Aerodynamic performance of an industrial centrifugal compressor variable inlet guide vane system

This item was submitted to Loughborough University's Institutional Repository by the/an author.

Additional Information:

- A Doctoral Thesis. Submitted in partial fulfillment of the requirements for the award of Doctor of Philosophy of Loughborough University.

Metadata Record: https://dspace.lboro.ac.uk/2134/7263

Publisher: © Miles Coppinger

Please cite the published version.
This item is held in Loughborough University’s Institutional Repository (https://dspace.lboro.ac.uk/) and was harvested from the British Library’s EThOS service (http://www.ethos.bl.uk/). It is made available under the following Creative Commons Licence conditions.

For the full text of this licence, please go to: http://creativecommons.org/licenses/by-nc-nd/2.5/
Aerodynamic Performance of an
Industrial Centrifugal Compressor
Variable Inlet Guide Vane System

by

Miles Coppinger

A Doctoral Thesis

Submitted in partial fulfilment of the requirements for the award of
Doctor of Philosophy of Loughborough University

September 1999.

© by Miles Coppinger, 1999.
ABSTRACT

Industrial centrifugal air compressors can require a combination of a large range of mass flow, high efficiency, constant pressure ratio, and constant rotational speed, specifically when used for sewage effluent aeration treatment. In order to achieve this performance it is common to use variable inlet guide vanes (VIGV's).

The performance characteristics of an existing VIGV design have been determined using both an experimental test facility and state of art numerical techniques. The results obtained from these techniques are far more comprehensive than earlier full-scale performance testing. Validation of the performance of the existing design using these techniques has led to the development of a new vane design and potential improvements to the inlet ducting geometry.

The aerodynamic interaction between the VIGV system and the centrifugal compressor impeller has also been investigated using a 3-D computational model of the complete variable geometry compressor stage. The results of these investigations have been validated by data available from full scale experimental testing. Strong correlation was obtained between numerical and experimental techniques, and a predicted improvement in polytropic efficiency up to 3% at low flow rates using the re-designed variable inlet guide vanes has been achieved.

The overall outcome of this research is a usable VIGV design technique for real industrial compressor environments, and confirmation that an acceptable design can be achieved that represents a rewarding improvement in performance.
I would like to express my sincere gratitude to many of my colleagues and friends for their support throughout the duration of this research project. I am especially grateful to the following people for the contributions they have made.

To my supervisor, Ed Swain, for his technical and moral support throughout this research project. Also to Henk Versteeg and Colin Garner for their guidance and direction of the project.

To Brian Mace, without whom I would probably still be building the experimental test rig.

To Richard Smith, David Brown, and Gerald Furness at Howden Donkin Blowers for their technical assistance and hardware.

To Olaf Niestroj, Chris Robinson, and everyone else at PCA Engineers for their guidance in developing the numerical technique.

To all of my colleagues at EDI for sharing my day-to-day problems.

To Amanda for her encouragement, hospitality, and profiteroles.

To Rob, Sue, Sarah, and Laura for providing welcome sanity and relief from this PhD.

Finally to my family for their understanding, encouragement, and inspiration to complete this work.
NOMENCLATURE

\( C_p \) specific heat \((J/kgK)\)
\( dQ \) heat added in time interval \((J)\)
\( dW \) work extracted in time interval \((J)\)
\( h \) specific enthalpy \((J/kg)\)
\( i \) incidence angle \((^\circ)\)
\( m \) mass flow rate \((kg/s)\)
\( P \) pressure \((Pa)\)
\( T \) temperature \((K)\)
\( U \) blade speed \((m/s)\)
\( V \) absolute velocity \((m/s)\)
\( V_A \) axial velocity \((m/s)\)
\( V_R \) radial velocity \((m/s)\)
\( V_W \) whirl velocity \((m/s)\)
\( W \) relative velocity \((m/s)\)
\( W \) power \((W)\)
\( Z \) height \((m)\)
\( \alpha_n \) absolute gas angle \((^\circ)\)
\( \alpha_{n+1} \) relative gas angle \((^\circ)\)
\( \beta \) blade angle \((^\circ)\)
\( \theta \) vane setting angle \((^\circ)\)

Subscripts:
- o stagnation
- 1,2 .. station 1,2 ..

Carter’s Rule:
\[ \delta = m\theta v(s/c) \]
where:
- \( \delta \) = deviation
- \( m = 0.23 \( (2a/c)^2 + 0.1 \( (\beta_2/50) \) \)
- \( \beta_2 \) = blade outlet angle
- \( \theta \) = blade camber angle
- \( s \) = spacing or pitch
- \( c \) = chord
- \( a \) = distance along chord to point of maximum camber
Chapter 1: Introduction

1.1 Background 1
1.2 Centrifugal Compressor Theory 3
1.2.1 Velocity Triangles and Performance 4
1.2.2 Limitations 7
1.2.3 Operation 9
1.3 Application 12

Chapter 2: State of Art

2.1 Variable Inlet Guide Vanes 14
2.1.1 Background 14
2.1.2 Function and Performance 15
2.2 VIGV and Variable Vaned Diffuser Combination 21
2.3 Aerodynamic Design 25
2.3.1 Inlet Ducting Design 25
2.3.2 Blade Design 32
2.4 Computational Fluid Dynamics 46
2.5 Conclusion 55

Chapter 3: Experimental Facility

3.1 Centrifugal Compressor Stage and VIGV System 57
3.2 Test Rig Design 58
3.3 Test Rig Components 60
3.3.1 Plenum Chamber 60
3.3.2 Variable Inlet Guide Vane System 61
3.3.3 Compressor 62
3.4 Instrumentation 62
3.4.1 Instrumentation Schedule 63
3.4.2 Instrumentation Accuracy 65
3.4.3 Instrumentation Control and Data Acquisition 66
3.5 Test Procedure 68
3.5.1 Data Reduction 68

Chapter 4: Experimental Results
4.1 Measurement Technique Validation 70
4.2 VIGV System Experimental Performance 71
4.2.1 Vane Performance 71
4.2.2 Overall Performance 75
4.2.3 Mass Flow Rate Scaling 78
4.3 Closure 79

