_2 impellers generally deliver a more uniform flow field into the diffuser. This could explain their better efficiencies and wider range of operation between choke and surge.

3. Inviscid Flow Models

The flow in conventional centrifugal impellers, as described in the previous section, is extremely complex. Clearly, the complexity is partly the result of not being able to design the impeller aerodynamically such that the real fluid behavior is properly accounted for. Improved aerodynamic design procedures and better performance predictions, however, both require the development of techniques to properly analyze the flow through the impeller. Such an analysis must contend with the adverse pressure gradient which is imposed on the flow, and the presence of turbulent viscosity variations which are strongly influenced by mean flow conditions, curvature of the stream lines and the Corriolis acceleration. As will be discussed later, a comprehensive mathematical model describing reality is too difficult to solve with the current state of the art. The simplest approach certainly, is to calculate the flow field in the impeller by assuming that the fluid is inviscid. Even with this assumption the task is formidable, and raises the immediate query as to whether the effort is justified in view of the big differences obtained sometimes between inviscid predictions and measured results [9, 10, 12, 18]. The answer however is affirmative: inviscid solutions can be useful in describing backswept impeller flow where the jet-wake flow does not exist at the design point, in describing the upstream regions of radial exit impellers and generally to predict the inviscid uncorrected core of any intra-impeller flow.

Existing methods for the prediction of inviscid centrifugal impeller flows are classified into four groups I) Solutions on the

Hub-to-Shroud (H-S) stream surface, II) Solutions in the Blade-to-Blade (B-B) stream surface, III) Transonic B-B Solutions and IV) Three dimensional (3-D) solutions. In each of these groups models can be reclassified into 1) streamline curvature methods, 2) partial differential methods.

1) Streamline curvature methods are based on a first order ordinary differential equation describing the force balance in the direction of the quasi-orthogonals to the streamlines. The equation is solved along these quasi-orthogonals marching forward along the passage from inlet to exit. The continuity equation is integrally satisfied along quasi-orthogonals. This is basically a non elliptic procedure. The elliptic character of the flow is retained somehow by the curve fitting of the streamlines which link upstream to downstream points along the passage. The curve fitting also determines the streamline curvature. The accuracy of the results and the ellipticity of the solution depend strongly on which curve fitting method is used; this is the "non natural" or "artificial" ingredient of the streamline curvature technique. The non-elliptitity of the technique also lies in the indirect manner in which downstream boundary conditions and backpressure are satisfied, and in the way their upstream effects on the entire flow field are predicted.

An early and extremely simple example of this approach is the method of Adler and Ilberg [19] where the assumption of linear variation of streamline curvature from hub to shroud along the quasi-orthogonals allowed integration of the equilibrium equation in the direction of the quasi-orthogonals. The method enabled the flow field to be obtained without the use of a computer.

2) Partial differential methods are based on the solution of the partial differential equation of the inviscid steady flow. Momentum equilibrium and continuity are both satisfied at a point, and the elliptic nature of the subsonic flow is well preserved both in the form of the partial differential equation and, consequently in the way the boundary conditions can be specified. The formulation is based in most cases on the classical work of Wu [20] or on equations similar to those given by Wu. Wu defined "special derivatives" valid only on H-S or B-B stream surfaces. These special derivatives allow an enormous simplification of the problem by splitting the 3-D formulation into two mathematically "two dimensional" formulations. These are, however, interconnected by the "stream sheet thickness" thus retaining the three dimensionality of the physical situation. The partial differential equations can be solved either by finite differences or by finite element methods. These will be described later.

After having briefly explained the streamline curvature and the partial differential approaches let us continue the review along the classification into H-S, B-B and 3-D groups. Due to the large number of methods known, only a brief description can be given here. After this the various methods will be compared to each other and to experiments.

3.1 H-S solutions: Historically, streamline curvature methods were the first to be developed. They required more modest computer storage and simpler computational techniques were needed because they involved only first order, ordinary differential equations. A frequently used streamline curvature program is Katsanis' early development [21]. Katsanis expressed the equation of motion of the H-S flow in terms of curved coordinates based on the streamlines and quasi-orthogonals. The resulting equation was solved simultaneously with the continuity equation in its integral form along the quasi-orthogonals. The numerical solution iterated on the constant mass flux between neighboring streamlines. The curvature of the streamlines was modified from iteration to iteration until convergence was achieved.

An additional method based on the streamline curvature technique was developed by Novak [22, 23]. As in the previous method the equation of motion was expressed in terms of the streamline curvature and the solution iterated on the constant mass flux between two successive streamlines. Unlike Katsanis, Novak modified his initially assumed streamlines along radii, rather than along quasi-orthogonals. This simplified the method but restricts its application to axial flow machines only. Davis [24] compared both Katsanis' and Novak's methods to a method developed by himself. He concluded that all three versions of the streamline curvature method were essentially identical. A general analysis of the streamline curvature approach can be found in a paper by Smith [25].

As mentioned above the partial differential methods are based in most cases on Wu's formulation [20] or its derivatives. An early finite difference solution was given by Wu [26], and later Marsh [27] used a modified version of Wu's model to obtain a solution of the flow field in the H-S surface. Later, in 1973, Davis published his technique [28] which was essentially similar to Marsh's. It was developed for axial machines but could be applied also to centrifugal compressors in an analogous way to Katsanis' and McNally's evolutionary work. Katsanis and McNally developed a finite difference method [29], which in its early version was limited to flows up to 45^o from axis. Their later version [30] extended the application to radial flows as well. Both methods, those of Davis and Katsanis and McNally, were based on the use of the stream function equations. The equations were solved with a finite difference technique, and the basic method was limited to subsonic flows. Locally supersonic flows could only be handled by a combination of the stream function solution and a velocity gradient (streamline curvature) technique [42].

A second way to solve the partial differential equations of the H-S flow is the finite element technique. The basic equations solved are essentially identical to those used in the finite difference solutions based on Wu's model, except for modifications introduced for the convenience of the technique. The first finite element solution was completed in 1974 by Adler and Krimerman [31]. This was followed in 1975 by Oates et al [32] and in 1976 by Hirsch and Warzee [33]. A basic difference between the finite element and the finite difference techniques is that the discretization of the continuous flow field in finite difference form is geometrically restricted by the coordinate system chosen for the problem (unless complicated and time consuming transformations are applied) while in finite element form no restriction is imposed on the location of the nodal points in the grid. The freedom in location of grid points enabled Adler and Krimerman [31] to locate their grid points on streamlines. This allowed a simpler satisfaction of the energy equation which, along streamlines, could be used in its integrated form. Further, the fact that grid points were located on streamlines made the incorporation of the solution into a fully 3-D solution easier. It was with this recognition that Adler and Krimerman choose the finite element technique for their solution.

Oates et al [32] applied a variational principle, producing, in effect, the integral of the H-S momentum, to create a system of non-linear algebraic equations. The system of equations was solved using the Newton-Raphson technique. To formulate their method Oates et al used the well-known actuator disc method. The technique was fundamentally different from that of Adler and Krimerman who did not use a variational principle specially derived for their principal equation. Rather, they used the variational principle of Poisson's equation, giving their governing equation an artificial shape of a Poisson equation. This in turn required an iterative procedure. Further Adler and Krimerman did not use the actuator disc approach at all.

Hirsch and Warzee [33] used a weighted residual Galerkin procedure. Like Adler and Krimerman they gave their governing equation the shape of a Poisson equation, avoiding in this way the problem of deriving a variational principle for their governing equation. They also introduced NASA correlations into their technique to specify losses and deviation angles. It should be pointed out here that all these techniques were initially applied to axial flow compressors. There is, however, no special reason to believe that these techniques will fail when used to predict the flow in centrifugal impellers. In fact, Adler and Krimerman used their technique [31] successfully to analyze Eckardt's impeller [12] without any difficulties. This will be discussed later.

No systematic comparison of the simultaneous application of all these techniques has been reported. Further, no systematic comparison of the predictions of the various methods with the results of experiments has appeared. There are, however, isolated comparisons between two or three techniques or between a numerical solution and an experiment to be found in literature. Davis compared his streamline curvature method to those of Katsanis and Novak and found them to be essentially identical [28]. In 1972 Frost [34] presented a streamline curvature technique and compared its results to Marsh's method [29]. The agreement between the two techniques was very good but both do not agree too well with experimental results (see Fig. 2). Differences were observed not only in velocity magnitude and direction but also in velocity profile shapes, indicating a fundamental problem; perhaps the effect of viscosity?

Adler and Krimerman [31] compared their results calculated using a finite element method to Wu's results [26] evaluated with a finite difference method. The two results agreed fairly well and showed the same trends in the distributions of velocity and static pressure along the casing and the hub. When compared to experimental data both analytical techniques gave static pressure distributions showing trends similar to the measured profiles (see Fig. 3). Adler and Krimerman's results were closer to the measured profile when blade thickness effects and a total loss coefficient were included in the calculations.

In 1974 Davis and Millar [35] reported a comparison between finite difference and streamline curvature methods. Their comparison is illustrated in Fig. 4 where test results of a NASA single stage transonic compressor rotor are also shown. The two calculations predict similar velocity, pressure and total temperature distributions and agreement with experiments is reasonable. Finally in 1976 Hirsch and Warzee's finite element method was compared to experimental data obtained in a single stage NASA compressor (see Fig. 5). Agreement seems to be exceptionally good for an inviscid theory.

A final word of explanation and caution should be added here. Most of the H-S flow prediction methods mentioned so far and the comparisons discussed were carried out for axial flow machines (Fig. 2-5). The methods themselves, however, are of interest to centrifugal impeller flow studies (see footnote*). But it should always be kept in mind, that three dimensional and viscous effects in centrifugal impellers are considerably more significant than in axial rotors, especially if a jet-wake flow is present. Therefore,

---

*For example the method of Ref. [31] was used unaltered in a centrifugal radial exit impeller.

in the case of conventional centrifugal impellers inviscid H-S techniques will give results which are further from reality than the comparisons in Figures 2 to 5 would suggest. Further, all the H-S methods reviewed so far assume axisymmetric flow. In the higher loaded centrifugal impellers this assumption could lead to big discrepancies as will be shown later in this paper.

3.2 B-B Solutions: As in the case of the H-S surface solution the formulation of the B-B stream surface flow solution is somewhat artificial in that the assumed surface does not exist physically in the centrifugal impeller. The B-B solutions, however, serve to analyze trends required to be known for passage optimization rather than predict exact flow fields. Further, they can serve as essential "building blocks" in the development of more complete 3-D and viscous methods. Basically, the two groups of solutions on the H-S and the B-B stream surfaces have much in common. However, the most pronounced difference between the two is the lack of well defined boundaries at the inlet to, and the exit from the blade passages on the B-B surface. These boundaries are given on the H-S surface by the hub and shroud extensions in the upstream and downstream directions, and are therefore considered to be input data. In constrast, the boundaries, in form of the leading - and trailing edge stagnation streamlines, are determined on the B-B surface in the course of the solution.

An approach to the B-B problem not used on the H-S surface is the method of singularities applied in all cases on a surface of revolution [36, 37, 38, 39, 40, 41]. The surface is the center of a stream sheet which, in reality, is of varying thickness because of continuity with compressibility. The method has problems at the leading edge and the trailing edge, and the ways to overcome them are not always general enough. Martensen's [36] and Ogawa's [40, 41] methods, for example, develop difficulties when the blades are very thin (a common case in centrifugal impellers) or when the trailing edge is thick and rounded (again common in centrifugal impellers). It is reported that Wilkinson [37] was able to solve this problem; however, for an axial machine. In a centrifugal compressor, where viscous effects on deviation angle (and slip) are more significant, this problem is not yet solved. A severe problem in the application of the singularities technique in centrifugal impellers is the strongly varying streamsheet width. Ogawa and Murata [41] solved this problem by approximating the width variation by an analytical expression and, in a more exact manner by a numerical solution of their equations. From their work, it appeared that in spite of a large stream sheet variation, the solution yielded acceptable results.

As on the H-S surface, the streamline curvature method can be applied to the B-B surface flow prediction. It has been used less often however, probably because of the stagnation streamlines problem. One of the first methods was developed by Katsanis [42]. The method was based on an equation for the velocity gradient along quasi-orthogonals between the blades and was similar to the method of quasi-orthogonals on the H-S surface. The solution was carried out on a surface of revolution generated by rotating a previously calculated H-S streamline around the axis. Later, Katsanis used this streamline curvature technique in the supersonic regions of a transonic finite difference method which he developed. Wilkinson also developed a streamline curvature method [43]. He gave much attention to the location and shape of leading- and trailing edge stagnation streamlines, which is essential in this type of calcualtion. Wilkinson, who carried out basic work on streamline curvature techniques, their accuracy and convergence characteristics as affected by the curve fit procedures [44], was able to optimize the techniques as to the number of iterations required until convergence by deriving an optimum damping factor. Comparison of his results with experiments is given in Fig. 6.

Most frequently used in the B-B flow problem solution are finite difference techniques. Stanitz, a pioneer in this field, published a finite difference method as early as 1948 [45, 46, 47]. He determined the flow field by solving stream function equations. Stanitz also calculated off-design B-B flow fields and predicted pressure side separations (see Fig 7) which, as is now well known, do not exist because of the important effect of viscosity on flow field. Stanitz's results illustrate clearly that inviscid methods are not able to predict, even approximately, off-design flow fields on the B-B stream surface. They should be used only at design point, and even here they will not be able to predict the jet-wake flow which has been measured by many investigators.

In 1968 Katsanis published a finite differences method to predict the B-B flow field [48]. Like most B-B flow calculation techniques, Katsanis' also suffered from the inaccuracy introduced by the sensivity of the results to the location and shape of the leading edge stagnation streamlines. To improve the accuracy of the prediction and to enable more detailed information in this critical part of the flow field, Katsanis and McNally [49] developed a method to predict the details of the flow field near the leading edge of the blades. They further developed their technique to handle splitter blades in centrifugal compressors [50]. The programs of Katsanis and McNally are popular and widely used but they also have the deficiency from which all inviscid B-B programs suffer; they can not predict the jet-wake flow, which is essentially a viscous phenomenon. This is a severe drawback of all inviscid B-B methods.

To complete the list of finite difference methods, the work of Smith and Frost should be noted [51]. Smith and Frost developed a finite difference technique based on Wu's formulation [20]. They programmed a solution using Marsh's scheme [27], and also programmed a streamlind curvature technique similar in its general features to those already mentioned. Using the two computer programs, Smith and Frost compared the two techniques by carrying out calculations for identical problems. In the case of an axial compressor a fair degree of agreement was achieved, except on the suction side leading edge where the velocity distribution computed by streamline curvature was oscilating. Smith and Frost believed that a finer mesh would have eliminated this problem. In the case of an axial turbine the streamline curvature technique behaved badly, and velocity oscillations were encountered both at the leading and the traling edges, caused probably by difficulties in evaluation of streamline curvatures. This problem did not exist in the finite difference solution. Generally it can be said that the streamline curvature technique required less computer storage than the finite difference technique, but the computing time was longer. In contrast to the finite difference method, the streamline curvature technique was not restricted by the local Mach number and could therefore be used in cases with local supersonic regions.

A general problem in numerical techniques is how to optimize convergence,accuracy and computation time. To demonstrate that the convergence and execution time of early programs can probably be improved in many cases, one can cite Deshpande's work [52]. Deshpande developed a new successive overrelaxation algorithm to improve Katsanis' original algorithm [48]. The new algorithm reduced execution time by a factor of 2.5 for a typical blade configuration. Further, Deshpande's algorithm enabled convergence for cases in which Katsanis' algorithm did not converge at all. Application of Deshpande's new algorithm in Katsanis' programs [48, 49, 50] required only minor modifications and is reported by Deshpande to have reduced computing costs considerably.

Davis and Millar [51] showed very clearly how complex the finite difference stencils can become in cases of unusual boundary geometry or with irregular grids imposed by the coordinate system. This problem can be removed by using the finite elements method.

Blade-to-blade finite element methods were published by Adler and Krimerman in 1977 [54], followed by Prince in 1978 [55]. Adler and Krimerman's approach was very similar to that taken in their H-S solution [31]. The main difference was that special care was taken to predict correctly the leading-and trailing edge stagnation streamlines. This was done by imposing the conditions of periodicity and constant angular momentum on the approaching flow field. The downstream stagnation streamline was evaluated using the Kutta-Jukowski condition and flow field periodicity. As in the H-S finite element solution, the B-B solution took advantage of the freedom to choose the location of grid points. Consequently, grid points were always located on streamlines, thus allowing again the use of the integrated energy equation. The model solved was Wu's B-B formulation [20], which was again given the artificial form of a Poisson equation to overcome the lack of a variational principle to Wu's principal equation. A comparison of Adler and Krimerman's results with experiments and with calculations by Stanitz, is shown in Fig. 8.

Prince [55] followed a somewhat different approach. He used a Galerkin variational analysis to satisfy the continuity equation for steady potential flow. He then used a Newton-Raphson method to solve the resulting system of nonlinear algebraic equations. Prince used continuity conditions to evaluate the upstream and downstream stagnation streamlines. To do this he used a distribution of stream sheet thickness which was required to be known a priori. Prince's calculated results compared well with experiments as is shown in Fig. 9.

3.3 Transonic B-B Flows: The case of transonic B-B flow is treated separately because the nature of this type of flow changes from elliptic when the Mach number is less than unity to hyperbolic when the Mach number is greater than unity. The difficulty in transonic solutions is that elliptic flow equations are solved with numerical techniques which differ considerably from the numerical techniques used in the solution of hyperbolic partial differential equations. The problem becomes even more complicated because the boundaries between the elliptic - and the hyperbolic regions of the flow (on which the equations are parabolic) is not known a priori but must be determined during the computation.

An early transonic solution was again derived by Katsanis [56]. Katsanis' method can be applied to any impeller geometry (axial or centrifugal). The solution was obtained as a combination of a finite difference technique, a stream function solution and a velocity gradient method. The finite difference solution, carried out at a reduced flow rate, provided information required for the velocity gradient method. For flow which was totally subsonic the finite difference method was used. In supersonic parts of the flow field, the streamline curvature technique was employed. Katsanis used different numerical techniques to obtain the solution in the different regions which required automatic detection of the region type and, accordingly, a switch over from technique to technique. These complications could be overcome if a single solution could handle both sub- and supersonic regions of the transonic flow field.

Although the equations for steady flow are hyperbolic for supersonic flows and elliptic for subsonic flows, for unsteady flow the equations are hyperbolic both for super- and for subsonic flows. Therefore, if the time dependent equations of continuity, momentum and energy are used, a single solution technique is applicable for both types of flow. The steady flow, including both super- and subsonic regions, can then be regarded as the final steady state of the time dependent flow. This concept leads to a unified method used identically in the two flow regimes. Generally an approximate flow field is used as an initial condition and then the equations are integrated forward in time until the final steady solution is reached. The technique is known as the "time marching" method. In some cases, severe stability problems are encountered. They can often be solved, however, by reducing the time steps or by using an artificial viscosity which has a damping effect.

In 1971, Marsh and Merryweather [57] described a stable time marching technique which was based on finite differences and did not rely on the use of artificial viscosity to achieve stability. It was found that several stable procedures could be developed for flow in convergent-divergent nozzles. The characteristic feature of the stable schemes was that the derivatives of all quantities other than pressure were approximated by backward differences, while the derivative of pressure contained a forward element. The computer program developed by Marsh and Merryweather [57] was relatively slow, some 2700 iterations being required to obtain a solution to an accuracy of 0.01 per cent. Further work by Daneshyar and Glynn [58] was based on the method of characteristics which led to a much faster method of calculation. The method has been extended by Glynn to deal with cascade flows.