Chapter 5: Numerical Techniques
5.1 Introduction 80
5.2 Computational Fluid Dynamics – An Overview 80
5.3 VIGV Simulations 81
5.3.1 Geometry Definition 81
5.3.2 Mesh Generation 84
5.3.3 Solving 88
5.4 Stage Simulations 90
5.4.1 Geometry Definition 90
5.4.2 Mesh Generation 92
5.4.3 Grid Interfaces 95
5.4.4 Solving 97
5.5 1-D Stage Performance Predictions 98
Chapter 9: Stage Performance

9.1 Introduction 160
9.2 VIGV – Impeller Interaction 161
9.3 Flat Plate Inlet Guide Vane Stage Performance Investigations 165
9.3.1 3-D Stage Computational Results - Flat Plate Vanes 166
9.3.2 Stage Experimental Results - Flat Plate Vanes 176
9.3.3 1-D Stage Computational Results - Flat Plate Vanes 177
9.4 Tandem Inlet Guide Vane Stage Performance Investigations 179
9.4.1 3-D Stage Computational Results – Tandem Vanes 179
9.4.2 1-D Stage Computational Results – Tandem Vanes 189
9.5 Comparison of Flat Plate and Tandem VIGV Stage Performance 190
9.6 Closure 191

Chapter 10: Conclusions

10.1 Contribution 194
10.2 Further Work 195
10.3 Publications 196

References 197
Appendix A 203
Appendix B 218
Appendix C 220
CHAPTER 1

INTRODUCTION

1.1 Background

Radial compressors have been employed for handling large volumes of air and gas since the early 1900's. Initially, these machines were classed as blowers due to their relatively low pressure ratio characteristics. However, during the 1940's, these turbomachines were developed to embrace more applications at higher pressures for both lesser and greater volumes. The requirements of gas and diesel engine superchargers, oil refineries, and petrochemical plants greatly extended the application of the centrifugal compressor. In addition to these duties, a major impetus for development was the use in gas turbines and originally they were preferred to axial compressors. A single stage radial compressor can deliver large mass flow rates of gases at high pressure ratios - as much as 10:1 in some gas turbine applications. The pressure ratios that a single stage centrifugal compressor can achieve would require several stages of an axial compressor. Typically, centrifugal compressors can be found in smaller gas turbines, particularly in helicopters, and all areas of process industry; gas boosting, process air duties, furnace applications, and gas exhausting.

Most modern industrial centrifugal compressors are required to operate over a broad range of flows and pressure ratios whilst maintaining optimum efficiency. The required operating range for a compressor is governed by the system the compressor is delivering fluid to or from, e.g. in the case of a turbocharger, the engine speed determines the volume flow rate required from the compressor. Unfortunately, the compressor can only operate in a stable and efficient manner within a certain region of its overall operating characteristic as shown in Figure 1.1.
At the minimum flow (surge), flow reversal through the compressor can occur. Not only is this unacceptable to the system in which the compressor is operating, but prolonged operation at this point can seriously damage the compressor itself. At the maximum flow (choke), the velocity of the fluid inside the compressor is at some point sonic and no further increase in flow is possible.

Fig. 1.1 Compressor performance map

It can be seen from the performance map that regulation of the mass flow rate at constant speed will result in a corresponding change in pressure ratio across the compressor stage. Although a centrifugal compressor can operate at any point along a constant speed line, its operating efficiency will vary as shown by the iso-efficiency lines above. For this reason, the design duty of the compressor will be aimed to lie within the region of maximum efficiency.
1.2 Centrifugal Compressor Theory

Figure 1.2 shows the components of a typical centrifugal compressor without inlet guide vanes. The fluid enters axially through the eye of the compressor, where it meets the leading edge of the inducer blades. The inducer is the upstream part of the impeller, and is required to ease the change in velocity of the fluid from the axial direction to a radial direction. Compressors without inducers are normally very noisy and inefficient due to large values of incidence and separation behind the blades (these concepts will be discussed later in this section). The main impeller vanes can be radial, backswep, or forward swept at the outer diameter (Figure 1.6), forged aluminium impellers may be rotating to give tip speeds as high as 550m/s. Each geometry results in different performance characteristics, but all share the same fundamental fluid mechanics, i.e. energy being imparted to the fluid by the rotating blades, resulting in a radial flow outwards toward the volute and a pressure rise due to the centrifugal and Coriolis forces within the impeller. The fluid, now at high velocity, enters a vaned or vaneless diffuser where the static pressure is recovered at the expense of velocity. From there it is collected in the scroll and passed to the diffusing part of the volute where a further increase in static pressure is effected.

Fig. 1.2 Typical centrifugal compressor
1.2.1 Velocity Triangles and Performance

Fluid flow within turbomachines can be conveniently represented by 2-dimensional velocity triangles. These triangles depict the individual vectors of flow patterns immediately upstream and downstream of aerodynamic components such as blades and vanes. Figure 1.3 shows a typical centrifugal compressor radial impeller and inducer with inlet velocity triangle. In this case the inlet is assumed to be axial (no prewhirl).

![Inlet velocity triangle](image1)

If, however, prewhirl is induced, the inlet velocity triangle will appear similar to that shown in Figure 1.4 below.

![Inlet velocity triangle with prewhirl](image2)
In the flows represented by the velocity triangles in Figures 1.3 and 1.4, it is assumed that the relative flow is in line with the vane. In reality, the inlet relative velocity vector is more likely to meet the blade at an angle of incidence (i), and the magnitude of the incidence will vary with mass flow rate as shown in Figure 1.5.

Fig. 1.5 Variation of incidence with mass flow rate

The fluid flow leaving the impeller can be represented by outlet velocity triangles. Figure 1.6 shows the effect of impeller backward or forward sweep on the shape of the resulting velocity triangle.

Fig. 1.6 Impeller outlet velocity triangles
As the fluid enters the impeller of the compressor a velocity is imparted to it along with a radial change in position, these result in a change in momentum and a resultant force on the impeller. Momentum changes in the tangential direction lead to an impeller torque and thus work, or power. The power absorbed by the centrifugal compressor stage is therefore governed by an equation derived from the velocity triangles known as the Euler Equation. In its simplest form, it describes the work done per unit mass flow in relation to the inlet and outlet velocity components as follows:

\[ W = \dot{m} \left( U_2 V_{W2} - U_1 V_{W1} \right) \]

In addition, the steady flow energy equation can be written as follows:

\[ \Delta Q - \Delta W = \dot{m} \left[ (h_2 - h_1) + \frac{(V_2^2 - V_1^2)}{2} + g(z_2 - z_1) \right] \]

If this is expressed in terms of work rate and mass flow rate, with adiabatic conditions (no heat exchange), ignoring the change in potential energy, and if stagnation enthalpy is used, it can be re-written as:

\[ W = \dot{m} (h_{02} - h_{01}) \quad \text{or} \quad W = \dot{m} C_p (T_{02} - T_{01}) \]

When combined with the Euler equation, it can be seen that the stagnation enthalpy change in a turbomachine is related to the velocity triangles:

\[ W = \dot{m} \Delta h = \dot{m} C_p \Delta T = \dot{m} \left( U_2 V_{W2} - U_1 V_{W1} \right) \quad (\text{Eqn. 1.1}) \]
1.2.2 Limitations

The operating range of a centrifugal compressor is limited, as described in section 1.1, by a number of aerodynamic phenomena; choke, surge, and rotating stall.

Choke arises when sonic conditions have been reached at some point within the compressor. No further increase in mass flow is achievable at this point and shock waves may well form, leading to increased viscous losses within the compressor.

Surge occurs when the flow within the compressor completely breaks down, resulting in rapid changes in the mass flow. This in turn induces violent pressure oscillations and the propagation of pressure waves, and in extreme cases a complete flow reversal. It results in severe vibration and damage to the compressor unit as well as reduced efficiency. Considerable effort has been expended to understand this phenomenon and control it. A concise list of surge detection and suppression techniques is offered by Botros and Henderson\(^1\). Suppression techniques are split into two categories; operational methods such as variable geometry, and design methods such as backsweped impellers, vaneless diffusers and casing treatment. Since surge is normally caused by stalling of the flow onto the diffuser vanes, the surge line of a vaneless diffuser compressor will be further to the left than that of the equivalent vaned machine.

Surge is also often attributed to rotating stall in the impeller, although rotating stall can exist as a separate phenomenon in an apparently stable operating condition. When the incidence angle onto a blade is excessive the blade will stall (there will be a breakdown in flow on the suction surface of the blade leading to high losses). If this blade is rotating, the flow breakdown will affect the neighbouring blades resulting in a propagation of stall around the periphery. This creates the effect of cyclic loading and unloading of blades, which after prolonged operation can result in catastrophic failure of the compressor.

Jansen et al\(^2\) experimented with impeller wall treatment and vaned diffuser treatment in an effort to extend the centrifugal compressor surge margin, i.e. to reduce the mass flow at which surge occurs for a particular machine. Circular grooves in the wall
were machined along with slotted diffuser vanes in order to inhibit the onset of rotating stall. The wall treatment methods offered limited success in extending the surge margin, and in some cases resulted in a slight decrease in efficiency, Figure 1.7. The vaned diffuser treatment resulted in a significant extension of the stable flow range by increasing the choke flow. Unfortunately an appreciable drop in efficiency was also observed, as shown in Figure 1.8.

![Fig. 1.7 Wall treatment](image1)

![Fig. 1.8 Vaned diffuser treatment](image2)

Work on casing treatment has continued in both centrifugal and axial compressors in an effort to increase the surge (or stall) margin. Kang et al [31] reported significant improvements in the stall margin of axial compressors using a recessed casing treatment that worked in a similar fashion to the casing grooves in centrifugal machines.
1.2.3 Operation

The efficiency of a centrifugal compressor is optimised at its design duty for a particular application. However, high efficiency is also required at lower flows as performance at these flow rates can have a considerable affect on plant operating costs. As the operating point moves to the left on the characteristic curve, i.e. lower flow, it approaches the surge line - an area of extremely unstable operation. If the design point of the compressor is already close to the surge line, then very little turndown in flow is possible. If, however, it was possible to move the surge line further to the left during operation, then a larger turndown in flow would be possible. The main methods of governing the mass flow rate delivered by the compressor fall into three categories; suction throttling, variable speed, and variable geometry.

Suction throttling via a valve located upstream of the compressor inlet regulates the flow by introducing a pressure loss upstream of the compressor. As the valve pressure loss is increased, the overall efficiency of the compressor system will decrease considerably. The user is therefore restricted to a small potential turndown in flow if significant losses are to be avoided.

Variable speed control allows the characteristic curve to be ‘moved up or down’, i.e. an increase in speed shifts the curve up, and a decrease in speed shifts the curve down. This can be seen from the constant speed curves shown in Figure 1.1. Thus, in order to operate at peak efficiency, this type of control is most appropriate when the user requires a change in head rather than a significant change in flow. If the compressor is turbine driven, speed variation is applicable and relatively inexpensive. However, if it is electric motor-driven, then speed control can be an expensive method.

The term ‘variable geometry’ is used here to describe both variable vaned diffusers and variable inlet guide vanes (VIGV's). Vaned diffusers are used to reduce the high kinetic energy leaving the impeller and rapidly recover the static pressure of the fluid. In a fixed vaned diffuser the incidence onto the diffuser vanes will vary with the mass flow rate of the fluid, and can result in a stall within the diffuser, which can lead to compressor surge. If however, the diffuser vane angle is variable, the incidence can
be controlled and the operating range of the compressor increased. The objective of variable inlet guide vanes is primarily to introduce a swirl component into the axial inlet flow velocity, thus reducing the work done by the stage. As a result of the blockage that they introduce into the flow path, they also create a pressure loss that has a similar effect to upstream throttling.