In 1971, McDonald [59] used a time marching method to calculate the pressure distribution around aerofoils in cascade. The problem was formulated in terms of a finite area approach which led to the conservation equations in an integral form. The flow was assumed to be isentropic on the grounds that only weak shocks were normally encountered in cascades. McDonald obtained very good agreement between his calculated pressure distribution and that measured experimentally in a cascade. The use of the isentropic flow assumption is interesting in that Marsh and Merryweather had tried the same assumption for purely subsonic flows and had experienced a severe numerical instability. In impellers, where the flow is certainly not isentropic, these methods [57, 58, 59] have to be modified.

Also in 1971, Gopalakrishnan and Bozzola [60] published a time marching transonic technique. They used a predictor-corrector, two step, time splitting method to solve the partial differential formulation of the transonic flow problems. The method had stability problems unless the time step was smaller than a value predicted through a linear analysis, but even this was not always sufficient to give stable computation. To overcome this difficulty, a spatial smoothing procedure could be used. Excessive smoothing, however, caused inaccuracies in the results, as would be expected, and the problem then is the prediction of the negative effect of the smoothing.

In 1974, Denton [61] proposed a time marching scheme for cascade flows using a simpler grid than that of McDonald. Denton's grid consisted of quasi-streamlines and straight lines across the blade passage. The conservation equations for mass, momentum and energy were derived for a control volume. Instead of assuming isentropic flow, Denton assumed constant stagnation enthalpy, an assumption which became exact when the solution converged to the steady state flow. In Denton's scheme, the pressure at the central point of an element was assumed to act on the upstream face of the element, whereas the velocity at the center controlled the flow through the downstream face. The maximum time step for this scheme was found to be larger than for the method of Marsh and Merryweather [57].

Denton has applied his time marching method to calculating the blade-to-blade flow in several cascades and has obtained encouraging results. He has also extended the method to three-dimensional flows, although this did require a large amount of storage in the computer. The predictions obtained with this program have been compared with experiments performed with a rectangular duct having 60° of turning. Good agreement was obtained between the calculated and experimental pressure variations for the four corners of the duct. It should be possible to extend the time marching scheme to deal with three-dimensional flow in cascades.

It is noted again that Denton's technique and some of the other methods mentioned were developed for axial flow stator cascades. The methods can, in principle, be modified to handle the flow in centrifugal impellers. It simply has not yet been done. A computational technique to solve the flow in axial rotors was developed in 1975 by Kurzrock and Novick [62] in an approach similar to that of Gopalakrishnan and Bozzola [60]. They solved the Navier-Stokes equations using the time marching technique. The method was based on a time splitting finite difference technique, using a two step numerical evaluation of the time derivatives of the partial differential equations, together with a predictor-corrector technique. The spatial derivatives were numerically evaluated by backward differences for the predictor step and by forward differences for the corrector step. Figure 10 illustrates the good agreement found between Kurzrock and Novick's results, and experimental measurements.

A completely different approach to the problem was introduced by Sobieczky, Fung and Seebass [63]. Their method was developed for single airfoils but can be extended to stationary or rotating cascades [64]. Sobieczky et al developed a technique which enabled the design of shock free transonic aerofoils, in principle eliminating the losses associated with shocks and shock-boundary layer interactions. Their procedure was based on the use of any numerical method for subsonic, compressible isentropic flow. The algorithm of the selected method was modified so that if the flow became hyperbolic, the basic equations in that region were altered, using a "fictitious gas" concept, such that the system of equations reverted to elliptic behaviour. In this way the complete flow field over a given configuration was first calculated using an elliptic procedure. The calculation served to define sonic surfaces.

Outside the sonic surface the solution obtained satisfied the correct unmodified equations, and the potential at infinity had the correct value for the circulation. The second step in the procedure was to modify the surface geometry inside the sonic surface, using a hyperbolic procedure for the real gas, to generate a shock-free supersonic flow with the same conditions along the sonic line. Here a problem arises as it is well known that shock free, two or three dimensional irrotational near-sonic flows are physically and mathematically isolated. Any small change in flow - or boundary conditions might in practice lead to the formation of shock waves. However, if the shock waves are very weak the method will have practical significance. While Sobieczky's method was developed for inviscid flow, the same procedure can be employed iteratively with a boundary layer calculation to achieve shock free viscous designs.

3.4 3-D Inviscid Solutions: The prediction techniques previously described were all two dimensional. Their relevance to describing real impeller flows must be questioned, especially in the case of highly loaded centrifugal impellers where three dimensional effects are mush more significant than in axial rotors or cascades. The answer to this question can be obtained in two ways: 1) comparison to experiments, 2) comparison between 2-D and 3-D solutions both carried out for a given impeller at the same operating conditions. First, currently available methods which can predict 3-D centrifugal impeller flow (or related axial rotor flow) will be reviewed.

An early attempt to predict the 3-D inviscid impeller flow was made by Schilhansl [65]. The work was restricted to incompressible flow and was developed with the severely restrictive assumption that all stream surfaces were surfaces of revolution. The method was based on cascade theories then available in 1965. In fact the method of singularities used by Schilhansl restricted the application even further to irrotational flow, a severe limitation in turbomachinery flows. No results were reported in Schilhansl's paper [65], but the work illustrated very clearly what the difficulties were in the solution of 3-D flows. Six years later, in 1971, Katsanis published a 3-D compressible flow prediction method [66]. Katsanis used the ordinary differential equations of the streamline curvature technique, which define the values of velocity gradients along quasi-orthogonals. The continuity equation was integrally satisfied as is normally done in the streamline curvature methods. Katsanis solved the H-S and the B-B velocity gradient equations simultaneously, with the condition that either the weight flow was specified or that the flow was choked, to determine the velocity distribution over the blade surfaces. The method incorporated a number of assumptions, some of which could affect considerably the accuracy of the results in extreme geometries or flow conditions. The most restrictive assumptions were, 1) that there was either a linear variation of curvature or of radius of curvature along an orthogonal, 2) that there was no change in radius along a B-B orthogonal, and 3) that the flow angle along a B-B orthogonal was constant.

In 1972 Senno and Nakase published a quasi 3-D method [67].

The method included again the limitation that the blade-to-blade stream surfaces were axisymmetric. (In reality these stream surfaces are highly twisted and distorted.) Senoo and Nakase's method was based on two interacting procedures; one, a streamline curvature H-S procedure and the other a partial differential blade-to-blade technique developed by Senoo and Nakase a year earlier [68]. To achieve a final solution the method iterated from H-S to B-B and back, but the B-B stream surfaces were unaltered during the process and remained surfaces of revolution. The H-S method was rather conventional while the B-B solution was based on Prasil's independent variable transformation. The transformation simplified the method, but required the B-B stream surfaces to be axisymmetric. A comparison between Senoo and Nakase's calculated results and experiments is given in Fig. 11.

Six years later, two further works were published; Bosman and El-Shaarawi [69], and Novak and Hearsey [70]. Bosman and El-Shaarawi used Wu's approach to provide the mathematical model. They worked iteratively with the solutions obtained alternately on the H-S and B-B stream surfaces. However, the B-B surface was again always a surface of revolution and the H-S surface was calculated for an averaged mass flux. In reality the H-S surfaces and the mass flux through them change considerably across the passages. Thus their method was not truly three-dimensional. The solution of the principal equations was obtained using Marsh's matrix inversion method rather than by a relaxation method, because comparative studies carried out by Hill [71] and Bosman [72] showed that the matrix inversion method was considerably faster than the relaxation method. No comparison with experimental results was given in Bosman and El-Shaarawi's paper. But the differences between a 2-D computed flow field and their quasi 3-D results are demonstrated to be significant enough in the case of a typical centrifugal compressor impeller to justify the more elaborate quasi 3-D computation.

Novak and Hearsey [70] also used two 2-D methods, one on the H-S surface and the other on the B-B surface to generate quasi 3-D results. Both 2-D component methods used the streamline curvature technique. The two programs were coordinated to yield a quasi 3-D solution, but as in the previous example, the B-B surfaces were surfaces of revolution, and the H-S surfaces were stream sheets roughly parallel to the blades. In other words, the H-S surfaces, which in reality vary from pressure side to suction side of the rotating passage, are represented in Novak's and Hearsey's work by a single surface. Comparisons with experimental measurements were given for an axial nozzle passage and an axial compressor cascade, and agreement was good. Centrifugal compressor flows were not analyzed, and therefore the accuracy of the technique in this case of more distorted flow is not known.

In 1978 two new 3-D inviscid methods were published, Hirsch and Warzee [73] and Krimerman and Adler [74] developed quasi 3-D methods based on Wu's approach. The method of Hirsch and Warzee [73], again used H-S and B-B formulations which were combined iteratively to yield a quasi 3-D flow. In Hirsch and Warzee's method the interaction between the H-S and the B-B solutions was stronger than in the previous methods because the stream sheet thickness and the streamline angles were transfered from the H-S solution to the B-B solution. Further, the fluctuation terms (as defined by Hirsch and Warzee [33]) were determined by a number of successive B-B solutions and then transfered to the H-S calculation. This was a step toward a more three-dimensional calculation; but nevertheless the B-B stream surfaces were always axisymmetric and the H-S solution, although corrected by computed fluctuation terms resulting from the B-B results, was still carried out on a single surface. As in their first method [33] Hirsch and Warzee used the finite elements technique for their solution.

Krimerman and Adler's method [74] was based on H-S and B-B solutions which they developed earlier [31, 54], and used the finite elements method of solution. However, in contrast to the previous methods in Krimerman and Adler's approach the B-B surface was not required to be axisymmetric and the real stream sheet thickness was taken into account as a spatially three-dimensional function. Further, the H-S flow was not represented any more by a single stream surface but was calculated on an arbitrary number of H-S surfaces which varied in shape from pressure side to suction side across the passage. Krimerman and Adler solved Wu's model, which they modified slightly. They began by calculating a single mean H-S solution. Then they calculated a number of axisymmetric B-B solutions between the hub and the shroud. As a third step, a number of H-S solutions were carried out, giving the circumferential variation of the flow field from pressure-to suction side. The fourth step was to calculate a number of B-B solutions in which the real variation of the H-S flows from pressure side to suction side of the passage was accounted for. Steps three and four were iteratively repeated until convergence was achieved. In this method [74] all the quantities and stream surface geometries were truly three-dimensional. No restricting assumptions about stream surface geometries are made and no pitch averages or fluctuation terms were required. The results were restricted only by the inviscid nature of the fluid and by the fact that corner streamlines were fixed by the boundary conditions to be identical with the geometrical passage corners. In reality streamlines can wrap around the geometrical corners of the passage. Figure 12 shows a sample of the results obtained by Krimerman and Adler for the flow in a centrifugal impeller. In Fig. 12, the intersection lines of quasi-orthogonal planes and the computed H-S and B-B surfaces are shown for four selected planes. The departure of the B-B surfaces, computed by the 3-D method from the axisymmetric B-B surfaces used previously is seen to be considerable. Figure 12 also shows the variation of the H-S stream surfaces from pressure side to the suction side of the passage. In previous methods this variation was either not taken into account or it was represented by pitch averages and fluctuation terms. Figure 13 shows how the method converged from the initial steps to the final 3-D solution.

The comparisons between the predictions of various inviscid techniques and the results of Figs. 3, 4, 5, 6, 8, 9, and 10 are encouraging. Only in Fig. 2 and Fig. 11 are discrepancies clearly evident. The comparison shown in Figure 14 fits well into this encouraging scene except for the data near the impeller exit, where the measured average pressure is far from the prediction. Is this a failure of the analysis or a measurement error? This question must be examined with greater care.

Adler and Krimerman [75] also used the technique described in ref. [74] to calculate the flow in Eckardt's impeller [12], using the geometry reconstructed by Moore [14]. The results for some selected blade-to-blade sections in the center of the pasage are given in Fig. 15. The predictions appear to be good except perhaps near to the exit. However, Fig. 16 gives the comparison between calculations and measurement in the same impeller but close to the shroud where Eckardt detected the wake. Here the prediction is seen to be poor. The reason for the poor agreement between calculation and experiment seen in Fig. 16 and at the exit regions in Fig. 14 and Fig. 15, is almost certainly the effect of viscosity which was eliminated in the analytical models.

The failure of the inviscid models to predict the flow in centrifugal impellers is manifested when a jet-wake flow is present. If the classification of Fig. 1 is supported by additional experiments it could be used also to specify classes of flows for which inviscid models are acceptable and those for which inviscid models fail. The situation here is quite different from that found in axial flow machines and is reminiscent of the inadequacy which is found in applying inviscid models in cases of separated flows. The wake is probably similar to a separated flow, although different in that it must be influenced by the strong centrifugal and the Coriolis acceleration fields which are present in rotating passages. These features will be discussed in the next paragraph.

Another major limitation of the inviscid models described in the previous paragraphs is their inability to predict correctly secondary flows, which are to a large extent dominated and initiated by viscous effects. The secondary flows, so well described by Eckardt [12], can have a significant effect on the impeller flow field and consequently on the impeller performance. Secondary flow effects must be included in a really representative prediction method.

4. The Structure of Turbulence and the Turbulent Viscosity in Rotating Passages

A conclusion in the previous paragraph was that viscosity can be a dominant influence in centrifugal impellers and that in many cases inviscid methods must fail to predict reality. The role of viscosity is influential, not only because of the adverse pressure gradients present but also because viscosity variations caused by curvature and Coriolis accelerations can be large. Because the flow in most practical impellers is turbulent, work on laminar viscous flows in centrifugal impellers are not in general reviewed here. One result however is noted. Grundmann's study [76] showed that in laminar flow, centrifugal and Coriolis acceleration terms and the additional curvature terms had a strong influence on the location predicted for the separation point using boundary layer theory. If the centrifugal acceleration exceeded a certain limit, the solution scheme failed. This makes the use of boundary layer theory doubtful for this class of flows.

Most practical impeller flows are turbulent, and the pattern of these flows is, among other factors, the result of significant turbulent viscosity gradients. The aim of this paragraph is to review the main reasons for the viscosity gradients, the nature of viscosity in rotating passages and the effect of viscosity gradients on the flow field. The literature on this subject has increased considerably in recent years. On the basis of reported studies it appears that the viscosity variation in the flow field inside a centrifugal impeller is the result of the interaction between a body force field and the velocity field. The viscosity variation depends on the relation between these two fields and is greatest at locations where velocity gradients are large. For the purpose of this review the literature will be classified into three groups: 1) effects of curvature, 2) effects of rotation, and 3) combined effects and application.

4.1 Curvature Effect: In 1969 Bradshaw described an analogy between streamline curvature-and buoyancy effects on turbulent viscosity [77]. He showed that there was a similarity between thermally stratified turbulent viscous flows, (which were by then better explored, especially in meteorological problems) and turbulent viscous flow along curved streamlines. He also discussed, briefly, effects of rotation. His major conclusion was that the effects of curvature, either concave or convex, on the viscosity were large if the thickness of the shear layer exceeded 1/300th of the radius of curvature of the streamline. These effects increased with increasing Mach number. Later, in 1973 and 1975, So and Mellor studied experimentally the effects of curvature on turbulent viscosity [78, 79]. They found that, along a convex wall, the Reynolds stress was decreased so strongly that it vanished in the middle of the boundary layer. In other words, they found that the boundary layer was "laminarized" and that turbulence was suppressed. In contrast, they found that along a concave wall the turbulent intensities inside the boundary layer were substantially increased. These phenomena have a striking effect on centrifugal impeller flows. In 1975, So [80], derived a formula for the variation of the turbulent velocity scale with the Richardson number for curved shear flows. He used the Reynolds stress equations and assumed that production of turbulent energy balanced the viscous dissipation. This provided a first quantitative account of the phenomenon. In the same year Irwin and Smith [81] also published a quantitative prediction of curvature effects. They verified that even small amounts of streamline curvature have a surprisingly strong effect on the eddy viscosity. Using data from curved boundary layers, they were able to predict the curvature effect properly if the curvature terms were included in the model. It is important to note the considerable effect of even small curvatures. It can explain some observed impeller flow field characteristics, such as suction side separation and pressure side attachment, which are contrary to inviscid theory predictions.

Recently Shivaprasad and Ramaprian published two experimental studies [82, 83] which again verified that turbulent viscosity was considerably affected by an even small curvature of the streamlines. Convex curvature decreased both the length and velocity scales of turbulent motions, whereas concave curvature caused the opposite. The effect of small curvature was found to be much larger than one would expect from a linear interpolation between the effects of zero and strong curvatures. That is to say, the effect is nonlinear, and is strong even at small curvatures. They also observed that curvature had a relatively larger influence on the turbulent shear stress than on the turbulent kinetic energy. Another important conclusion was that convex curvature had a stronger effect on the behaviour of the boundary layer than concave curvature of the same magnitude, (this conclusion is again important in the understanding of impeller flows, especially the onset of the wake on the convex suction side of the blades). In addition Shivaprasad and Ramaprian provided a way to calculate these effects using a correction factor to existing phonomenological methods.

A simple description can be given for the mechanism of the curvature effect. A flow is considered to be stable* if a fluid particle, on being perturbed perpendicularly to the average flow, encounters a net restoring force and decelerates. In flows over curved surfaces, the centrifugal force is largely balanced by a normal pressure gradient. If a fluid particle is displaced away from the wall into a region of higher mean velocity, its movement in this direction is on the average reduced if it flows along a convex surface. This is because in its perturbed position the centrifugal force acting on the fluid particle will be smaller than the mean normal pressure force existing in the flow field. The result is a net restoring force. The opposite is true for flow along concave walls. Similar considerations also hold for particle displacement towards the wall. The effects of rotation on turbulence are similar, with the difference that the centrifugal acceleration due to curvature is replaced by the component of the Coriolis acceleration normal to the blade surface.

4.2 Effects of Rotation: As in flow along curved streamlines, the influence of rotation is a result of the relation between pressure forces and inertia forces in the flow. Here the inertia forces

*Stable here refers to the stability of turbulent fluctuations and not the stability of the averaged flow to resist possible separation.

are caused by the Coriolis acceleration. In turbomachines the Coriolis acceleration has only two components, radial and circumferential. The circumferential component, which causes the main effect of rotation on turbulence since it is almost perpendicular to the blade surface, is very small in axial flow machines because the radial component of the relative velocity is small. In centrifugal machines, on the other hand, the circumferential Coriolis component has a considerable magnitude, and therefore effects of rotation on turbulence play a major role. Johnston [84] pointed out that there are two basic effects associated with boundary layers on rotating surfaces: 1) If a component of the Coriolis acceleration exists parallel to the solid wall, secondary flows will tend to develop in the mean flow field of the boundary layer. This effect is present both in axial flow and in radial flow machines 2) If a component of the Coriolis acceleration exists perpendicular to a solid surface, damping or amplifying effects are observed in the structure of turbulence. Because of geometrical reasons this effect is not very strong in axial machines. In radial machines, however, it is of dominant importance.

In 1972 Johnston, Halleen, and Lezius [85] published experimental results from a slowly rotating water channel. They concluded that three different effects can be distinguished in rotating boundary layers; 1) The reduction of turbulent wall layer streak bursting rate in locally stabilized layers. In a centrifugal impeller these layers are on the suction side of the rotating passage. The opposite is true for locally destabilized layers, which in a centrifugal impeller are located on the pressure side;

2) Production of turbulence is totally suppressed in locally stabilized layers; 3) Roll cells are developed on the destabilized side of the channel. These three mechanisms were correlated to the gradient Richardson number and to the rotation number as defined in [85]. Later, in 1973, Johnston extended his observations to derive a predictive method for this type of boundary layer flow [86]. Using the Eddy Reynolds number, Johnston tried to predict the rotating boundary layers using a mixing length correction that accounts for rotation. Lack of sufficient data prevented verification of his results. On the other hand, application of the Eddy Reynolds number criterion to the prediction of transition, also observed experimentally, was quite successful. Johnston's results were derived for rotating passages of constant cross section.