Stebbins\(^4\) presented sample calculations on the installation and running costs of various flow control devices, and calculated the approximate payback time for each device. Through the consideration of installation and running costs of variable speed drives and inlet guide vanes, a payback time was derived in comparison with suction throttling. His findings are summarised in Table 1.1 and demonstrate that, in this case, variable speed drives offer a significantly shorter payback than inlet guide vanes.

<table>
<thead>
<tr>
<th></th>
<th>Suction Throttling</th>
<th>Inlet Guide Vanes</th>
<th>Variable Speed</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weighted hp</td>
<td>35</td>
<td>24.76</td>
<td>18.06</td>
</tr>
<tr>
<td>x 0.746 kW/hp</td>
<td>0.746</td>
<td>0.746</td>
<td>0.746</td>
</tr>
<tr>
<td>x 8000 hr/yr</td>
<td>8000</td>
<td>8000</td>
<td>8000</td>
</tr>
<tr>
<td>= kWh/yr</td>
<td>208,880</td>
<td>147,768</td>
<td>107,782</td>
</tr>
<tr>
<td>x $0.08/kWh</td>
<td>$0.08</td>
<td>$0.08</td>
<td>$0.08</td>
</tr>
<tr>
<td>= $/yr</td>
<td>$16,710</td>
<td>$11,821</td>
<td>$8,623</td>
</tr>
<tr>
<td>Annual saving over suction throttling</td>
<td>0</td>
<td>$4,889</td>
<td>$8,087</td>
</tr>
<tr>
<td>Estimated cost to implement</td>
<td>0</td>
<td>$7,620</td>
<td>$8,145</td>
</tr>
<tr>
<td>Simple payback in years</td>
<td>__</td>
<td>1.56</td>
<td>1.01</td>
</tr>
</tbody>
</table>

Table 1.1 Payback times for flow control methods

Turton\(^5\), however, suggested that an inverter motor drive for speed control would cost £4500 involving a payback time of 2.6 years in terms of the saved energy at 1982 prices. The cost of an inlet guide vane system could be of the order of £1750 with a
payback time of 1.3 years. The application being considered was a variable air-volume air conditioning system.

Other test cases and predictions for a variety of compressor configurations incorporating variable geometry systems have been presented by Kano et al[6], Ledder[7], Lo[8], Halloran[9], and Eshelman[10]. They all highlight the need for the designer to consider the nature and operating schedule of the system in which the fan or compressor is to be incorporated. These factors should be considered when choosing the most appropriate method of flow control.

An experimental comparison of variable speed control and inlet guide vanes is offered by Williams[11]. He discovered that the efficiency losses associated with flow control were not greatly different for the two methods. The losses observed with inlet guide vanes were attributed to the vanes themselves rather than their effect on the actual compressor stage. Turndown in flow up to 60 percent of the design duty was possible with inlet guide vanes.

Williams suggested that inlet guide vanes are the favoured method for achieving large turndowns in flow for the following reasons:

i) A wide range of flows can be accommodated simply, efficiently and cost effectively, especially for simpler processes such as waste water treatment.

ii) Inlet guide vanes allow for a potential for future growth of around 15-20 percent in flow whilst maintaining a substantial turndown for operational use. This is of particular relevance in petrochemical processes where the composition of the gas can vary with time.

iii) Operation with two compressors driven from the same prime mover where the flow through one of the compressors might need to be regulated.

He concluded that wide flow ranges and high overall efficiencies can be achieved by a combination of good impeller design and inlet guide vanes.

It should be noted that, although prewhirl is normally imparted through variable inlet guide vanes, aerodynamically induced prewhirl has also been investigated in a
number of test cases. Various proposals have been made to use high pressure jets or inlet volutes to produce prewhirl, Kyrtatos\textsuperscript{12}, Kyrtatos and Watson\textsuperscript{13}, Whitfield and Abdullah\textsuperscript{14}.

1.3 Application

A particularly demanding application for centrifugal compressors is that of aeration treatment of sewage. The compressors supply the aeration tanks with a flow of air which is bubbled up through the effluent. Since the effluent is maintained at a constant depth, the compressors must supply air at a constant pressure. The flow rate of the effluent may however vary, resulting in a change in the number of tanks used and the consequential varying air flow rate demand. The overall requirement on the compressors is an extremely wide range of flow at a constant pressure ratio, whilst maintaining the highest possible compressor efficiency. The combination of these operating requirements with the fact that the compressors are normally driven by fixed speed electric motors represents a considerable challenge to the designer. It has been suggested by Swain and Connor\textsuperscript{15} that the most appropriate solution to this problem is the combination of a variable vaned diffuser, variable inlet guide vanes and a backswept impeller.

In this application, typically uncambered flat plate VIGV's are used. At $0^\circ$ setting angle the loss must be at a minimum, in this position cambered vanes would result in a significant disturbance to the axial flow and hence higher losses than would be caused by flat plates. Unfortunately, the use of flat plate vanes at large setting angles results in both high losses and a significant deviation (i.e. low turning).

In order to establish the exact source of the losses, the flow characteristics at higher setting angles must be determined. Limited experimental testing has been carried out on VIGV's, and interpretation of results is difficult due to the overall geometry of the IGV section. As well as experimental work, attempts have been made to model this geometry with a computational fluid dynamics (CFD) solver of the Navier-Stokes
equations. Software such as BTOB3D, created by Dawes\textsuperscript{16}, has been employed to model the flow and has provided a useful visualisation of the effects of the VIGV's.

The programme of research detailed in this thesis combines theoretical studies with experimental work to provide a sound basis for the design of improved VIGV's for industrial centrifugal compressors, particularly those used in this application. The study has used modern computational fluid dynamics techniques as well as conventional experimentation methods to develop a generalised technique for future development of efficient VIGV systems.
2.1 Variable Inlet Guide Vanes

2.1.1 Background

Competition among centrifugal compressor manufacturers is mainly dominated by the drive to attain the highest compressor efficiency, and to sustain this efficiency over a large stable flow range. Some advances in compressor efficiency can be achieved by improved component design, but the most appropriate solution to this requirement is the use of variable geometry devices.