To approximate more closely the conditions in centrifugal impellers, where the passage cross section increases in the flow direction, Rothe and Johnston carried out experiments in rotating diffusers [87]. Their experiments demonstrated that rotation strongly enhanced the tendency of a diffuser to stall. This stall, however, appeared only on the suction side of the rotating diffuser, and was highly steady and two dimensional. The appearance of the suction side stall was correlated by Rothe and Johnston to the rotation number, and a stall regime map was drawn to enable stall prediction in rotating diffusers. It is noted that the appearance of stall on the suction side only is fully consistent with previous observations of the effect of rotation on the structure of turbulence and is explained by them. In 1976 Rothe and Johnston published more data to provide a quantitative account of the onset of flow separation in rotating diffusers [88]. The conclusions were similar to the conclusions in the previous work [87]. It was, however, emphasized in addition that the suppression of turbulent mixing and shear stress by the Coriolis acceleration was powerful even when fully turbulent upstream wall layers were present, and even at small rotation numbers relative to those typical of centrifugal impellers. This conclusion is analogous to the observation that even very small streamline curvatures have a relatively strong effect on turbulent viscosity.

Very recently Koyama, et al, [89] reported a study confirming the observations made thus far. They contributed additional information on the quantitative effect of rotation on the skin friction coefficient, and on the Monin Oboukhov coefficient, to enable more reliable predictions.

4.3 Combined Effects of Surface Curvature and Rotation and Their Application:

So far the effects of streamline curvature without rotation and of rotation without streamline curvature were discussed. In reality, however, they occur together and must be so treated. The effects of the impeller rotation and the curvature of its blades on the stability and structure of the internal boundary layers must be properly described if the flow pattern in a centrifugal impeller is to be predicted correctly. Especially affected by curvature and rotation is the onset of boundary layer separation and stall in the impeller. The regions of stalled or low velocity flow (wakes) reduce efficiency, reduce operating range, and distort the flow delivered at the impeller outlet.

Johnston and Eide [90] devised a method, using previous experimental information, to include the effects of curvature and rotation in an existing differential method for turbulent boundary layer calculation; namely a slightly modified version [91] of STAN 5 [92]. STAN 5 was based on Patankar and Spalding's computing scheme [93]. Johnston and Eide modified the turbulent viscosity model in STAN 5 to include the effects of curvature and rotation. They assumed that the mixing length for the boundary layer on a flat wall with zero rotation must be multiplied by a correction factor F given by

$$F = 1 - \beta_c Ri_c - \beta_\Omega Ri_\Omega \qquad (1)$$

where β_c and β_Ω are empirical constants for curvature and rotational effects, Ri are the corresponding Richardson numbers. This simple correction factor, which assumes that the effects of curvature and rotation are linearly combined, was tested on a number of boundary layers. Unfortunately, all the available cases were either curved or rotating and the combination of the two effects could not be checked. The agreement between the description provided by Eq. (1) and the experiments was good in cases where the thin-shear-layer approximation was applicable.

In 1977 Launder, Priddin and Sharma [94] published a method which was applied to spinning cones. It is nevertheless of interest here. Again Patankar and Spalding's approach [93] was used for the boundary layer calculation in an adapted version of the

GENMIX program. They corrected the turbulent viscosity for curvature and rotation effects using a model of trubulence kinetic energy and its local rate of dissipation. The direct effect of curvature in the model was limited to a single empirical coefficient whose magnitude was directly proportional to a Richardson number based on a time scale of the energy-containing eddies. They carried out computations for shear flows on curved or spinning surfaces. They found that the turbulent kinetic energy-dissipation rate model did describe the experimental results, provided that the transport equation for dissipation was modified to include an extra term to account for curvature and swirl effects. The success on curved surfaces was better than on spinning surfaces, where the predictions obtained were only marginally superior to those obtained with a mixing length model [95]. They concluded that when a mixing length profile can be predicted with reasonable certainty, its use would be advantageous for spinning flows because of the smaller computing time required to obtain a solution.

The present knowledge on turbulence in rotating passages allows, at least, qualitative understanding of many of the internal passage flow mechanisms. It can explain the occurence of the jet-wake flow. It can explain why inviscid pressure side separations like the one shown in figure 7 were never detected experimentally. It can also provide qualitative justification for observed three dimensional flow effects. However, the present knowledge is not sufficient to allow accurate quantitative predictions to be made of the turbulent viscosity and incorporation of those predictions in the 3-D viscous computing techniques now under development.

Computing techniques, once reliable, would allow for example a detailed study to be carried out of the influence of blade sweepback and curvature on stage efficiency and range of operation. Presently, the effects of blade sweepback are qualitative inferred from an examination of global overall information such as stage performance maps, which correlate blade geometry with efficiency and range, but always leave the internal flow details in the dark. A detailed analytical study could, for example, examine the apparently large and harmful effects of even a small amount of convex suction-side blade curvature on the onset of jet-wake flow, and might correlate this information with observations of the triggering of surge. Also, viscous computations with correct viscosity models could be used to create diagrams like the one in Fig. 1, and give a better understanding of the processes causing jet-wake flow.

5. Inner-Outer Solution Combinations (Patching Techniques)

A standard approach to solving for a complete flow field, is first to divide the flow field into regions. In each region, a different simplifying assumption is made in order to allow an easy solution to be obtained. The regions are connected by common boundary conditions on common boundaries. The classical example of this approach is boundary layer theory. In centrifugal impellers a similar approach was used by a number of investigators in a very simplified way to attack the problem of the jet-wake flow*. The problem is of course complicated by the fact that the region boundaries are not known a-priori. Consequently, the techniques are either iterative or depend strongly on empirical results. At present, the information quoted above on turbulent viscosity variation in curved rotating passages is not directly incorporated in viscous flow field calculations, and some methods avoid viscosity altogether.

In 1973 John Moore [96] published a method of calculating the jet-wake flow in a rotating radial flow passage. Moore divided the flow field into four parts; 1) Potential flow; 2) Top and bottom wall boundary layers; 3) Corner flows, and 4) Side wall boundary layers. Moore calculated the potential core flow accounting for the displacement and acceleration of the potential flow by the wake on the suction side wall. The development of the cross flows on the top and bottom walls was computed on the basis of Moses' strip integral method [97] modified to include the three dimensional and rotational terms for steady incompressible flow in rotating passages. Thus on top and bottom the wall boundary layers as well as radial and tangential flows were predicted. The corner flows were modeled by assuming that continuity holds over the corner cross section, and that the fluid turning through the corner maintains its momentum in the radial direction. The side wall boundary layers were computed with an analysis which included a Coriolis' acceleration term associated with velocities

*There are also a number of works published on boundary layers and secondary flows in impellers. Though relevant, these studies are not included in order to keep this review within reasonable limits.

normal to the wall and a Coriolis term affecting the pressure gradient normal to the wall. The model gave a momentum integral equation similar to the one derived by Moon [98]. However, it was applied with irrotational potential flow at the edge of the boundary layer, and retained the terms which arose due to the two Coriolis effects. Moore calculated the wake assuming that it started at a point where the shape factor was 3.0. Subsequent flow was considered to consist of similarity profiles, including a cross flow entrainment into the wake from the corners. These four partial flows were combined to yield a good agreement between calculations and experiment (Fig. 17).

In 1975 Sturge and Cumpsty developed a two dimensional method to predict the incompressible jet-wake flow [99]. For simplicity, they avoided the direct modelling of viscous effects, which led to major drawbacks in their method. The method was later extended to compressible flows [100]. The two dimensional blade to blade passage was divided into two regions; 1) a jet region where the flow was assumed to be inviscid and irrotational; 2) a wake region where there was no flow. The boundary between the two regions was treated as a free shear layer without mixing between the two regions, similar to an air-water interface. The position of the shear layer was characterised by the condition that there was no change in static pressure across it, and that in the wake the pressure was governed only by hydrostatic effects due to the centrifugal acceleration field. The flow in the jet was analysed using irrotationality and continuity and defining a stream function, and was solved numerically with the shear layer between the jet and the wake as one of the boundaries. This required an iterative process which was shortened by a variable transformation which left the stream function as one of the independent variables. The resulting second order partial differential equation was solved using a finite difference technique. Besides the fact that the method was restricted to irrotational flow, a much more disturbing feature was that the separation point on the suction side could not be predicted, but had to be known a-priori or assumed. (This was the penalty for not modelling the viscous effects). Though a separation criterion based on a velocity ratio between separation and the highest velocity attained on the suction side was suggested by Sturge and Cumpsty the method was still not predictive unless the calculation of a separation point on the suction side was incorporated. Another drawback of the technique was that no flow was assumed in the wake. This is not in accordance with Eckardt's observations [12]. Eckardt measured significant velocities inside the wake. The magnitude of the velocities was about 30% of the jet velocities. No comparison to experiments were given in Sturge and Cumpsty's work.

Howard and Osborne [101] developed a further simplified method to predict the jet-wake flow. The method was based on Katsanis' streamline curvature inviscid method on the H-S stream surface [21] combined with an extremely simplified calculation on the B-B surface [102]. In contrast to Sturge and Cumpsty's method the analysis did not determine an exit flow angle and required a separate correlation for this purpose. The wake was treated, as in the previous method, as an additional blockage in the passage. The inviscid calculation was carried out in the jet only with the mid passage H-S surface, moved towards the pressure side by one-half the wake width. Subsequently, the B-B analysis was carried out in the region of the jet only. The continuity equation at each quasi-orthogonal was adjusted to account for growth of the wake width. The wake width and the jet flow angle, required in the calculation, were not predicted by the technique but had to be established either by empirical correlations or from measured data. The ability of the method to be really predictive depended on the accuracy and the generality of this empirical information. Although some three dimensional aspects of the jet-wake flow were included, they were possibly oversimplified by the assumption that the wake was evenly distributed along the suction side of the passage from shroud to hub. This is not consistent with Eckardt's observations [12].

The problem of calculating the jet-wake flow properly is formidable. Only three attempts have been reported so far, and none was based on a partial differential approach. Two (Sturge and Cumpsty, and Howard and Osborne) avoided viscosity altogether by replacing the wake with a region without flow. Moore, in a more detailed model, applied an integral boundary layer technique. The full task is still not accomplished and further attempts are anticipated.

6. Fully Viscous Solutions

A large volume of work on the solution of viscous flows was published in the past few years and a complete review will not be attempted. Of direct interest here are solutions for flows which are both rotating and turbulent. A number of relevant works will be chronologically reviewed.

In 1975 Walitt, Harp and Liu published a technique [103], to predict the viscous flow field on the B-B stream surface. It was the first phase of a technique developed to solve the steady three dimensional compressible Navier-Stokes equations in rotating curvilinear coordinates. Any turbulence model could be incorporated in the method which was applied (in Phase I) to laminar flow and to turbulent flow described by a simple mixing length correlation not incorporating rotation and curvature effects. In 1978 the second phase of the work, solving the flow on cross sectional surfaces perpendicular to the direction of the primary flow, was published [104]. The full technique solved the equation of motion by iterations. The solution started from a zeroth iteration which was a 3-D inviscid solution; Katsanis' method [21] was used. The solution marched alternately from hub to shroud (solving on B-B surfaces) and from inlet to outlet (solving on cross-sectional surfaces). The cross sectional iteration employed the B-B flow field as the previous iterate. The method was an explicit two dimensional time marching scheme extended to three dimensional flows using the equivalence principle of Hayes. In its original derivation, the equivalence principle states that for slender bodies at hypersonic speeds the three dimensional steady equations of motion for inviscid flow reduce identically to unsteady equations in two space dimensions. An extension of the principle was used in forming an analytical model which calculated the viscous cross flow with a known flow from a previous iterate in the marching direction. The coordinate in the marching direction was made proportional to a time like variable by forming the product with a characteristic velocity.

The method was used to predict the flows in a radial exit, and in a backswept compressor impeller. In the radial exit impeller a separation zone was predicted on the suction side of the passage close to the outlet. The flow in the backswept impeller was attached. No conclusions can be drawn with regard to the effectiveness of sweepback in reducing or eliminating the jet-wake flow because the radial exit impeller was calculated with a laminar viscosity model, and the backswept impeller was calculated with a turbulent mixing length model for steady flows along straight surfaces. No comparison to experiments was given, but in [104] a qualitatively correct jet-wake pattern was predicted.

In 1976 Briley, Kreskovsky and McDonald published a method developed for stationary axial trubomachinery passages [105]. It is included here because it can in principle be extended to rotating centrifugal passages. The method was based on an approximate formulation which was solved in a forward marching technique in a primary spacial direction. The complete flow was obtained in a sequence of two dimensional calculations, resulting in a substantial saving of computer time and storage over that which would be required for the solution of the full, elliptic Navier-Stokes equations. Essential to the method was the derivation of a particular coordinate system. One coordinate direction was identified with the primary flow direction along which the solution was to march, while the two remaining coordinate directions could be associated with secondary flows. The full elliptic flow model was simplified by a number of assumptions. For high Reynolds numbers, viscous effects were negligible except in the boundary layers on all four walls. The method was an extension of 3-D boundary layer methods, but the approximate equations were also used in the inviscid core flow. For entirely supersonic flows the equations, together with initial and boundary conditions, could be solved by forward marching in the primary direction without any assumptions about the pressure field. For subsonic, however, where the inviscid core was elliptic, downstream boundary conditions were required to enable a solution. This problem was overcome by imposing on the solution, an entirely inviscid pressure field which was calculated a priori with proper elliptic boundary conditions. The pressure field then had to be corrected as part of the solution procedure. The viscous pressure drop in the primary direction was treated separately as a function of the distance in this direction only, and was computed with the forward marching integration using integral mass conservations in that direction. No assumptions were made concerning the pressure gradients in the secondary flow directions. The turbulence model incorporated in the method was based on the solution of conservation equations for turbulent kinetic energy and dissipation, and as the technique was developed for stationary passages, rotation effects were not included. Curvature effects were also neglected.

Another viscous computation technique for stationary turbomachinery ducts was reported by Dodge [106]. Again the model was not based on the full Navier-Stokes equations in order to save computer time and storage space. The method was based on separating the momentum and continuity equations into coupled elliptic and parabolic parts which were then solved iteratively. Dodge separated the velocity into a viscous and a potential component. Substitution of the decomposed velocity into the equation of motion yielded a simpler equation of motion from which the pressure gradient was eliminated using the velocity potential. The velocity potential in turn was expressed using the continuity equation. The simplified equation of motion was parabolic with a primary direction in the flow direction, and was solved using a marching technique similar to those used in boundary layer theories. The equation for the potential was elliptic and was solved with a relaxation technique. Since both equations contained coupling terms, iteration was required between the two. Dodge developed and used his technique for stationary passages; consequently his turbulence model did not include effects of rotation. He also did not correct the turbulence model for curvature effects despite the presence of curvature in the passages which were analysed. The turbulent viscosity was calculated using a mixing length concept near the walls (including Van Driest's wall shear correction and Cebeci-Smith pressure correction) and a constant free stream viscosity away from the walls. A nearly constant viscosity was used in wakes. Dodge's technique failed in the presence of reversed flow, the forward marching differences then becoming unstable.

By replacing the variation of the dependent variable in the normal direction by a spectral series [107] this problem was overcome. A number of comparisons with the experiments of Schubauer and Klebanoff revealed good agreement.

In 1977 a paper [108] was published by Majudmar, Pratap and Spalding which described a method based on Patankar and Spalding's boundary layer calculation [109]. The method of Patankar and Spalding for parabolic flows is based on the uncoupling of the pressure; using in the momentum equation for the primary direction (the radial direction) a pressure value which is averaged over the cross section. In the method of Majudmar et al the model reported in [109] was modified through addition of a Coriolis term in the circumferential momentum equation and both a centrifugal and a Coriolis term in the radial momentum equation. The turbulence model used was not corrected for rotational and curvature effects; it was based on Launder's and Spalding's kinetic energy-dissipation concept [110]. The model was solved in a single forward marching sweep from inlet to outlet in the following steps:

1) The pressure distribution in a cross section perpendicular to the primary direction, and the average value of the pressure in that cross section were assigned preliminary guessed values.
2) The three momentum equations at a downstream position were solved, using flow properties at an upstream position, to yield a first approximation to the velocity distribution.

3) The mean cross sectional pressure was corrected to give new radial velocity components which satisfied continuity. This was done using a linearized radial momentum equation.

4) A corrected pressure field on the cross section surface was computed using an elliptic equation derived from continuity, and two linearized momentum equations in that surface. The cross sectional velocities were corrected according to this pressure field.

5) A new turbulent viscosity distribution was evaluated from a solution of the kinetic energy and dissipation equations.

Figure 18 illustrates the good agreement obtained between this technique and Moore's experiments [10]. Majumdar, Pratap and Spalding concluded, however, that their parabolic technique was only accurate for low values of $\Omega B/\bar{w}$ (where Ω is the angular velocity, B is the circumferential width of passage, and $\bar{w}$ is the bulk mean velocity). For high values of $\Omega B/\bar{w}$, partially parabolic procedures [111, 112] in which the pressure uncoupling was abandoned yielded more accurate results. Both the partially parabolic technique [111, 112] and the parabolic technique [108] were based on identical mathematical and turbulence models. The difference was in the method of solution. While in the parabolic technique a single marching sweep was carried out from inlet to outlet, the partially parabolic technique was necessarily iterative. A number of marching sweeps, each similar to the single sweep of the parabolic method, were required. In each sweep, or iteration, the pressure field was corrected until convergence was achieved. Thus through the pressure field, downstream effects were transmitted into the flow field and the solution was given a radially, or streamwise, elliptic character. Results for secondary flows were given in [112] and compared to measurements of Wagner and Velkoff [113]. Agreement was good in some cases, but significant disagreements were found in others.

Very recently Bosman, Chan and Hatton published a finite difference technique which was simple and probably not expensive to run [114]. The model was, however, subjected to a number of restrictions which limited its application to two dimensional flows in radial surfaces which were absolutely plane and perpendicular to the axis of rotation. The flow had to be incompressible and the viscosity constant in the entire flow field. Further, the technique utilized a nondimensional slip parameter to represent the blade surface shear stresses. The slip parameter was assumed to be constant along the blades from inlet to outlet, and reliable information was not given on its evaluation. Using the incompressible equations of continuity and momentum and a stream function, the flow was formulated with two coupled partial differential elliptic equations. This was the distinctive feature of this technique in contrast to the parabolic, or partially parabolic nature of the forward marching methods. The unknown functions to be evaluated were the stream function and the relative vorticity. The equations were solved with a relaxation technique with finite difference approximations. Central differences were used for diffusion terms and up-wind differences for convection terms. The two equations were solved iteratively until convergence was achieved. The results were compared to measurements carried out in a water rig [115, 116] and to results of an inviscid calculation based on Wu. At the design point of the impeller, agreement between the experiments, the inviscid calculation, and the viscous calculation was not bad. This is not surprising in view of the fact that the impeller was two dimensional and strongly backswept. In such a configuration, viscous effects are not very strong (see Fig. 1) and inviscid calculations should give fairly accurate results. The situation at off design points, with positive or negative inlet incidence angles, was not much different downstream and close to the impeller exit. At upstream positions, closer to the inlet, the agreement between experiments and viscous results was better than agreement between experiments and inviscid results. This again is not surprising in view of the fact that inviscid theories should not be used at all at off design conditions since they can not represent properly the viscous effects associated with the inlet shock. The technique of Bosman et al [114], when further developed to remove some of its restrictions could become a useful design aid.