Variable geometry devices include variable inlet guide vanes and variable-vaned diffusers, the two devices used either together or independently. Variable geometry impeller blades have been used successfully in hydraulic turbomachines, but this concept is considered impractical in compressor impellers due to their relatively high rotational speed and small size.

Centrifugal compressor VIGV's normally consist of a non-rotating annular cascade of vanes positioned upstream of the impeller inlet, which can be rotated about a radial axis using an external actuating mechanism. Radial guide vane cascades where the swirling flow is set up in a radial inlet plane and is then turned through 90° into the axial inlet duct have also been designed and tested. Shouman and Anderson\textsuperscript{[17]} tested such a system and found that its performance was similar to that of an axial cascade. Radial VIGV systems are however more complex to assemble and generally favoured only in gas turbine applications where a shorter axial machine length is desirable.
Chapter Two: State of Art

Figure 2.1 shows a typical centrifugal compressor along with an axial VIGV system and actuation assembly.

Although many of the studies referenced in this report are conducted on compressors with both variable inlet guide vanes and diffuser vanes, the research reported here will focus on the function and design of variable inlet guide vanes alone.

Fig. 2.1 Typical VIGV system

2.1.2 Function & Performance

For a centrifugal compressor with an axial inlet flow, the inlet whirl component ($V_{wl}$) is theoretically zero. If, however, inlet guide vanes are introduced upstream of the inducer, the magnitude and direction of $V_{wl}$ can be directly controlled. It can therefore be seen from the Euler equation (Equation 1.1) that the application of inlet swirl directly affects the value of the work done by the impeller on the fluid.

If the inlet guide vanes are positioned at a setting angle which induces swirl in the same direction as the rotation of the impeller this is known as ‘positive prewhirl’, and results in a decrease in energy rise through the compressor. Conversely, if the prewhirl opposes the impeller rotation, ‘negative prewhirl’, the energy rise is increased. The effect of inlet prewhirl is demonstrated in the inlet velocity triangles shown in Figure 2.2.

Fig. 2.2 Inlet whirl representation
A backswept impeller sometimes accompanies variable inlet guide vanes as this combination provides an increase in the work characteristic as shown in Figure 2.3. This figure can be derived by consideration of the velocity triangles and Euler equation. It can be seen from the outlet velocity triangle for a backswept impeller, Figure 2.4, that $V_{w2}$ increases as the radial velocity $V_{r2}$ (or mass flow) reduces. Also, increasing the positive inlet prewhirl as the mass flow reduces results in an increase in the inlet swirl velocity component ($V_w$), Figure 2.2. At constant rotational speed, it can therefore be seen that the net result of these flow characteristics is constant work.

In addition to the effect on the Euler equation, VIGV’s and a backswept impeller also result in a reduction in the inlet relative Mach number, and a reduction in the discharge absolute Mach number. The combined effect is a potential further increase in efficiency.
Early work on variable geometry gas turbine radial compressors was conducted by Rodgers\textsuperscript{18}. He assessed the effects of inlet prewhirl in terms of compressor efficiency ratio, maximum flow ratio, and a non-dimensional work factor. The results are shown in Figure 2.5 and clearly indicate the high mass flow range at which high efficiencies were obtainable. The trend shown by the work factor ($q$) can be derived directly from the Euler equation mentioned previously.

Fig. 2.5 Inlet prewhirl effects

Equally important to the aerodynamicist is the requirement to extend the stable flow range offered by the centrifugal compressor. The peak efficiency normally occurs at a mass flow close to the surge point of the compressor, this is because the maximum efficiency of aerodynamic blading normally occurs with a slight degree of flow separation (or incidence).

Consideration of all of the aerodynamic components of a centrifugal compressor is necessary when analysing the onset of surge. The parameters that directly affect the stability of the compressor have been summarised by Elder and Gill\textsuperscript{19}:

i) Gradient of compressor pressure rise characteristic and the limited aerodynamic performance range of each component - a narrower, steep characteristic will be more stable, but result in higher losses during off-design operation.

ii) Inducer incidence - if this exceeds a specific value, the inducer will stall.

iii) Impeller blade backsweep and inlet swirl - as discussed above.

iv) Number of diffuser and impeller vanes.
v) Pressure recovery within the semi-vaneless-space and incidence on the diffuser vane leading edge.

vi) Diffuser channel pressure recovery and collector type.

vii) Casing treatments - these can lead to an extended stable operating range.

Most of the parameters listed above can be affected by variable geometry, either variable inlet guide vanes or variable diffuser vanes. The implication of VIGV's with respect to surge margin can be explained by consideration of the first three parameters.

In addition to increasing the efficiency of the compressor, the VIGV's can also increase the surge margin, i.e. extend the stable operating range into lower mass flow rates.

The regulation of relative inlet flow angle ($\alpha_1$) using variable inlet swirl can be simply demonstrated by consideration of the inlet velocity triangle, Figure 2.6. Three different mass flow rates are indicated by the varying magnitudes of axial velocity, but the application of inlet swirl means that the relative inlet flow angle ($\alpha_1$) does not change. It can be seen that a small degree of negative prewhirl is required at high mass flow rates, with much larger positive prewhirl being employed at low flow rates.

---

Fig. 2.6 Control of relative inlet flow angle using inlet prewhirl
The configuration proposed by Swain and Connor\textsuperscript{[15]}, including a variable vaned diffuser, variable inlet guide vanes and a backswept impeller, has also been assessed by Rodgers\textsuperscript{[20]}. He investigated the effects of varying IGV setting angle, impeller speed and diffuser angle. An impeller with a 40° backsweep angle was used in the experiments, which yielded the results shown in Figure 2.7. This shows that the prewhirl moves the surge line to lower flow rates at high impeller speeds, but at lower speeds (and hence pressure ratios), the compressor exhibits little change in its operating characteristic. Results from his analyses indicated that, at high impeller speeds, inducer stall was responsible for the onset of surge. He also showed that regulation of the inlet flow using VIGV's allowed the compressor to operate with the diffuser stalled. The trends deduced from testing and analyses suggested that compressor surge was probably triggered by the diffuser at zero prewhirl conditions. Rodgers concluded from this study: “It provided a high degree of confidence that inducer incidence control through the use of VIGV's can provide significant performance improvements for this type of centrifugal compressor stage.”

**Fig. 2.7 Effect of prewhirl on surge line**

It can be seen from Figure 1.5 that the inducer incidence ($i$) varies as the magnitude of the mass flow and hence the axial velocity vector varies. The incidence can be positive at low flows and negative at higher flows, either way it leads to an increase in the loading of the leading edge of the blade and a decrease in the efficiency. The inducer is sensitive to incidence and rotating stall will develop if the incidence is too high. Variable inlet guide vanes have the additional effect of regulating this incidence.
Whitfield\textsuperscript{[21]} discussed the implications of inlet prewhirl both as a reduction in the inlet relative Mach number, and as a regulation of the incidence at the inducer tip. Minimising the inlet relative Mach number results in an extended flow range to the choked flow condition. In addition to this role, the guide vanes must also regulate the direction of the relative velocity vector in order to maintain the optimum incidence (typically approximately $6^\circ$). A prewhirl angle is therefore necessary to effect both of these requirements, thus resulting in a high overall efficiency and broad operating range. For this reason, Whitfield concluded that variable prewhirl may be desirable despite the additional cost and complexity.

Whitfield and Abdullah\textsuperscript{[22]} investigated the effect of inlet prewhirl on the inlet relative flow angle into the impeller. They commented that large swirl angles must be adopted for the inlet guide vane system to be effective at reducing inducer incidence thus suppressing surge in a low pressure ratio (2:1) machine. Consequently, their overall conclusion was that: "There is a need for a swirl generating device which will be efficient at both zero and high swirl conditions."

Whitfield\textsuperscript{[21]} proposed a non-dimensional design procedure including the effect of inlet prewhirl. He examined the effect of prewhirl on the inlet relative Mach number and absolute inlet Mach number. Minimising the inlet relative Mach number results in the reduction of passage friction loss and ensures adequate flow range to the choked flow condition. From the inlet velocity triangles (Figure 1.4), it can be seen that the inlet relative Mach number can be minimised for any fixed prewhirl angle by appropriate selection of the relative flow angle. This relationship can be applied as a design consideration by selection of the most appropriate impeller size and radius ratio and non-dimensional mass flow rate. As an alternative approach to the reduction of inlet relative Mach number, variable inlet prewhirl could be employed to regulate the value of the relative flow angle. As a result, the magnitude of the absolute Mach number will increase.
2.2 VIGV and Variable Vaned Diffuser Combination

Further increase in the performance and efficiency of centrifugal compressors is possible with the introduction of vaned diffusers rather than vaneless diffusers. The purpose of the diffuser is to collect the high kinetic energy flow from the outlet of the impeller and convert it into a static pressure rise. Diffusion through the vane passages of vaned diffusers is far more controlled and efficient than vaneless diffusers, although vaned diffusers offer a narrower operating range, beyond which the diffuser vanes can stall. Simon et al\cite{23} reported these differences using a centrifugal compressor incorporating backswept impeller and a vaned and vaneless diffuser.

Tests were initially carried out on a single stage compressor with an axial inlet and a vaneless diffuser, resulting in the characteristic shown in Figure 2.8. The vaneless diffuser was then replaced with a fixed vane diffuser (vane angle = 64° to radial), Figure 2.9. Comparison of Figure 2.8 and Figure 2.9 clearly shows the increase in efficiency offered by the fixed vane diffuser, along with the corresponding decrease in operating range.

Fig. 2.8 Vaneless diffuser performance

Fig. 2.9 Vaned diffuser performance
Similar experiments with refrigerant R-134a as the working fluid rather than air were conducted by Harada\cite{24}. A closed-loop test stand was used, incorporating a centrifugal compressor with a backswept impeller and variable inlet and diffuser vanes. Figure 2.10 shows the performance obtained with a fixed vane diffuser and variable inlet guide vanes.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{fig2_10}
\caption{Fixed vane diffuser & VIGV performance}
\end{figure}

It is clear that the surge point is moved to lower flow rates by regulation of the guide vanes, although the overall operating range remains very narrow. This was probably due to the large values of incidence onto the diffuser vanes that eventually triggered gross separation and surge.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{fig2_11}
\caption{Variable vaned diffuser & VIGV performance}
\end{figure}

The introduction of variable diffuser vanes substantially altered the performance characteristic (Figure 2.11). It was possible to operate the compressor well below the flow rates at which surge occurred when the fixed vane diffuser was used. The optimum diffuser vane angle was calculated by measuring the static pressure difference between the suction and pressure surfaces of the vanes. The vane angle was adjusted so that this pressure difference remained below a defined threshold value.
The results obtained by Harada proved that the use of a variable vaned diffuser in conjunction with variable inlet guide vanes allowed the compressor to operate at flow rates almost as low as shut-off flow without any apparent surge.

Inlet guide vanes have also often been successfully used at the first stage of multistage axial compressors. Venkatrayulu et al\textsuperscript{[25]} reported an increase of 35 percent in the useful operating range of a single stage axial flow fan when using inlet guide vanes.

In addition to the application of inlet guide vanes to high pressure ratio centrifugal compressors for process industry and turbocharger applications, relatively low pressure ratio compressors, or blowers, can use the variable geometry technology. Investigations into the effect of VIGV's on a radial single-stage blower were presented by Kryllowicz et al\textsuperscript{[26]}. Their work focused on the potential of VIGV's to extend the stable flow range by moving the surge limit towards lower mass flow rates. The performance characteristic shown in Figure 2.12 clearly shows the turndown in stable flow as a result of IGV regulation.

\begin{figure}
\centering
\includegraphics[width=\textwidth]{fig212.jpg}
\caption{Turndown in flow achievable through IGV regulation}
\end{figure}
They also observed a region of flow reversal (inlet recirculation) when operating near the deep surge region at IGV setting angles between $45^\circ$ and $65^\circ$. This results in a higher level of noise and a decrease in efficiency, although the blower can continue to operate in a stable manner. Figure 2.13 shows the radial distribution of inlet velocity for the particular IGV geometry tested here.