Also in 1978 Goulas and Baker [117] published a method for calculating the viscous flow on either the H-S or on the B-B stream surfaces. The method was not three dimensional thus far, but the author's stated intention was to couple the two solutions in an iterative sequence to obtain the three dimensional flow field. The published method was based on the usual conservation equations written for a stream surface. Introducing a stream function, the two equations of motion for the stream surface were combined into a Poisson like equation, $\nabla^2\psi = q(r,\phi,Z)$, in which the function q was not known. The function q contained all the differences between incompressible irrotational flow and real viscous compressible flow under consideration. The solution by finite differences on a stream surface was carried out in the following steps:

1) A stream function distribution was assumed from which the initial two components of the velocity on the surface were calculated. The third velocity component was calculated using the geometry of the stream surface.

2) With this information the turbulent viscosity was computed using Launder's and Spalding's kinetic energy-dissipation model [110]. A set of linear equations was formed and solved simultaneously for each quasi-orthogonal line. Starting from the inlet where the kinetic energy and dissipation were known as initial values, the solution proceeded downstream to the exit.

3) The streamwise momentum equation was integrated along streamlines, using the calculated turbulent viscosity, to yield an entropy field.

4) With velocities, entropy and density known, the function q was calculated, and Poisson equation for the stream function was then solved.

The process was iteratively repeated until convergence of the stream function was achieved.

Results were compared with the inviscid solution of the flow field in a turbocharger compressor and viscous effects were qualitatively clearly visible [117]. No comparison with experiments was given.

7. Comparison of Computation Times

The solution techniques mentioned in this review require numerical methods. A comparison of their drawbacks and merits is not complete without consideration of their demands for computer time and computer storage. For most numerical techniques the storage requirements are not published, though the particular computer model used can sometimes indicate what might be required. The following table gives an approximate indication of computation times for some of the techniques mentioned in this review. An exact accounting, however, is not possible as some authors mention virtual execution time, and others indicate a total time which includes compilation of the program. Some authors even neglect to define the time that they quote. Furthermore, the time stated depends strongly on the problem solved and the accuracy obtained in the results. The accuracy, in turn, is affected by mesh size and convergence criterions adopted. The values given in the following table should therefore, be treated as an order of magnitude indication only. Table I classifies the techniques according to the following abbreviations:

SC - Streamline Curvature

FD - Finite Differences

FE - Finite Elements

Table 1. Comparison of Approximate Computation Times

Author	Ref. No.	Computer Used	Time (Mins)	Flow	Technique
Katsanis	21	IBM 360	10	H-S inviscid	SC
Marsh	27	KDF 9	10-15	H-S inviscid	FD
Katsanis/McNally	29	IBM 360	30	H-S inviscid	FD
Katsanis/McNally	30	IBM 360	3-15	H-S inviscid	FD
Adler/Krimerman	31	IBM 370	1.5	H-S inviscid	FE
Frost	34	SDS 90/300	10-15	H-S inviscid	SC
Davis/Millar	35	?	1	H-S inviscid	SC
Davis/Millar	35	?	.5	H-S inviscid	FD
Wilkinson	43	IBM 360	.5	B-B inviscid	SC
Katsanis/McNally	49	IBM 360	30	B-B inviscid	FD
Adler/Krimerman	54	IBM 370	2	B-B inviscid	FE
Katsanis	56	IBM 360	10	B-B transonic/inviscid	SC/FD
Senoo/Nakase	68	Facom 230-60	10(sec)	B-B inviscid (cpu)	FD
Katsanis	66	IBM 360	2	3-D inviscid	SC/FD
Senoo/Nakase	67	Facom 230-60	16	3-D inviscid	SC/FD
Bosman/Shaarawi	69	CDC 7600	3.5	3-D inviscid	FD
Novak/Hearsey	70	CDC 6600	1.5	3-D inviscid	SC
Hirsch/Warzee	73	CDC 6500	1	3-D inviscid (cpu)	FE
Krimerman/Adler	74	IBM 370	5	3-D inviscid	FE
Wallit/Harp/Liu	103	CDC 6400	3000	3-D viscous	
Dodge	106	CDC 6400	60	3-D viscous	
Dodge/Lieber	109	CDC 6400	7	3-D viscous	
Dodge/Lieber	109	CDC 7600	.5	3-D viscous	
Madjumar/Pratap	108	CDC 6600	2.5	3-D viscous	
Madjumar/Spalding	111	CDC 6600	2.5	3-D viscous	

<u>References</u>

1. J. H. Stang, "Designing Adiabatic Engine Components", <u>SAE</u> paper 780069 (1978).

2. A. Carrard, "On Calculations for Centrifugal Wheels", <u>La Technique Moderne</u> T. XV-No. 3, (1 February 1923).

3. H. Oertli, "Untersuchungen der Wasserströmungdurch ein rotierendes Zellen-Kreiselrad", <u>Dissertation E.T.H.</u> No. 317 (1923).

4. P. Prášil, "Verschiedene Strömungserscheinungen", Second International Congress for Applied Mechanics, Zürich (1926).

5. K. Fischer and D. Thoma, "Investigation of the Flow Conditions in a Centrifugal Pump", <u>ASME Trans</u>, HYD-54-8 (1932).

6. W. J. Keanton, "Influence of the Number of Impeller Blades on the Pressure Generated in a Centrifugal Compressor and on its General Performance", <u>Proc. Fust. Mech. Engs.</u>, Vol. 124, pp. 481-568 (1933).

7. A. J. Acosta, R. D. Bowerman, "An Experimental Study of Centrifugal Pump Impellers", <u>Transaction of the ASME</u>, Vol. 79, pp. 1821-1839 (1957).

8. K. Fujie, "Three Dimensional Investigation of Flow in Centrifugal Impeller with Straight-Radial Blades", <u>Bulletin of JSME</u>, Vol. 1, pp. 42-49 (1958).

9. H. S. Fowler, "The Distribution and Stability of Flow in a Rotating Passage", <u>Journal of Engineering for Power</u>, Vol. 90, pp. 229-236 (1968).

10. J. Moore, "A Wake and an Eddy in a Rotating Radial Flow Passage", <u>Journal of Engineering for Power</u>, Vol. 95, pp. 205-219 (1973).

11. J.H.G. Howard, C. W. Kittmer, "Measured Passage Velocities in a Radial Impeller with Shrouded and Unshrouded Configuration", <u>Journal of Engineering for Power</u>, Vol. 97, pp. 207-213 (1975).

12. D. Eckardt, "Detailed Flow Investigation within a High Speed Centrifugal Compressor Impeller", Journal of Fluid Engineering, Vol. 98, pp. 390 - (1970).

13. R. Schodl, "A Laser Dual-Beam Method for Flow Measurements in Turbo-machines", ASME Paper, 74-GT-157 (1974).

14. J. Moore, "Eckardt's Impeller - a Ghost From Ages Past", University of Cambridge, Report CUED/A-Turbo/TR 83 (1976).

15. S. Mizuki, T. Hattore, I. Arign and I. Watanabe, "Reversed Flow Phenomena within a Centrifugal Compressor Channels at Lower Flow Rates", ASME Paper 76-GT-86 (1976).

16. D. Adler, Y. Levy, "Laser Doppler Flow Investigation Inside a Backswept Closed Centrifugal Impeller", Journal of Mechanical Engineering Science, Vol. 21 pp. (1979).

17. D. Eckarts, "Instantaneous Measurements in the Jet-Wake Discharge Flow of a Centrifugal Compressor Impeller", Journal of Engineering for Power, Vol. 97, pp. 337-346 (1975).

18. Y. Seeno. H. Hayami, Y. Kinoshita, H. Yamasaki, Experimental Study on Flow in a Supersonic Centrifugal Impeller", Journal of Engineering for Power, Vol. 101, pp. 32-41 (1979).

19. D. Adler and M. Ilberg, "A Simplified Method for Calculation of the Flow Field at the Entrance of a Radial or a Mixed Flow Impeller", ASME Paper 70-FE-36 (1970).

20. C. H. Wu, "A General Theory of Three-Dimensional Flow in Subsonic and Supersonic Turbomachines of Axial-Radial-, and Mixed Flow Types", NACA TN 2604 (1952).

21. T. Katsanis, "Use of Arbitrary Quasi-Orthogonals for Calculating Flow Distribution in the Meriodional Plane of a Turbomachine", NASA TN D-2546 (1964).

22. R. A. Novak, "Stream Line Curvature Computing Procedures for Fluid Flow Problems", Trans ASME, Vol. pp. (1967).

23. R. A. Novak, "The Mean Stream-Sheet Momentum Continuity Solution Techniques for Turbomachinery", Iowa State University, July 15-25 (1975).

24. W. R. Davis, "A Comparison of Different Forms of the Quasi-Three-Dimensional Radial Equilibrium Equation of Turbomachines", Carleton University Report No. ME/A 68-1 (1968).

25. L. H. Smith, "The Radial Equilibrium Equation of Turbomachinery", Journal of Engineering for Power, Vol. 88, pp. 1-12 (1966).

26. C. H. Wu and A. Curtis et al, "Analysis of Flow in a Subsonic Mixed Flow Impeller", NACA TN-2749 (1952).

27. H. Marsh, "A Digital Computer Program for the Through-Flow Fluid Mechanics in an Arbitrary Turbomachine Using a Matrix Method", R&M No. 3509 (1968).

28. W. R. Davis, "Axial Flow Compressor Analysis Using a Matrix Method", Carleton University, Report No. MW/A 73-1 (1973).

29. T. Katsanis, W. D. McNally, "Fortran Program for Calculating Velocity and Streamlines on the Hub-Shroud Mid Channel Flow Surface of an Axial-or Mixed Flow Turbo-machine", NASA TN D-7343 (1973).

30. T. Katsanis and W. D. McNally, "Revised Fortran Program Calculating Velocities and Streamlines on the Hub-Shroud Mid Channel Stream Surface of an Axial-, Radial-, or Mixed Flow Turbomachine or Annular Duct. NASA TN D-8430/1 (1977).

31. D. Adler and Y. Krimerman, "The Numerical Calculation of the Meridional Flow Field in Turbomachines Using the Finite Elements Method", Israel Journal of Technology, Vol. 12, pp. 268-274 (1974).

32. B. C. Oates, C. J. Knight and G. F. Carey, "A Variational Formulation of The Compressible Through Flow Problem", ASME paper 75-GT-32 (1975).

33. Ch. Hirsch and G. Warzee, "A Finite Element Method for Through Flow Calculation in Turbomachines", Journal of Fluids Engineering, Vol. 98, p. 403 (1976).

34. D. H. Frost, "A Streamline Curvature Through Flow Computer Program for Analyzing the Flow Through Axial Flow Turbomachines", Aero. Res. Council R&E No. 3687 (1972).

35. W. R. Davis and D. A. J. Millar, "A Comparison of the Matrix and Streamline Curvature Methods of Axial Flow Turbomachinery Analysis from a User's Point of View", ASME paper 74-WA/GT-4 (1974).

36. G. E. Martensen, Juris Verlag, Prom. No. 3042 (1964).

37. D. H. Wilkinson, Aeronautical Research Council, R&M 3545 (1968).

38. B. S. Rauchman, "Blade Cascade on Axisymmetric Flow Sheet of Variable Thickness of Subsonic Flow", Energomachinostroenie N11 (1972 (in Russian).

39. Von Der Braembussche, "Transonic Flow in Turbomachinery", Von Karman Institute Lecture Series, Lecture 59 (1973).

40. T. Ogawa and S. Murata, "On the Flow in the Centrifugal Impeller with Arbitrary Airfoil Blades", Bulletine of the JSME, Vol. 17, p. 713 (1974).

41. T. Ogawa and S. Murata, "On the Flow in the Centrifugal Impeller with Arbitrary Airfoil Blades", Bulletine of the JSME, Vol. 17, p. 723 (1974).

42. T. Katsanis, "Use of Arbitrary Quasi-Orthogonals for Calculating Flow Distribution on a Blade-to-Blade Surface in a Turbomachine", NASA TN D-2809 (1965).

43. D. H. Wilkinson, "Calculation of Blade-to-Blade Flow in a Turbomachine by Streamline Curvature", Aeronautical Reasearch Council, R&M No. 3704 (1972).

44. D. H. Wilkinson, "Streamline Curvature Methods for Calculating the Flow in Turbomachines", English Electric Co., Report No. W/M (3F) (1969).

45. J. D. Stanitz, "Two Dimensional Compressible Flow in Conical Mixed Flow Compressors", NACN TN 1744 (1948).

46. J. D. Stanitz, "Two Dimensional Compressible Flow in Turbomachines with Conical Flow Surfaces", NACA TO 935 (1949).

47. J. D. Stanitz and G. O. Ellis, "Two Dimensional Compressible Flow in Centrifugal Compressors with Straight Blades", NACA TR 954 (1950).

48. T. Katsanis, "Computer Program for Calculating Velocities and Streamlines on a Blade-to-Blade Stream Surface of a Turbomachine", NASA TN D-4525 (1968).

49. T. Katsanis and W. D. McNally, "Fortran Program for Calculating Velocities in a Magnified Region on a Blade-to-Blade Stream Surface of a Turbomachine", NASA TN D-5044 (1969).

50. T. Katsanis and W. D. McNally, "Fortran Program for Calculating Velocities and Streamlines on a Blade-to-Blade Stream Surface of a Tandem Turbomachine", NASA TN D-5044 (1969).

51. D. J. L. Smith and D. H. Frost, "Calculation of the Flow Past Turbomachine Blades", Proc. Institution of Mechanical Engineers, Vol. 184, part 3G (II) (1969/70).

52. R. B. Deshpande, "A New Algorithm for the Solution of Turbomachinery Flow Problems", Journal of Fluids Engineering, Vol. 97, pp. 372-374 (1975).

53. W. R. Davis and D. A. J. Millar, "A Matrix Method Applied to the Analysis of the Flow Past Turbomachine Blades", Carleton University, Report ME/A72-7 (1972).

54. D. Adler and Y. Krimerman, "Calculation of the Blade-to-Blade Compressible Flow Field in Turbo Impellers Using the Finite-Element Method", Journal of Mechanical Engineering Science, Vol. 19, pp. 108-112 (1977).

55. T. C. Prince, "Prediction of Steady Inviscid Compressible Flow on a Blade-to-Blade Surface by Finite Element Method", AIAA paper 78-244 (1978).

56. T. Katsanis "Fortran Program for Calculating Transonic Velocities on a Blade-to-Blade Stream Surface of a Turbomachine", NASA TN D-5427 (1969).

57. H. Marsh and H. Merryweather, "The Calculation of Subsonic and Supersonic Flow in Nozzles", Institution of Mechanical Engineers, Symp. on Internal Flows, paper 22 (1971).

58. H. Daneshyar and D. R. Glynn, "The Calculation of Flow in Nozzles Using a Time Marching Technique Based on the Method of Characteristics", International Journal of Mechanical Eng. Science, Vol. p. 921 (1973).

59. R. W. McDonald, "The Computation of Transonic Flow Through Two Dimensional Gas Turbine Cascades", ASME Paper 71-GT-89 (1971).

60. S. Gopalakrishran and R. Bozzola, "A Numerical Technique for the Calculation of Transonic Flows in Turbomachinery Cascades", ASME Paper 71-GT-42 (1971).

61. J. D. Denton, "A Time Marching Method for Two and Three Dimensional Blade-to-Blade Flows", Aeronautical Research Council, 35567 (1974).

62. J. W. Kurzrock and A. S. Novick, "Transonic Flow Around Rotor Blade Elements", ASME Paper 75-FE-24 (1975).

63. H. Sobieczky, K. Y. Fung and A. R. Seebass, "A New Method for Designing Shock Free Transonic Configurations", AIAA Paper 78-144 (1978).

64. H. Sobieczky, "Design of Shock Free Transonic Flow in Turbomachinery", USN Naval Postgraduate School Report NPS 67-78-005 (1978).

65. M. J. Schilhansl, "Three-Dimensional Theory of Incompressible and Inviscid Flow through Mixed Flow Turbomachines," Journal of Engineering for Power, Vol. 87, pp 361-373 (1965).

66. T. Katsanis, "Fortran Program for Quasi-Three-Dimensional Calculation of Surface Velocities and Choking Flow for Turbomachine Blade Rows", NASA TN D-6177 (1971).

67. Y. Senoo and Y. Nakase, "An Analysis of Flow Through a Mixed Flow Impeller", Journal of Engineering for Power, Vol. 94, pp 43-50 (1972).

68. Y. Senoo and Y. Nakase, "A Blade Theory of an Impeller with an Arbitrary Surface of Revolution", Journal of Engineering for Power, Vol. 93, pp 454-460 (1971).

69. C. Bosman and M.A.I. El-Shaarawi, "Quasi-Hsec-Dimensional Numerical Solution of Flow in Turbomachines", Journal of Fluids Engineering, Vol. 99, pp 132-140 (1977).

70. R. A. Novak and R. M. Hearsey, "A Nearly Three-Dimensional Interblade Computing System for Turbomachinery", Journal of Fluids Engineering, Vol. 99, pp 154-166 (1977).

71. M. J. Hill, "Numerical Solutions for Mixed Flow Turbomachines", University of Manchester Institute of Science and Technology, Ph. D. Thesis (1974).

72. C. Bosman, "Computation Aspects of Fluid Flow Through a Turbomachine", University of Manchester Institute of Science and Technology, Ph.D. Thesis (1977).

73. Ch. Hirsch and G. Warzee, "An Integrated Quasi-3-D Finite Element Calculation Program for Turbomachinery Flows", Journal of Engineering for Power, Vol. 101 pp. 141-148 (1979).

74. Y. Krimerman and D. Adler, "The Complete Three-Dimensional Calculation of the Compressible Flow Field in Turbo Impellers", Journal of Mechanical Engineering Science, Vol. 20 pp 149-158 (1978).

75. D. Adler and Y. Krimerman, "The Limitation of 3-D Inviscid Models in Describing the Flow in Centrifugal Impellers", (in preparation).

76. R. Grundmann, "Two Dimensional, Laminar, Compressible Boundary Layer Calculation in Turbomachines", DFVLR DLR-FB 76-38 (1976).

77. P. Bradshaw, "The Analogy Between Streamline Curvature and Buoyancy in Turbulent Flow", Journal of Fluid Mechanics, Vol. 36, pp. 177-191 (1969).

78. R. C. So and G. L. Mellor, "Experiment on Convex Curvature Effects in Turbulent Boundary Layers", Journal of Fluid Mechanics, Vol. 60, pp. 43-62 (1973).

79. R. C. So and G. L. Mellor, "Experiment on Turbulent Boundary Layers on a Concave Wall", The Aeronautical Quarterly, Vol. 26, pp. 25-40 (1975),

80. R. C. So, "A Turbulence Velocity Scale for Curved Shear Flows, Journal of Fluid Mechanics, Vol. 70, pp. 37-57 (1975).

81. H. P. A. M. Irwin and P. A. Smith, "Prediction of the Effect of Streamline Curvature on Turbulence", The Physics of Fluids, Vol. 18, pp. 624-630 (1975).

82. B. G. Shivaprasad and B. R. Ramaprian, "Turbulence Measurements in Boundary Layers Along Mildly Curved Surfaces", ASME paper 77-WA/FE-8 (1977).

83. B. R. Ramaprian and B. G. Shivaprasad, "The Structure of Turbulent Boundary Layers Along Mildly Curved Surfaces", Journal of Fluid Mechanics, Vol. 85, pp. 273-303 (1978).

84. J. P. Johnston, "The Effects of Rotation on Boundary Layers in Turbomachine Rotors", Stanford University Report MO-24 (1970).

85. J. P. Johnston, R. M. Halleen and D. K. Lazins, "Effects of Spanwise Rotation on the Structure of Two-Dimensional Fully Developed Turbulent Channel Flow", Journal of Fluid Mechanics, Vol. 56, pp. 533-557 (1972).

86. J. P. Johnston, "The Suppression of Shear Layer Turbulence in Rotating Systems", Journal of Fluids Engineering, Vol. 95, pp. 229-236 (1973).

87. P. H. Rothe and J. P. Johnston, "Stall in Rotating Two-Dimensional Diffusers", Journal of Fluids Engineering, Vol. 97, pp. 252-253 (1975).

88. P. H. Rothe and J. P. Johnston, "Effect of System Rotation on the Performance of Two Dimensional Diffusers", Journal of Fluids Engineering, Vol. 98, pp. 422-430 (1976).

89. H. Koyama, S. Masuda, I. Ariga and I. Watanbe, "Stabilizing and Destabilizing Effects of Cariolis Force on Two-Dimensional Laminar and Turbulent Boundary Layers", Journal of Engineering for Power, Vol. 101 pp. 23-31 (1979).

90. J. P. Johnston and S. A. Eide, "Turbulent Boundary Layers on Centrifugal Compressor Blades: Prediction of the Effects of Surface Curvature and Rotation", Journal of Fluids Engineering, Vol. 98, pp. 374-381 (1976).

91. S. A. Eide and J. P. Johnston, "Prediction of the Effects of Longitudinal Wall Curvature and System Rotation on Turbulent Boundary Layers", Stanford University Report PD-19 (1975).

92. M. E. Crawford and W. M. Kays, "STAN5 - A Program for Numerical Computation of Two-Dimensional Internal/External Boundary Layer Flows", Stanford University Report HMT-23 (1975).

93. S. V. Patankar and O. B. Spalding, "A Finite Difference Procedure for Solving the Equations of the Two Dimensional Boundary Layer", International Journal of Heat and Mass Transfer, Vol. 10, pp. 1389-1412 (1967).

94. B. E. Launder, C. H. Priddin and B. I. Sharma, "The Calculation of Turbulent Boundary Layers on Spinning and Curved Surfaces", Journal of Fluids Engineering, Vol. 99, pp. 231-239 (1977).

95. M. L. Koosinlin, B. E. Launder and B. I. Sharma, "Prediction of Momentum Heat and Mass Transfer in Swirling Turbulent Boundary Layers", Journal of Heat Transfer, Vol. 96, p. 204 (1974).

96. J. Moore, "A Wake and an Eddy in a Rotating Radial Flow Passage, Part 2: Flow Model", Journal of Engineering for Power, Vol. 95, pp. 213-219 (1973).

97. H. L. Moses, "A Strip Integral Method for Predicting the Behaviour of Turbulent Boundary Layers", Stanford University, Symposium on Turbulent Boundary Layers (August 1968).

98. I. M. Moon, "Effects of Coriolis Force on the Turbulent Boundary Layer in Rotating Fluid Mechanics:, M.I.T. Gas Turbine Laboratory, Report No. 74 (1964).

99. D. P. Sturge and N. A. Cumpsty, "Two-Dimensional Method for Calculating Separated Flow in a Centrifugal Impeller", Journal of Fluids Engineering, Vol. 97, pp. 581-597 (1975).

100. D. P. Sturge, "Compressible Flow in a Centrifugal Impeller with Separation: A Two Dimensional Calculation Method", Journal of Fluids Engineering, Vol. 100, pp. 37-46, (1978).

101. J. H. G. Howard and C. Osborne, "A Centrifugal Compressor Flow Analysis Employing a Set-Wake Passage Flow Model", Journal of Fluids Engineering, Vol. 99, pp. 141-147 (1977).

102. J. D. Stanitz and V. D. Prian, "A Rapid Approximate Method for Determining Velocity Destributions on Impeller Blades of Centrifugal Compressors", NACA TN 2421 (1951).

103. L. Walitt, J. L. Harp and C. Y. Liu, "Numerical Calculation of the Internal Flow Field in a Centrifugal Compressor Impeller", NASA CR-134984 (1975).

104. L. Walitt, C. Y. Liu and J. L. Harp Jr. "An Alternating Direction Explicit Method for Computing Three-Dimensional Viscous Flow Fields in Turbomachines" SAE Paper 781001 (1978).

105. W. R. Briley, J. P. Kreskovsky and H. McDonald, "Computation of the Three Dimensional Viscous Flow in Straight and Curved Passages", United Technologies Center Report R76-911841-9 for NAVAIR Systems Command (1976).

106. P. R. Dodge, "A Numerical Method for 2-D and 3-D Viscous Flows", AIAA paper No. 76-425 (1976).

107. P. R. Dodge and L. Lieber, "A Numerical Method for the Solution of the Navier-Stokes Equations with Separated Flow", AIAA paper 77-170 (1977).

108. A. K. Majumder, V. S. Pratap and D. B. Spalding, "Numerical Computation of Flow in Rotating Ducts," Journal of Fluids Engineering, Vol. 99 pp. 148-153 (1977).

109. S. V. Patankar and D. B. Spalding, "A Calculation Procedure for Heat, Mass and Momentum Transfer in Three Dimensional Parabolic Flows", International Journal of Heat and Mass Transfer, Vol 15, pp 1787-1806 (1972).

110. B. E. Launder and D. B. Spalding, "The Numerical Computation of Turbulent Flows", Computer Methods in Applied Engineering, Vol. 3, pp. 269-289 (1974).

111. A. K. Majumdar and D. B. Spalding, "A Numerical Investigation of Flow in Rotating Radial Diffusers", Imperial College of Science and Technology Report HTS/76/4 (1976).

112. A. K. Majumdar and D. B. Spalding, "A Numerical Investigation of Three Dimensional Flows in a Rotating Duct by a Partially-Parabolic Procedure", ASME paper 77-WA/FE-7 (1977).

113. R. E. Wagner and H. R. Velkoff, "Measurement of Secondary Flows in a Rotating Duct", Journal of Engineering for Power, Vol. 94, pp. 261-270 (1972).

114. C. Bosman, K. C. Chan and A. P. Hatton, "A Calculation Method for Incompressible, Blade-to-Blade Flow Through Radial Turbomachines with Log-Spiral Blade Surfaces", ASME paper 78-GT-195 (1978).

115. W. G. Cartwright, "The Determination of the Static Pressure and Relative Velocity Distribution in a Two Dimensional Radially Bladed Rotor", Institution of Mechanical Engineers Conference, University of Warwick, April 1973.

116. J. K. Unwala, "Velocity Distribution in the Rotor of a Raidal Turbomachine", University of Manchester Institute of Science and Technology, M.Sc. Thesis (1972).

117. A. Goulas and R. C. Baker, "Through Flow Analysis of Centrifugal Compressors", ASME paper 78-FT-110 (1978).

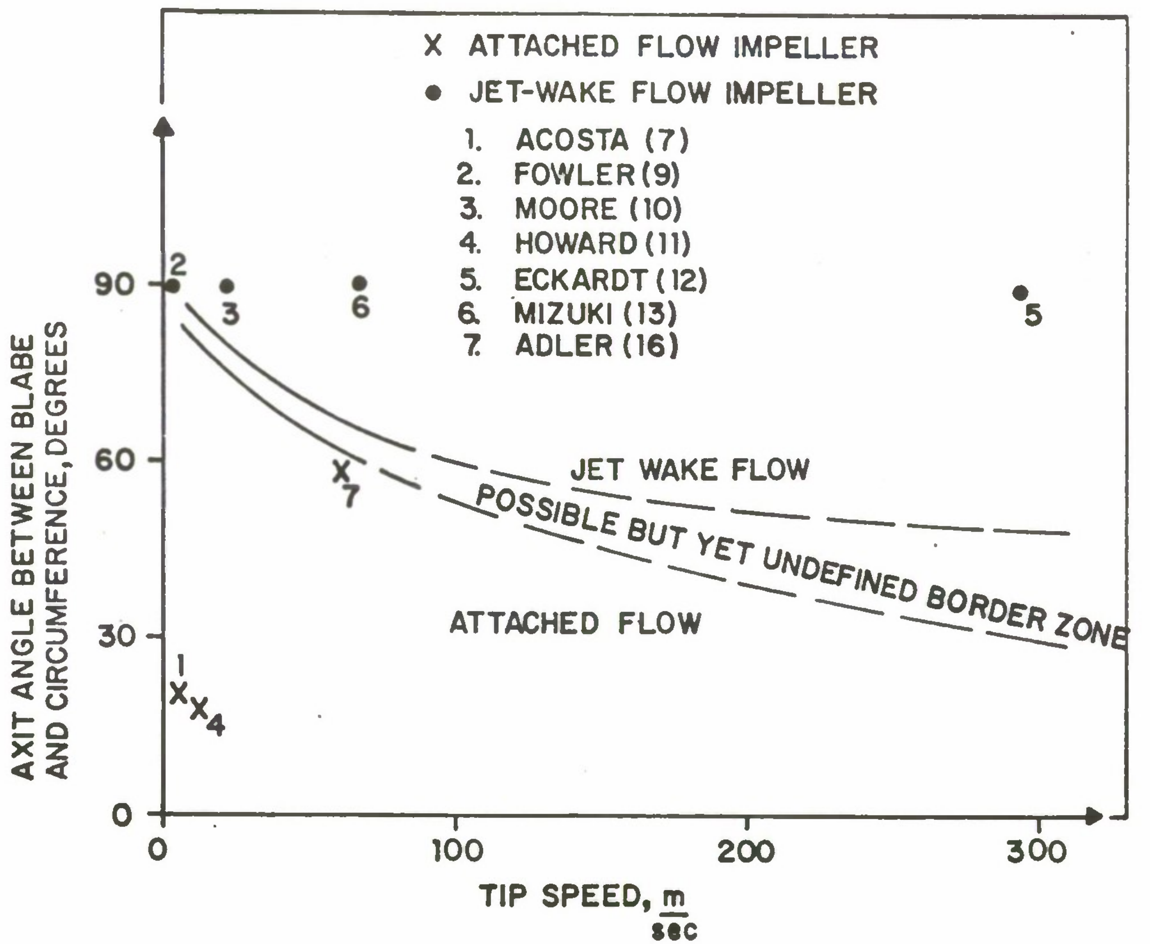


Figure 1: Possible flow regime classification in centrifugal impellers (the location and width of the border zone are assumed and have yet to be verified).

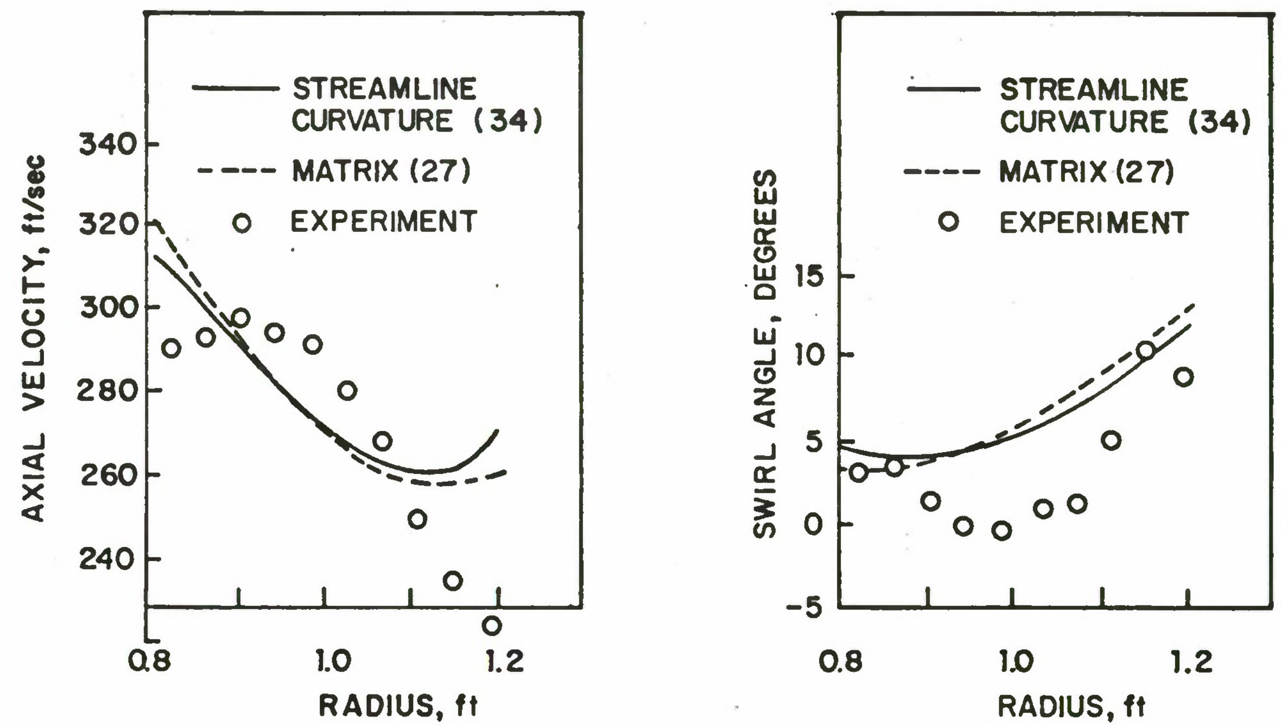


Figure 2: Comparison between experiment, Marsh's H-S matrix method [27] and Frost's streamline curvature method [34]. The example is an axial turbine taken from Ref. 34.

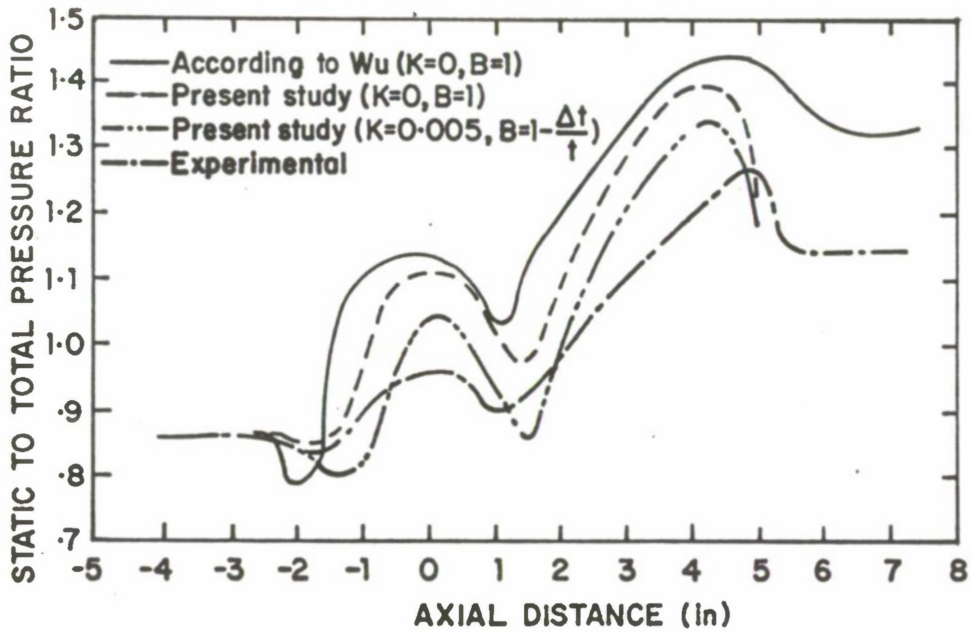


Figure 3: Comparison between experiment, Wu's H-S matrix method and Adler and Krimerman's H-S finite elements method [31]. The example is a mixed flow compressor taken from Ref. 31 (K is a total pressure loss coefficient, B is the dimensionless circumferential stream sheet thickness, t is the impeller pitch).

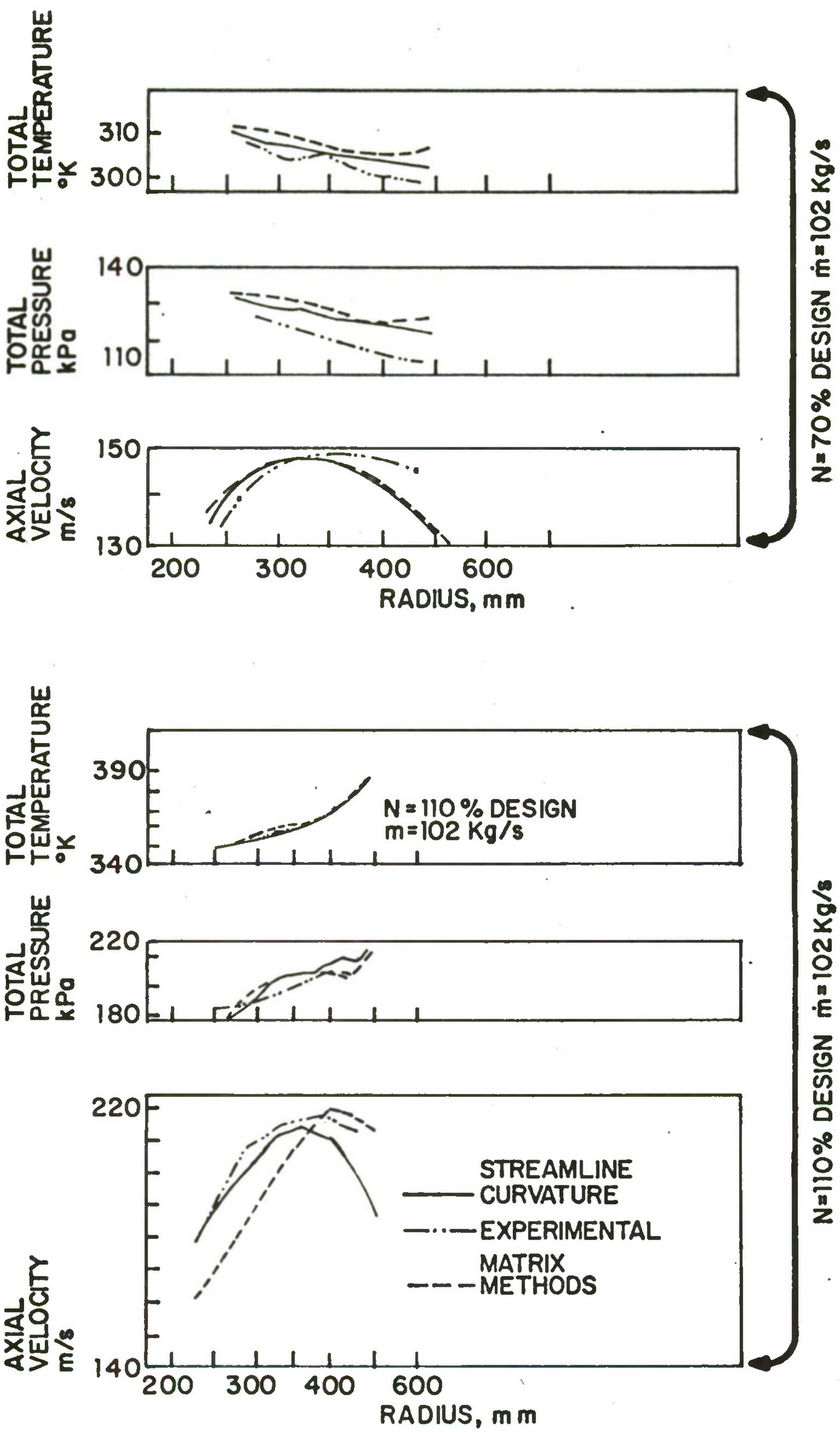


Figure 4: Comparison between experiments, a H-S matrix method and a H-S streamline curvature technique according to Davis and Millar [35]. The example is an axial compressor taken from Ref. 35.