![Fig. 2.13 Radial distribution of meridional ($c_m$) velocity and tangential velocity ($c_n$)](image)

The velocity profiles presented here are not only related to the vane shape and angle, but also the shape of the hub and shroud within which the vanes are housed. For this reason, the aerodynamic design of the geometry of the inlet guide vanes and their housing must be carefully considered as well as the aerodynamic components of the compressor stage itself.
2.3 Aerodynamic Design

The objective of inlet guide vanes is to introduce swirl upstream of the compressor impeller over a range of vane angles and flow rates without incurring high losses. The designer is not only faced with the challenge of developing an aerodynamically acceptable geometry, but also one which can be easily assembled and must withstand the substantial loads which will be exerted on the vanes when positioned at high setting angles. In addition to the design of the vanes, the shape and configuration of the ducting must be considered.

2.3.1 Inlet Ducting Design

Careful consideration in the design of the inlet ducting can be as aerodynamically important as the design of the blades themselves. More primitive inlet guide vane models consist simply of an annular cascade of vanes anchored through holes around the circumference of the compressor inlet pipe. An example of this configuration is shown in Figure 2.14.

Fig. 2.14 Simple VIGV arrangement
Although this is a relatively simple and cheap configuration to manufacture, its aerodynamic performance is substantially impaired by a number of features:

i) At high stagger angles a significant leakage area exists at the centre of the inlet ducting. This allows an axial jet to pass into the impeller as a core flow within an outer swirling flow.

ii) At mid stagger angles (30° to 60°) a clearance area will open up at the outer edge of the blades. This clearance results in an increase in losses and a distorted boundary layer growth in the outer flow region.

iii) At high stagger angles, a significant pressure drop will occur across the vanes. Since the blades are cantilevered about their rotational axis at their outer edges, this results in a considerable bending moment on the blade spindles.

A slight improvement on this design is to place the vanes in an almost spherical section of ducting as in Figure 2.14. This has the advantage of minimising the blade tip clearance when the blades are positioned at high stagger angles. The losses associated with the inner edges of the blades are still present however.

Fig. 2.14 Spherical shroud VIGV ducting
In order to reduce the losses associated with the central core of axial flow, a hub can be incorporated into the design, as in Figure 2.15. The hub is normally supported by radial struts in front of the leading edge of the guide vanes. In this position, minimal disturbance to the flow is incurred by the struts as no prewhirl has yet been imparted by the guide vanes. Figure 2.15 shows the simplest form of hub - a straight cylindrical section, which eliminates the axial jet flow described earlier.

Blade clearance problems similar to those described previously will occur at the hub as well as the shroud when the vanes are set at large stagger angles.

If the axial length of the hub does not extend as far as the impeller an inlet vortex will form (Figure 2.16). Chen et al.\textsuperscript{[27]} investigated the nature of such a vortex and described it as ‘...a central core of axially stagnant fluid. This core may also precess and then becomes a whirling helical rope’. The vortex can lead to strong pressure pulsations at the impeller, which in turn result in severe noise and vibration.

**Fig. 2.15 Cylindrical hub VIGV ducting**

**Fig. 2.16 Inlet vortex formation**
Such a vortex may be avoided by the extension of the hub as far as the inducer of the compressor. An example of this type of geometry is the inlet guide vane section tested by Kryllowicz et al.\textsuperscript{[26]. Figure 2.17 shows the hub with its radial support struts immediately upstream and downstream of the blade row. The upstream end of the hub is aerodynamically profiled to reduce losses at inlet to the IGV’s.

![Fig. 2.17 Profiled hub VIGV system](image)

Lakshminarayana and Sitaram\textsuperscript{[28]} described a similar 3-dimensional secondary vorticity at the tip region of IGV’s, resulting in considerable flow deviation from the inviscid flow.

Ideally, the IGV ducting will be designed with a spherical section hub and shroud. Minimal blade hub and tip clearances will result from this geometry at any blade stagger angle, although the hub and shroud could be more expensive to manufacture than simpler models.
If the hub and shroud are designed to be spherical sections, the passage through the IGV section must converge from the radius at the vanes to the radius of the impeller. Swain\cite{29} proposed basic rules that govern the design of the inlet ducting geometry:

- The inlet section should meet the hub and shroud at the impeller leading edge tangentially in the meridional view.
- The shroud profile between the VIGV and the impeller leading edge should be formed using two sweeps of equal radii meeting tangentially. The centre of the radius furthest from the leading edge should lie on the impeller axis of rotation and it, therefore, represents the sphere forming the outer profile of the vane section. The value of this radius determines the distance of the vane rotation axis from the leading edge.
- The hub profile between the vane section sphere and the impeller leading edge should be formed from a single radius and meet these surfaces tangentially in the meridional plane.
- The passage upstream of the vane should be straight (conical) sections formed tangentially to the vane leading edge position at zero stagger.

Figure 2.18 shows the basic parametric geometry. A range of three ducting geometries that evolved from these design rules were analysed by Swain. An older existing design, 'b1', which was not designed in accordance with the rules, was also modelled. This swan-neck design ensures that the walls of the passage are smooth so that the flow field is well developed with no disturbances to the wall boundary layer on entry into the impeller.