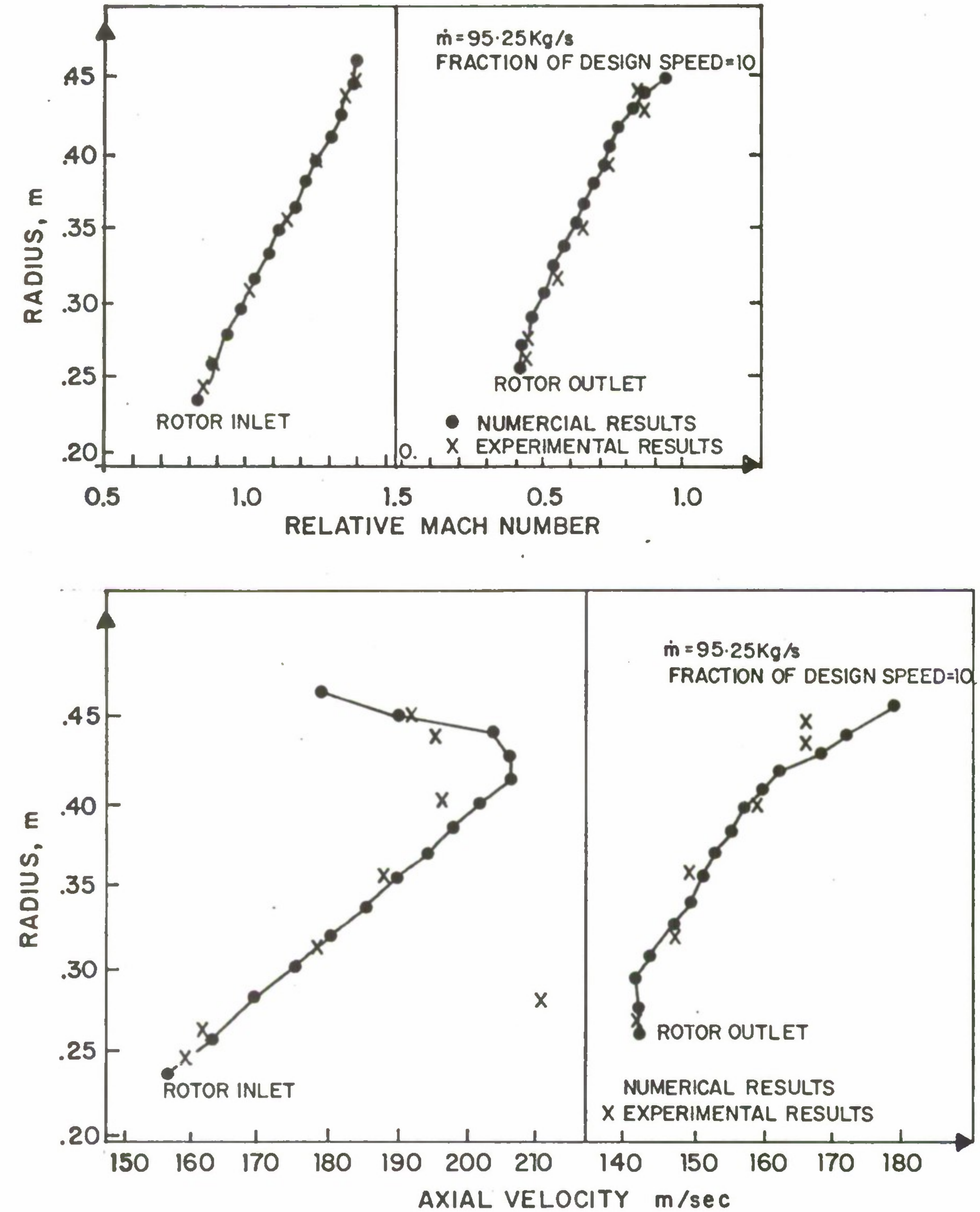


Figure 5: Comparison between experiments and Hirsch and Warzee's H-S finite elements calculations [33]. The example is an axial compressor taken from Ref. 33.

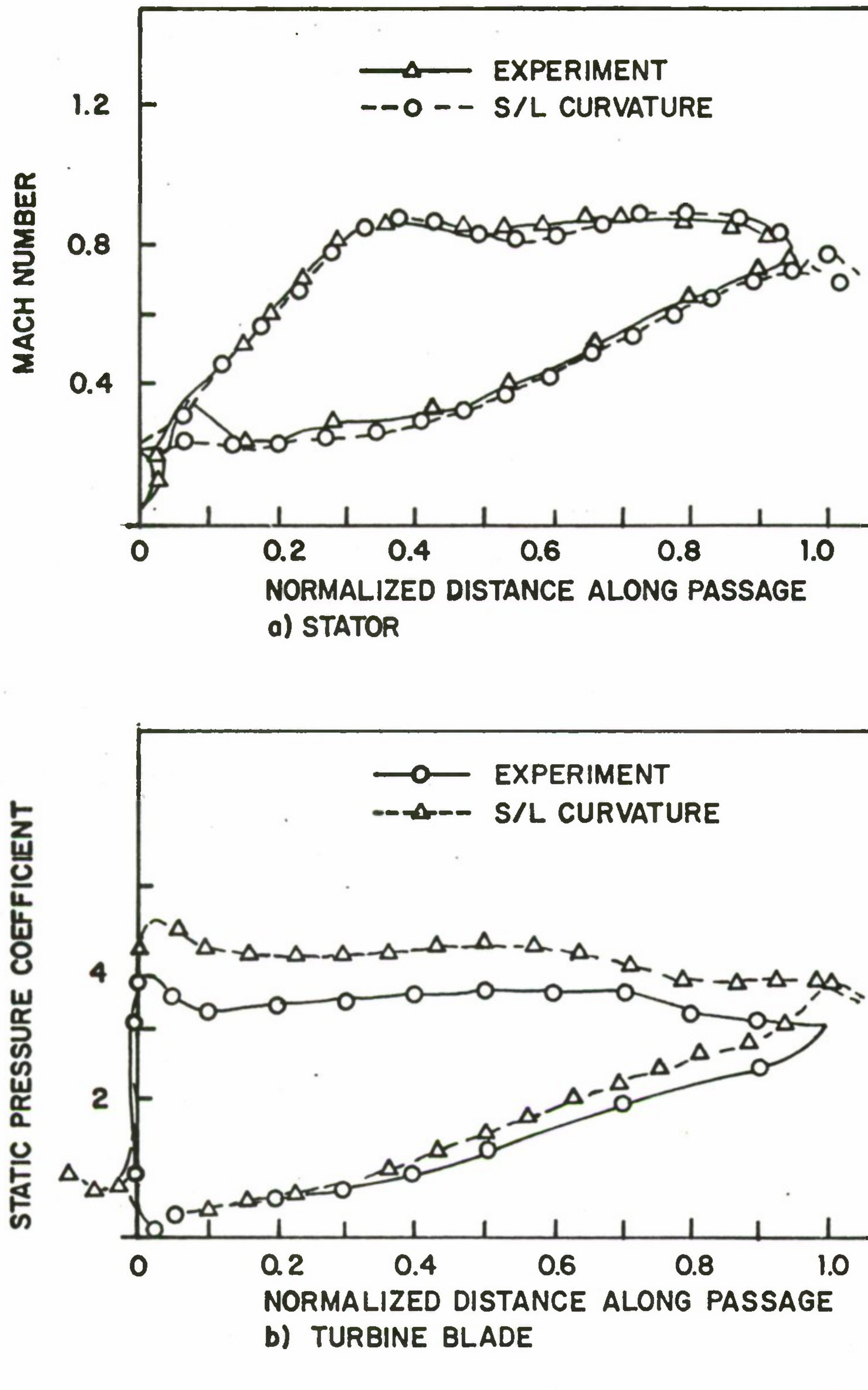


Figure 6: Comparison between experiments and Wilkinson's B-B streamline curvature calculations [43]. The examples are an axial compressor and an axial turbine taken from Ref. 43.

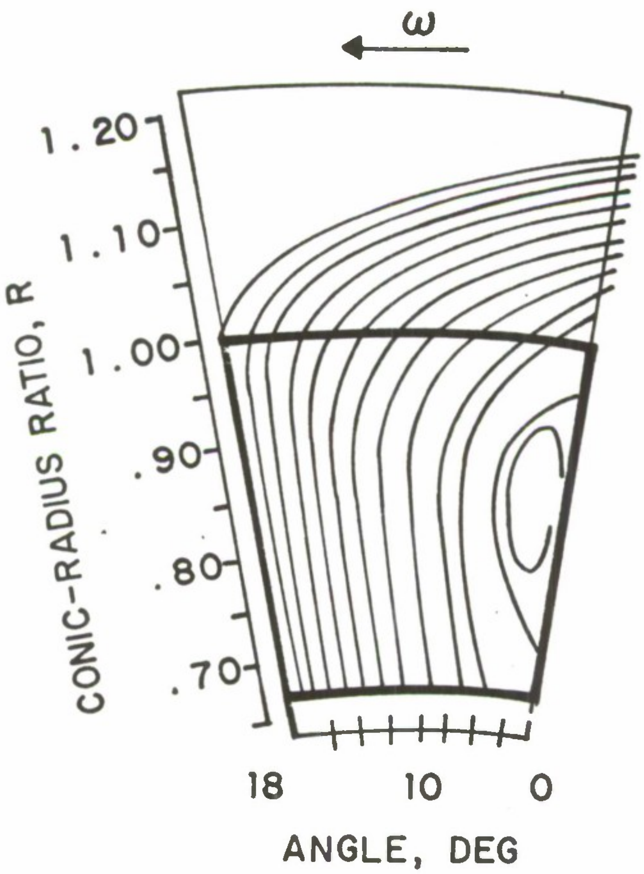
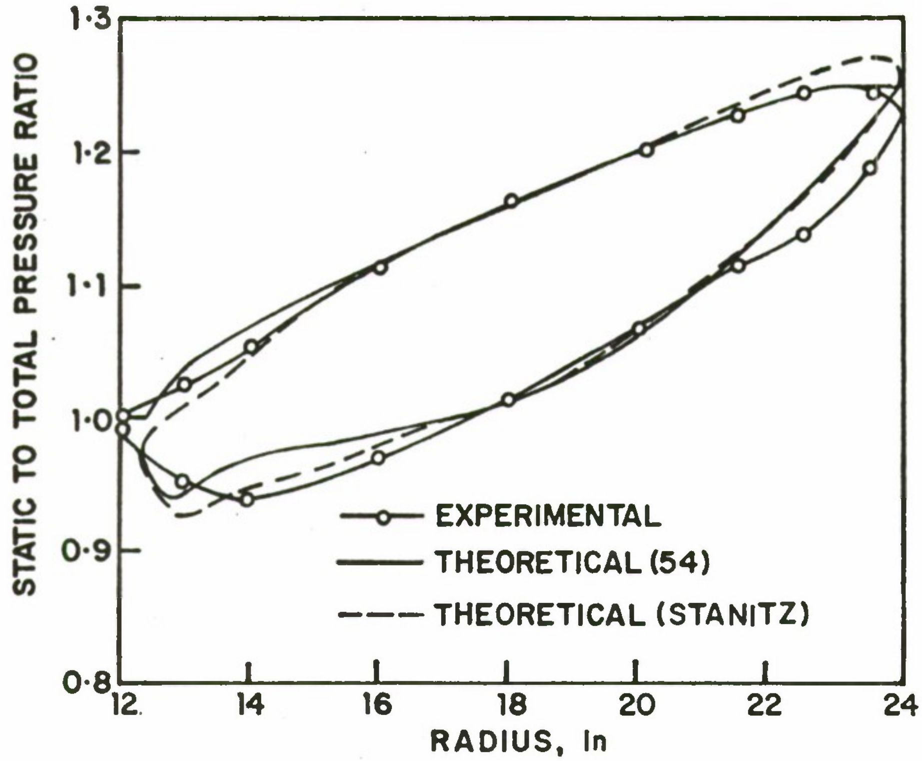


Figure 7: Streamlines inside the rotating impeller as calculated by Stanitz [45]. The predicted inviscid pressure side separation is well visible on the right side. Taken from Ref. 45.



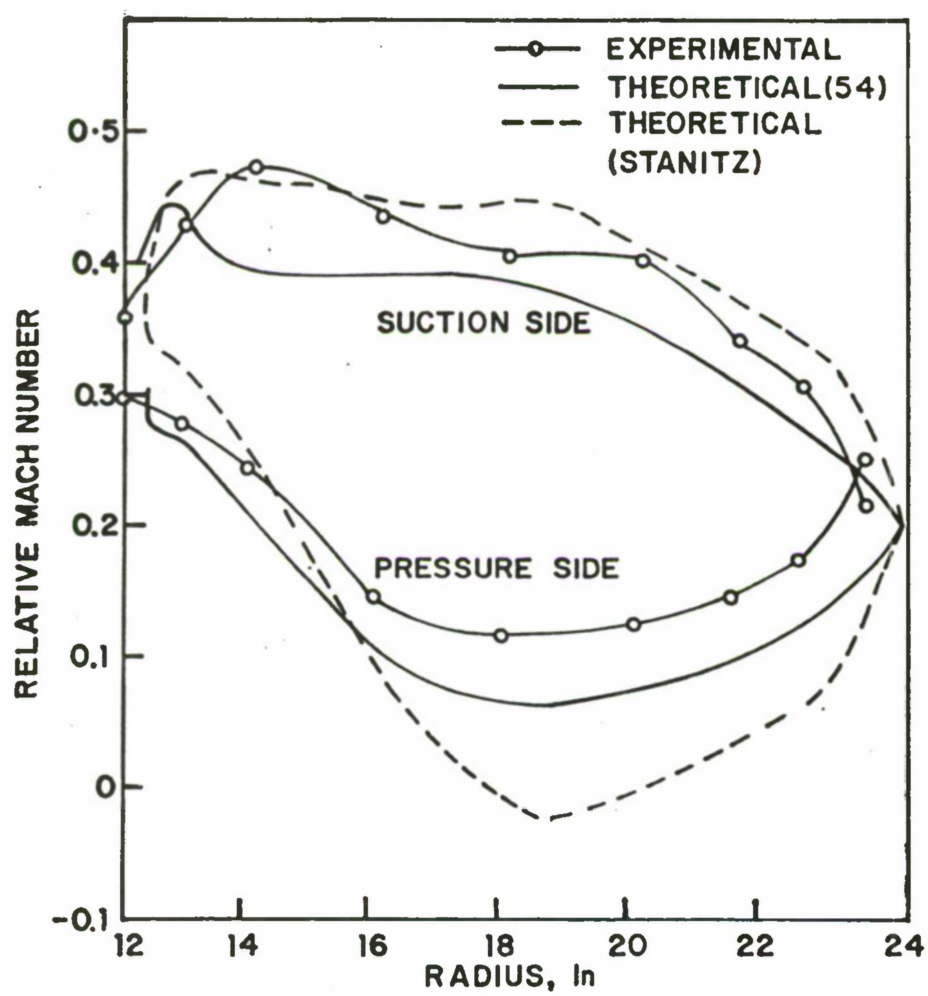


Figure 8: Comparison between experiments, Stanitz's B-B calculations and Adler and Krimerman's B-B calculations [54]. The example is a centrifugal impeller taken from Ref. 54.

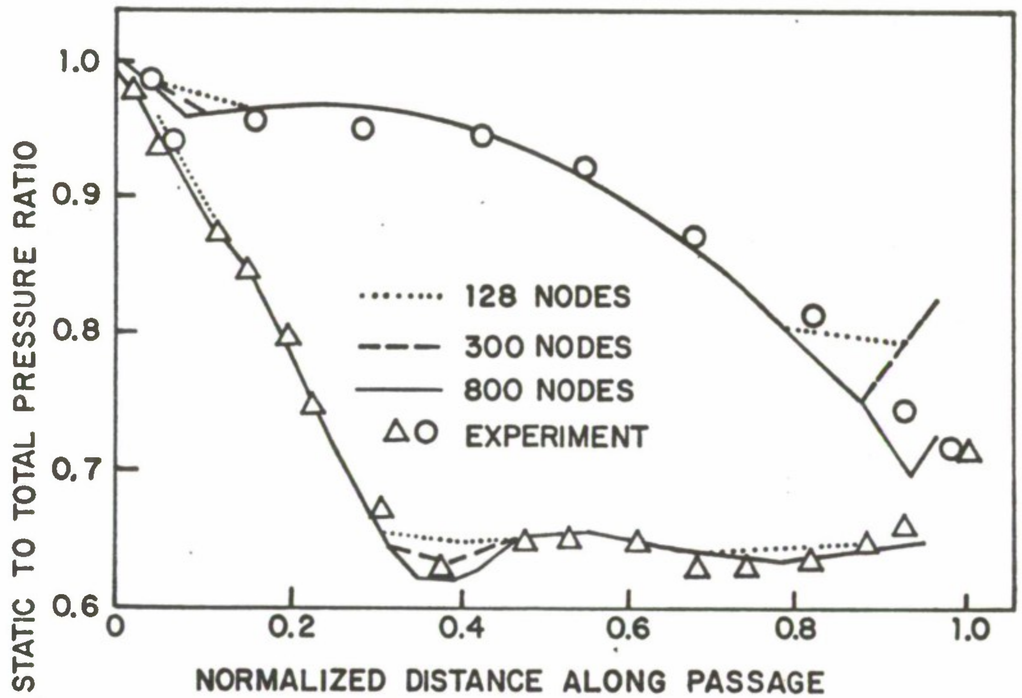


Figure 9: Comparison between experiments and B-B calculations by Prince [55]. The example is an axial turbine stator taken from Ref. 55.

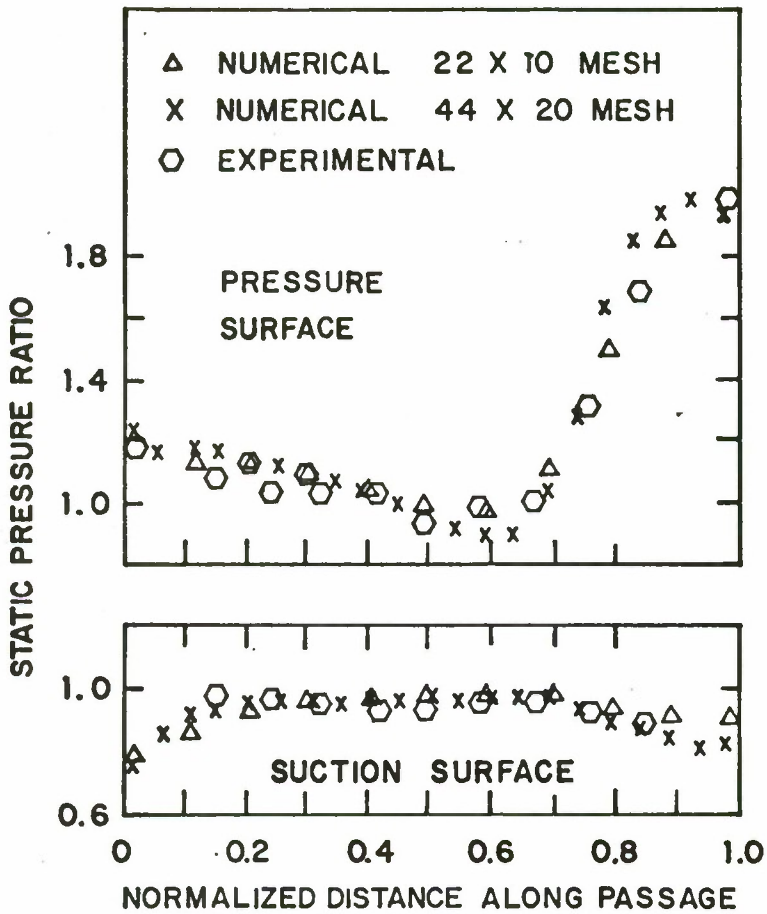


Figure 10: Comparison between transonic experiments and Kurzrock and Novick's B-B calculations [62]. The example is an axial compressor cascade taken from Ref. 62.

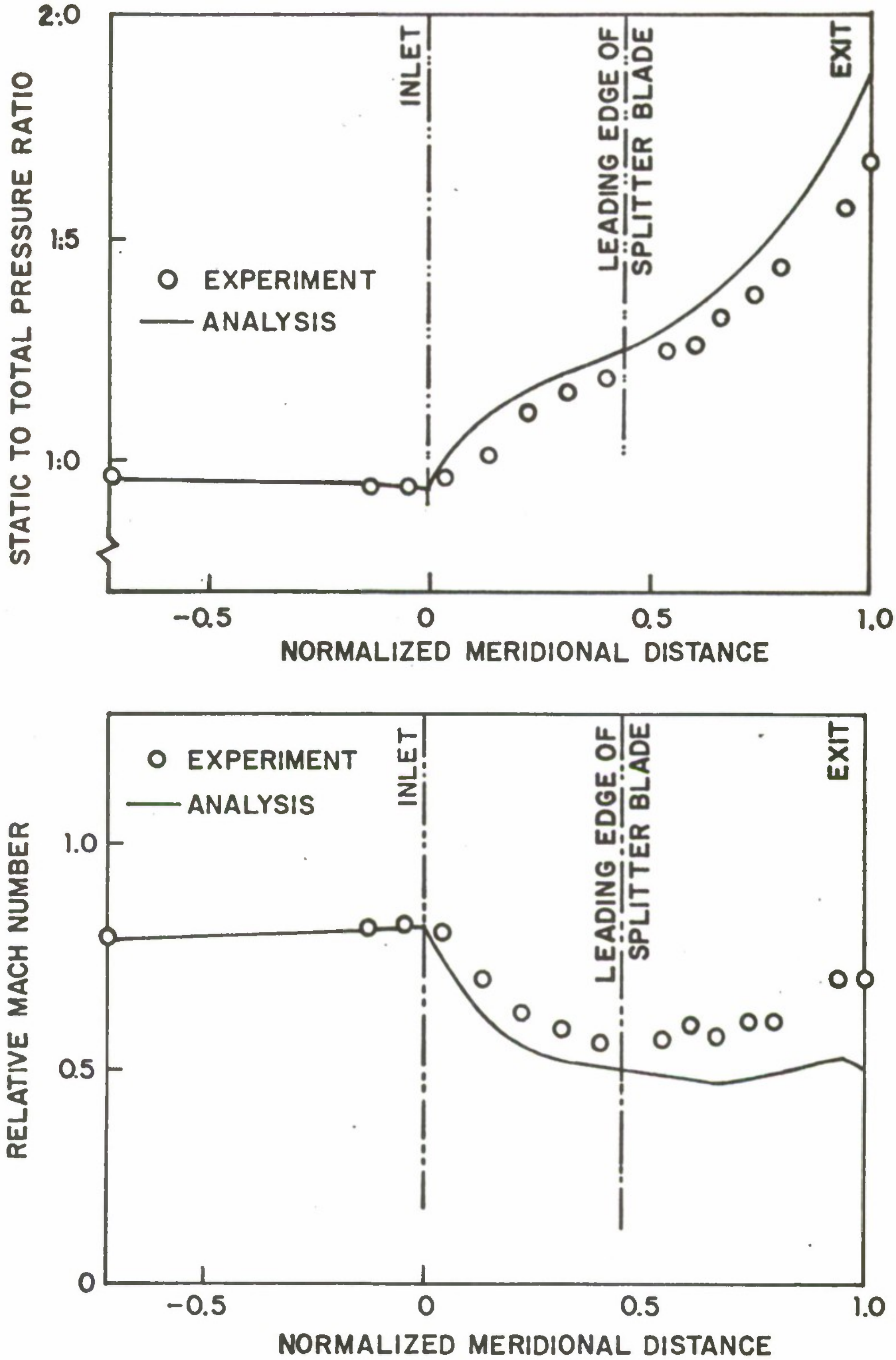


Figure 11: Comparison between experiments and Senoo and Nakase's quasi 3-D calculations [67]. The example is a centrifugal impeller taken from Ref. 18.

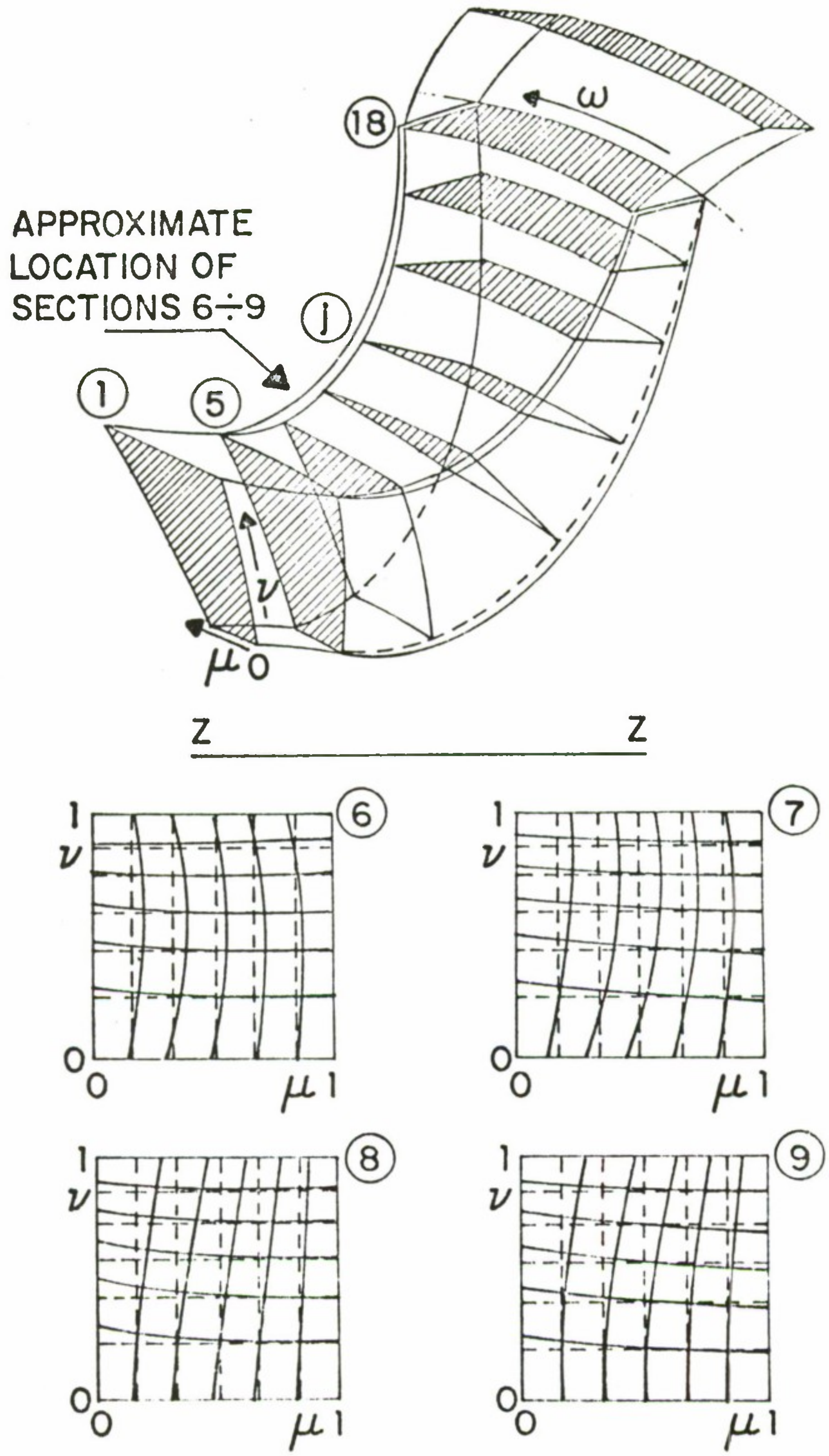


Figure 12: The difference between the traces of 2-D surfaces (dashed lines) and 3-D stream surface traces, calculated by Krimerman and Adler [74]. The example is a centrifugal impeller taken from Ref. 74. (the section 6 to 9 shown are located in the inducer and beyond, in the region where the flow undergoes a double-change of direction both in the tangential direction and from axial to radial).

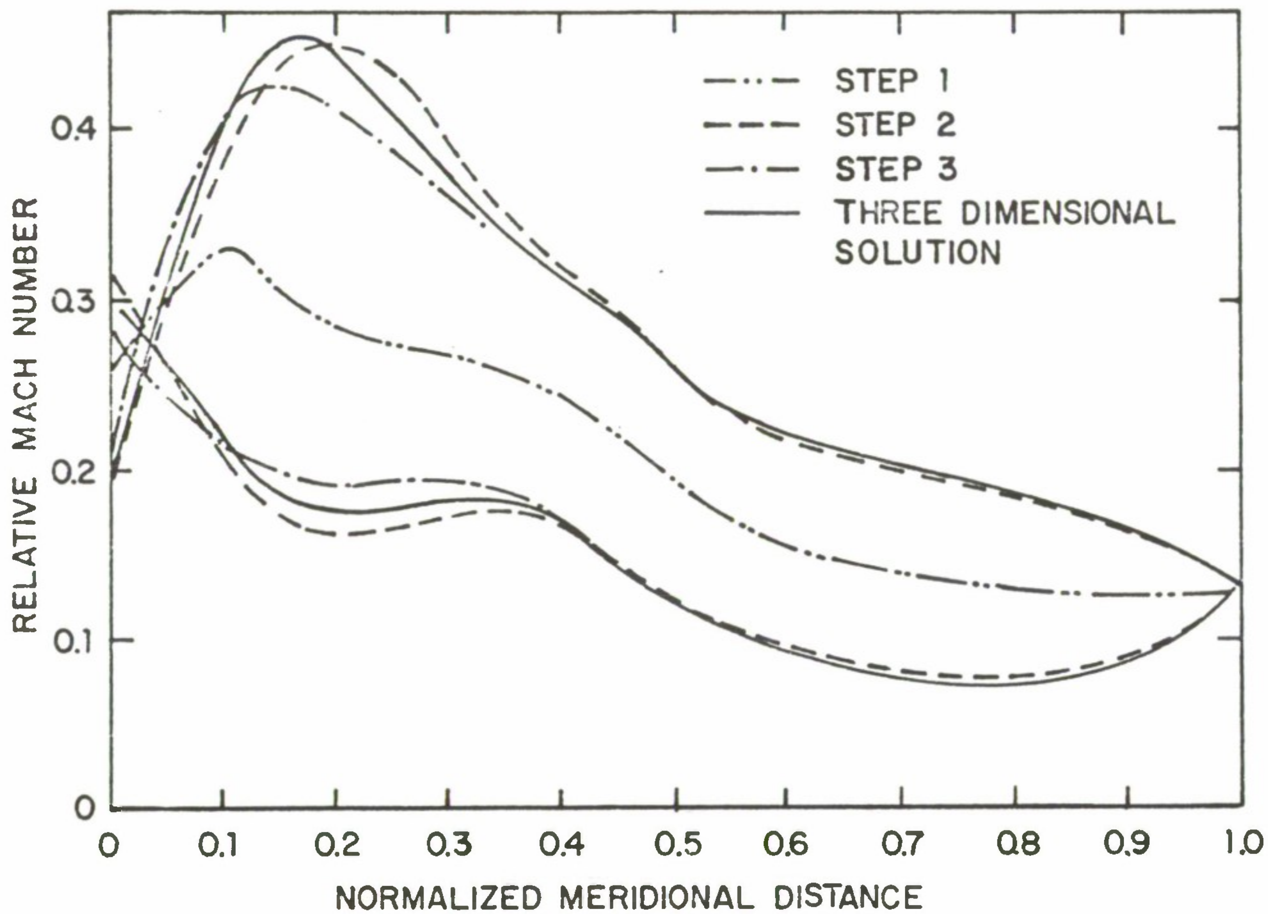


Figure 13: The convergence into the 3-D solution of the relative-velocity Mach number distribution around a centrifugal compressor blade, according to Krimerman and Adler's calculations [74]. Example taken from Ref. 74.

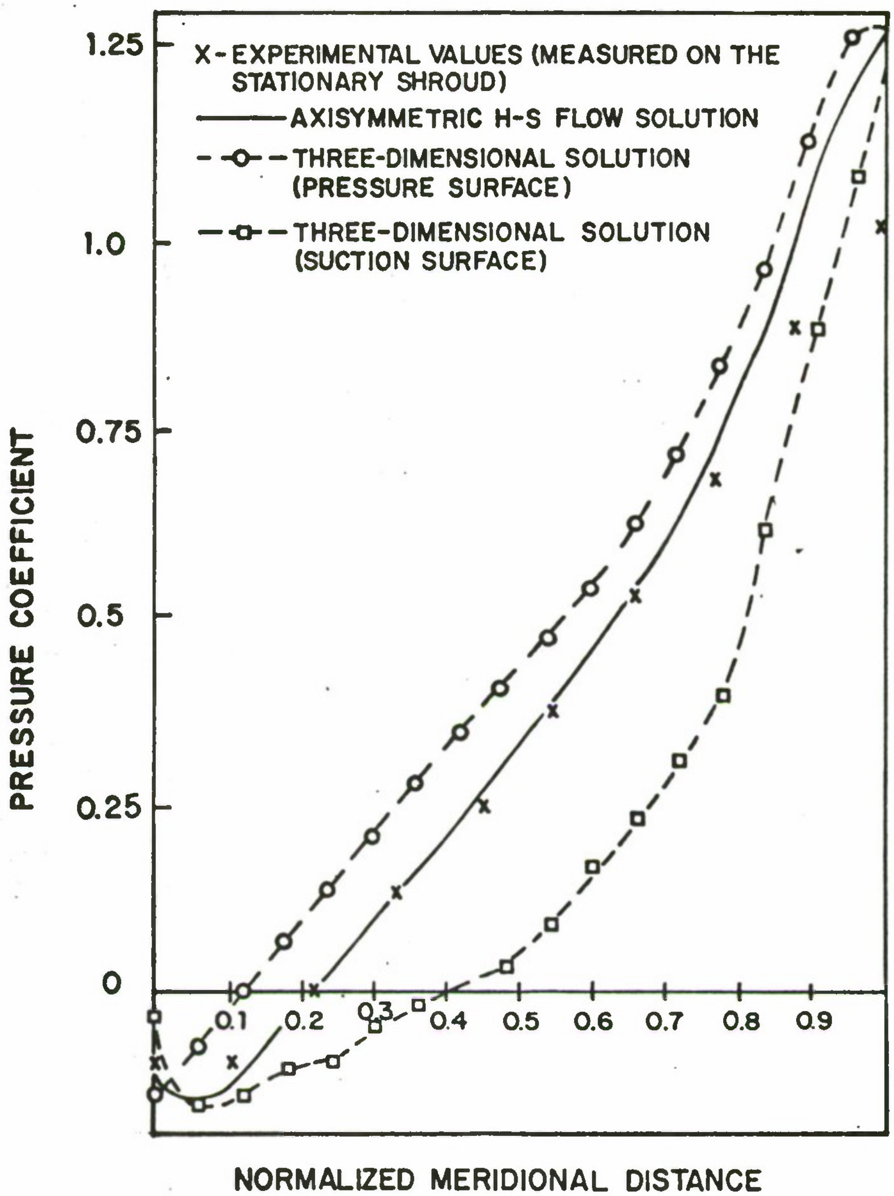


Figure 14: Comparison between experiments and 3-D inviscid calculations by Krimerman and Adler [74]. The example is a centrifugal radial exit impeller taken from Ref. 74.

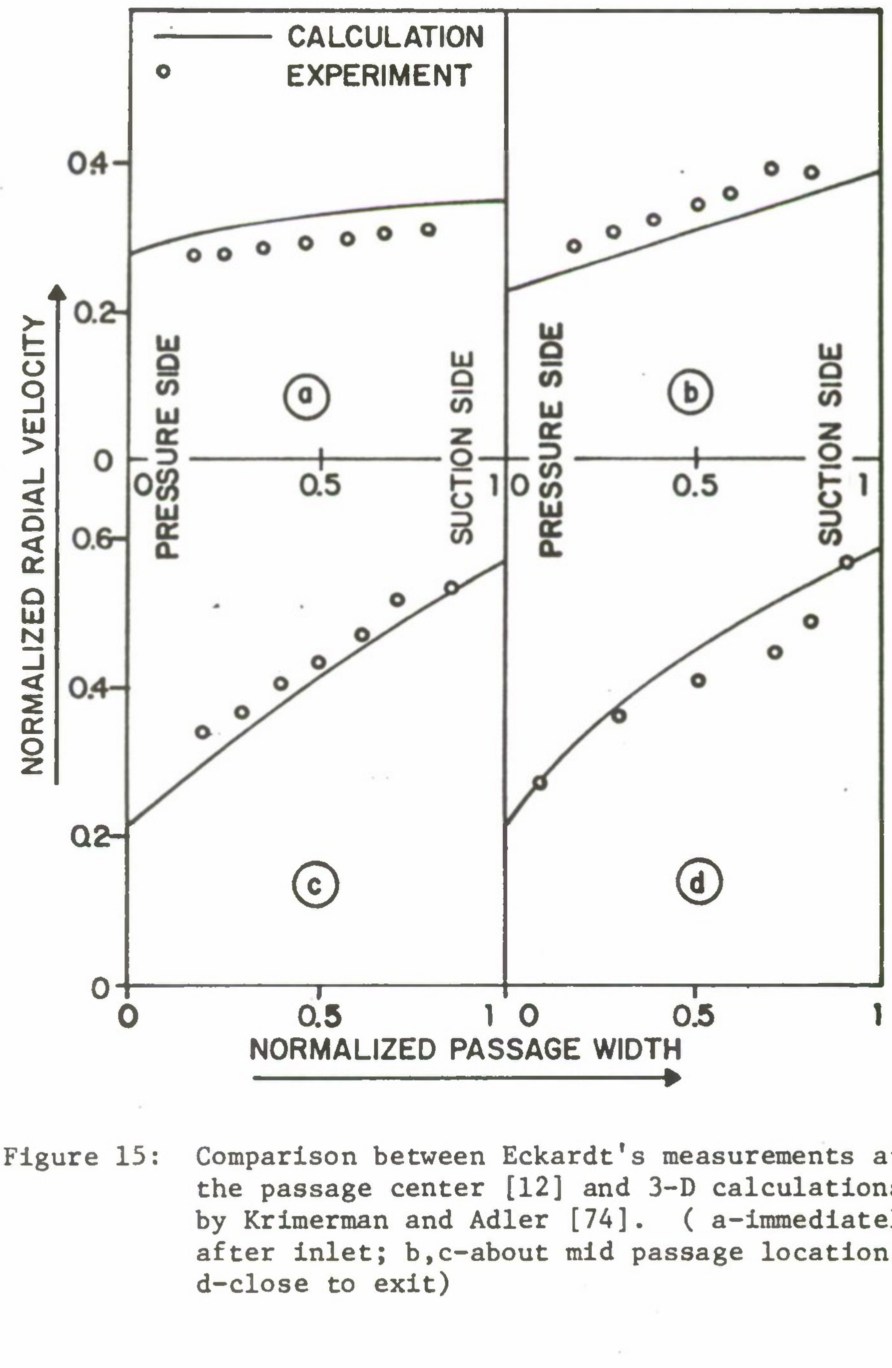


Figure 15: Comparison between Eckardt's measurements at the passage center [12] and 3-D calculations by Krimerman and Adler [74]. (a-immediately after inlet; b,c-about mid passage location; d-close to exit)

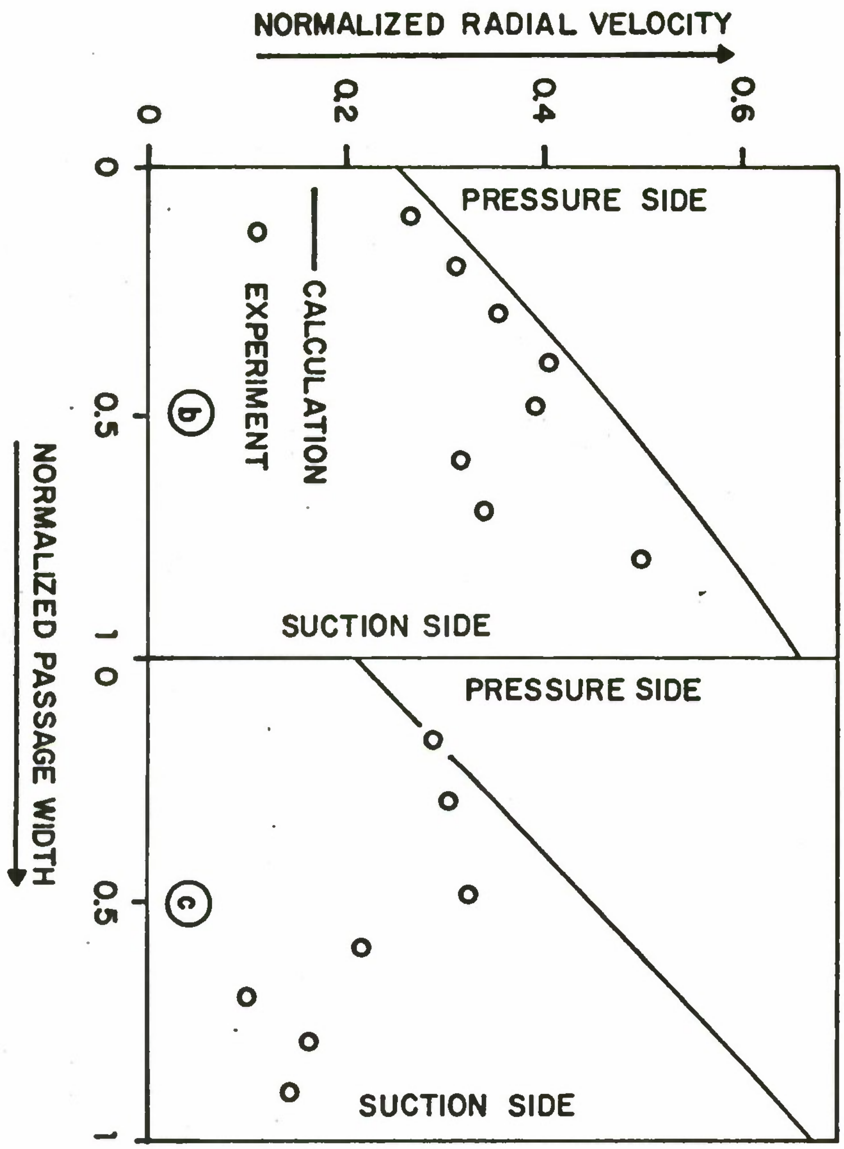


Figure 16: Comparison between Eckardt's measurements near the Shroud and 3-D calculations by Krimerman and Adler [74]. (b,c-about mid passage location)

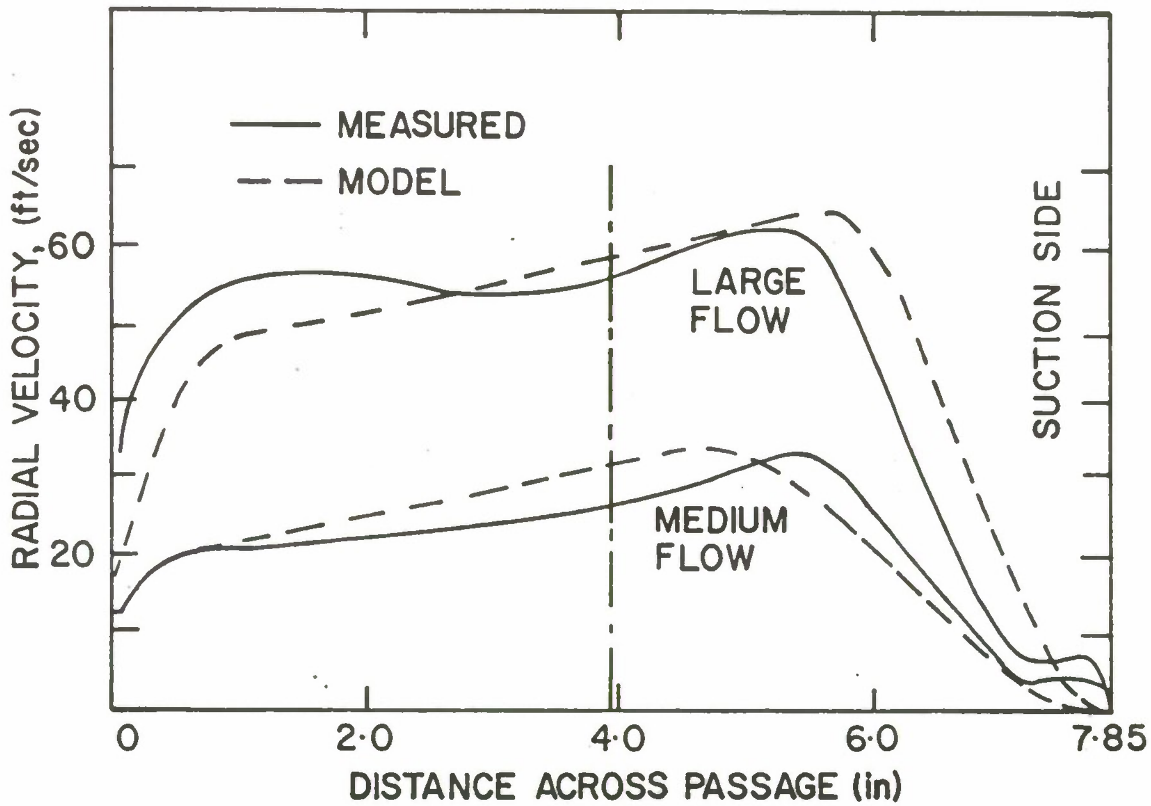


Figure 17: Comparison between the experiments and the calculations of J. Moore [96].

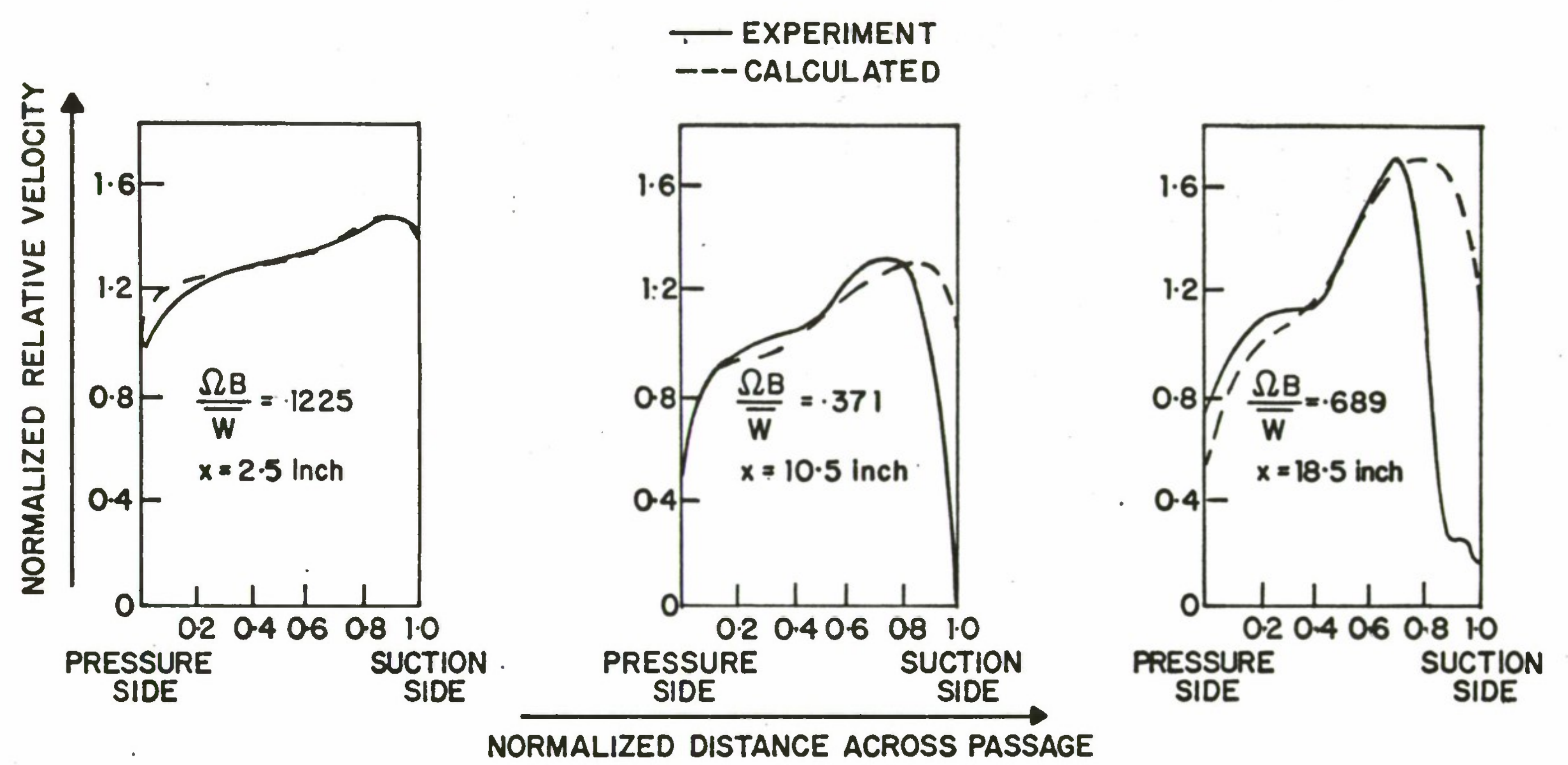


Figure 18: Comparison between the experiments of Moore [10] and Majumdar's et. al., calculations [108].

DISTRIBUTION LIST

		No. of Copies
1.	Defense Documentation Center Cameron Station Alexandria, Virginia 22314	2
2.	Library Code 0212 Naval Postgraduate School Monterey, California 93940	2
3.	Office of Research Administration Code 012A Naval Postgraduate School Monterey, California 93940	1
4.	Chairman Department of Aeronautics Code 57 Naval Postgraduate School Monterey, California 93940	1
5.	Turbo-Propulsion Laboratory Department of Aeronautics Naval Postgraduate School Monterey, California 93940	30
6.	Dr. H. J. Mueller Research Administrator Code 310A Naval Air Systems Command Navy Department Washington, D. C. 20360	2
7.	Mr. Karl H. Guttmann Code 330C Naval Air Systems Command Navy Department Washington, D. C. 20360	2
8.	Mr. James R. Patton, Jr. Power Program, Code 473 Office of Naval Research Arlington, Virginia 22218	1
9.	Commanding Officer Naval Air Propulsion Test Center Attn: Mr. Vernon Lubosky Trenton, New Jersey 08628	1

10. Mr. R. A. Langworthy 1
Army Aviation Materiel Laboratories
Department of the Army
Fort Eustis, Virginia 23604

11. National Aeronautics and Space Administration 1
Lewis Research Center (Library)
2100 Brookpark Road
Cleveland, Ohio 44135

12. Mr. Robert O. Bullock 1
AiResearch Mfg. Corporation
Division of Garrett Corporation
402 South 36th Street
Phoenix, Arizona 85034

13. CAG Library 1
The Boeing Company
Seattle, Washington 98124

14. Dr. F. O. Carta 1
United Technologies Research Labs
400 Main Street
Hartford, Connecticut 06108

15. Prof Jacques Chauvin 1
Universite D'Aix-Marseille
1 Rue Honnorat
Marseille, France

16. Mr. James V. Davis 1
Teledyne CAE
1330 Laskey Road
Toledo, Ohio 43601

17. Mr. Jean Fabri 1
ONERA
29, Ave. de la Division Leclerc
92 Chatillon
France

18. Mr. P. Tramm 1
Manager, Research Labs
Detroit Diesel Allison Division
General Motors
P. O. Box 894
Indianapolis, Indiana 46206

19. Mr. M. J. Hartmann 1
Chief, Fan and Compressor Branch
Mail Stop 5-9
NASA Lewis Research Center
21000 Brookpark Road
Cleveland, Ohio 44135

20. Prof Jack L. Kerrebrock, Chairman 1
Aeronautics and Astronautics Department
31-265 Massachusetts Institute of Technology
Cambridge, Massachusetts 02139

21. Dr. B. Lakshminarayana 1
Professor of Aerospace Engineering
The Pennsylvania State University
233 Hammond Building
University Park, Pennsylvania 16802

22. Library 1
General Electric Company
Aircraft Engine Technology Division
DTO Mail Drop H43
Cincinnati, Ohio 45215

23. Library 1
Pratt and Whitney Aircraft
Post Office Box 2691
West Palm Beach, Florida 33402

24. Library 1
Pratt and Whitney Aircraft
East Hartford, Connecticut 06108

25. Dr. J. F. Louis 1
Professor
Massachusetts Institute of Technology
Gas Turbine Lab 31-254
Cambridge, Massachusetts 02139

26. Dr. A. A. Mikolajczak 1
Pratt and Whitney Aircraft
Engineering 2H
East Hartford, Connecticut 06108

27. Prof Dr. L. G. Napolitano 1
Director
Institute of Aerodynamics
University of Naples
Viale C. Augusto
80125 Napoli
Italy

28. Prof Erik Nilsson 1
Institutionen for Stromningsmaskinteknik
Chalmers Tekniska Hogskola
Fack, 402 20 Goteborg 5
Sweden

29. Prof Gordon C. Oates 1
Department of Aeronautics and Astronautics
University of Washington
Seattle, Washington 98105

30. Prof Dr. K. Oswatitsch 1
Technische Hochschule
Karlsplatz 13
Vienna, Austria

31. Dr. P. A. Paranjee 1
Head, Propulsion Division
National Aeronautical Laboratory
Post Bag 1799
Bangalore - 17
India

32. Dr. Bruce A. Reese 1
Director, Jet Propulsion Center
School of Mechanical Engineering
Purdue University
Layfayette, Indiana 47907

33. Dr. W. Schlachter 1
Brown, Boveri-Sulver Turbomachinery Ltd
Dept. TDE
Escher Wyss Platz
CH-8023 Zurich
Switzerland

34. Professor Wu Chunghua 1
Academician
Chinese Academy of Sciences
Beijing, China

35. Dr. George K. Serovy 1
Professor of Mechanical Engineering
208 Mechanical Engineering Building
Iowa State University
Ames, Iowa 50010

36. Associate Prof R. P. Shreeve 1
Code 67sf
Naval Postgraduate School
Monterey, California 93940

37. Dr. Fernando Sisto 1
Professor and Head of Mechanical Engineering Department
Stevens Institute of Technology
Castle Point, Hoboken, New Jersey 07030

38. Dr. Leroy H. Smith, Jr. 1
Manager, Compressor and Fan Technology Operation
General Electric Company
Aircraft Engine Technology Division
DTO Mail Drop H43
Cincinnati, Ohio 45215

39. Dr. W. Tabakoff 1
Professor, Department of Aerospace Engineering
University of Cincinnati
Cincinnati, Ohio 45221

40. Prof Dr. W. Traupel 1
Institut fur Thermische Turbomaschinen
Eidg. Technische Hochschule
Sonneggstr. 3
8006 Zurich
Switzerland

41. Dr. Arthur J. Wennerstrom 1
ARL/LF
Wright-Patterson AFB
Dayton, Ohio 45433

42. Dr. H. Weyer 1
DFVLR
Linder Hoehe
505 Porz-Wahn
Germany

43. Mr. P. F. Yaggy 1
Director
U. S. Army Aeronautical Research Laboratory
AMES Research Center
Moffett Field, California 94035

44. Mr. J. S. Yampolsky 1
Gulf General Atomics, Incorporated
P. O. Box 608
San Diego, California 92112

45. Prof A. E. Breugelmans 1
Institut von Karman de la Dynamique des Fluides
72 Chausee de Waterloo
1640 Rhode-St. Genese
Belgium

46. Prof D. Adler 1
Technion Israel Institute of Technology
Department of Mechanical Engineering
Haifa 32000
Israel

47. Prof Dr. Ing Heinz E. Gallus 1
Lehrstuhl und Institut fur Strahlantiebe und
Turbourbeitsmashinen
Rhein.-Westf. Techn. Hochschule Aachen
Templergraben 55
5100 Aachen, Germany

48. Energy Laboratory 30
Technion Israel Institute of Technology
Department of Mechanical Engineering
Haifa 32000
Israel

49. Dr. Winterfeld 1
DFVLR
Porz-Wahn
W. Germany

50. Dr. Heilmann 1
Motoren-und Turbinen Union
P.O. Box 500640 8000 Munich 50
W. Germany

51. Mr. J. Sinai 1
Beit Shemesh Engines
Beit - Shemesh
Israel

52. Mr. Liban 1
Israel Aircraft Industries
Ben Gurion Airport
Israel

53. Professor V. Bhinder 1
Hatfield Polytechnic
England

54. Professor B. Gal-Or 1
Dept. of Aeronautical Engineering
Technion I. I. T.
Israel

55. Professor J. P. Gostelow 1
School of Mechanical Engineering
The New South Wales Institute of Technology
Australia

56. Sir W. Hawthorne 1
Churchill College
Cambridge
England

57. Professor J. P. Johnston 1
Stanford University
Department of Mechanical Engineering
Stanford, California 94305

58. Dr. J. Krimerman 1
Department of Mechanical Engineering
Technion I. I. T.
Haifa
Israel

59. Professor J. E. McCune 1
Department of Aeronautics
31-267 MIT
Cambridge, Massachusetts 02139

60. Dr. W. McNally 1
NASA
Lewis Center
Cleveland, Ohio 44135

61. Dr. T. Katsanis 1
NASA
Lewis Center
Cleveland, Ohio 44135

62. Professor J. Raabe 1
Technische Universität
Munich
W. Germany

63. Professor J. W. Railly 1
Department of Mechanical Engineering
University of Birmingham
Birmingham
England

64. Dr. K. Toyama 1
Komatsu Ltd.
3-20-1 Nakase
Kawasaki-Ku Kawasakishi 210
Japan

65. Professor F. Wallace 1
School of Engineering
University of Bath
Bath
England

66. Professor A. Stotter 1
Department of Mechanical Engineering
Technion I. I. T.
Haifa 32000
Israel

67. Professor Ch. Hirsch 1
Vrije Universiteit Brussel
Pleinlaan 2
1050 Brussel