**Fig. 2.18 VIGV ducting parametric geometry**
The basic dimensions of the different geometries are summarised in Table 2.1, the area ratio being defined as follows:

\[ \text{area ratio} = \frac{\text{annulus area at the VIGV centre of rotation}}{\text{annulus area at impeller inlet}} \]

<table>
<thead>
<tr>
<th>geometry variant</th>
<th>improved</th>
<th>original</th>
</tr>
</thead>
<tbody>
<tr>
<td>c6</td>
<td>c12</td>
<td>c17</td>
</tr>
<tr>
<td>area ratio</td>
<td>2.3</td>
<td>2.3</td>
</tr>
<tr>
<td>shroud radius R (mm)</td>
<td>178.6</td>
<td>197.9</td>
</tr>
<tr>
<td>hub radius Rh (mm)</td>
<td>73.9</td>
<td>112.8</td>
</tr>
<tr>
<td>vane centreline to impeller leading edge</td>
<td>206.6</td>
<td>245.2</td>
</tr>
</tbody>
</table>

Table 2.1 Geometry dimensions

Each of the geometries was analysed using the Dawes\textsuperscript{16} CFD code, BTOB3D. With the blades positioned at a stagger angle of 30°, the average swirl at impeller inlet was measured along with the calculation of a non-dimensional pressure loss coefficient defined as:

\[ \frac{P_{01} - P_{02}}{P_{02} - P_2} \]

where \( P_{01} \) and \( P_{02} \) are the stagnation pressures at inlet and exit to the guide vane section respectively, and \( P_2 \) is the static pressure at exit.

The trend in the average swirl values given in Table 2.2 can be attributed to a combination of area ratios and radius ratios, i.e. the ratio of the mean radius at the VIGV's to that at the impeller inlet. Larger radius ratios lead to an increase in swirl due to the effect of conservation of angular momentum as the flow approaches the impeller inlet. However, larger area ratios have the effect of accelerating the axial component of the flow, thus 'straightening out' the flow at the impeller inlet and reducing the magnitude of the swirl angle.
Table 2.2 Predicted ducting performance

<table>
<thead>
<tr>
<th>Geometry variant</th>
<th>improved</th>
<th></th>
<th>original</th>
</tr>
</thead>
<tbody>
<tr>
<td>β average swirl (°)</td>
<td>c6 29</td>
<td>c12 41</td>
<td>c17 30</td>
</tr>
<tr>
<td>area ratio</td>
<td>2.3</td>
<td>2.3</td>
<td>3.3</td>
</tr>
<tr>
<td>radius ratio</td>
<td>1.71</td>
<td>2.1</td>
<td>2.29</td>
</tr>
<tr>
<td>radius ratio / area ratio</td>
<td>0.74</td>
<td>0.91</td>
<td>0.69</td>
</tr>
<tr>
<td>β' estimated swirl (°)</td>
<td>23</td>
<td>28</td>
<td>22</td>
</tr>
</tbody>
</table>

Figure 2.19 shows that a significant improvement in the pressure loss characteristics is exhibited by the new designs. If the pressure loss is interpreted as an equivalent compressor overall efficiency loss, Figure 2.20, it can be seen that at the design flow rate, the losses associated with the new designs are negligible, whereas the old design resulted in a loss of at least 2 percent.

It should be borne in mind that the losses discussed here are not all caused by poor ducting design - a significant percentage of the losses is attributed to the aerodynamic effects of the blades.
2.3.2 Blade Design

Industrial centrifugal compressor variable inlet guide vanes are faced with a much greater challenge than that of, say, gas-turbine IGV's. The gas-turbine IGV is required to operate with minimal loss and relatively low deflection for a small range of incidence, whereas centrifugal compressor flat plate IGV's have to accommodate a large range of incidence and produce large deflections. Since the compressor will be designed to operate at peak efficiency at its design duty, the IGV's must also allow a purely axial flow to enter the impeller with minimal disturbance to the flow. The requirements of centrifugal compressor IGV blades can be summarised as follows:

- They must be able to produce a large range of positive and negative deflection.
- Minimal disturbance or loss to the flow must be caused when they are set in their axial (0° stagger) position.
- Significant pressure losses must be avoided as a result of flow separation or blade stalling when the blades are set at high stagger angles.
- At high stagger angles the blades must be able to withstand the forces exerted on them by the pressure differential and flow rate across them.

The simplest approach to partially satisfying these requirements is to use untwisted, uncambered flat plate blades. Unfortunately, the performance of such blading rapidly deteriorates as the stagger angle is increased from the axial position, as proven by Swain[29]. Typically, centrifugal compressor IGV's can be required to operate at stagger angles of between -20° and +80°, therefore if flat plate vanes are used they are presented with an incidence range of 100°. Increasing the stagger angle results in large pockets of separation on the suction surface of the IGV's as shown in the flow visualisation in Figure 2.21. The result of this separation is an increase in the non-dimensional pressure loss coefficient, which is depicted in Figure 2.22. As the stagger angle is increased, the size of the separation pocket will increase along with the magnitude of flow deviation at the trailing edge of the blade. The poor performance of such blading can be highlighted, as in Figure 2.23, if the pressure losses are interpreted as an equivalent overall stage efficiency loss.
In addition to its magnitude, the type of prewhirl to be induced must also be selected. This can take the form of free or forced vortex, or constant swirl velocity with radius. These prewhirl vortices can be described mathematically with the following relationship:

\[ V_w = A \left( \frac{r}{r_s} \right)^n \]

where: \( A = \text{constant} \)
\( r = \text{radius} \)
\( r_s = \text{shroud radius} \)

and:
\( n = -1 \) for free vortex (swirl \( \propto 1/radius \))
\( n = 0 \) for constant swirl velocity
\( n = +1 \) for forced vortex (swirl \( \propto \text{radius} \))
The aim of applying a vortex pattern at the inlet guide vane section is to reduce the magnitude of incidence across the span of a specific inducer. Whitfield\textsuperscript{[30]} derived the incidence distribution at an inducer for each vortex type, Figure 2.24. He demonstrated that the introduction of free vortex prewhirl leads to a significant increase in incidence angle at low inducer radius ratios. For zero incidence at the shroud radius a free vortex yields approximately 20° of incidence at a radius ratio of 0.6. It can also be seen that the use of forced vortex yields a more satisfactory incidence distribution. If the forced vortex model is extended to that of a quadratic prewhirl distribution ($n = +2$), as demonstrated by Steinke and Crouse\textsuperscript{[31]}, the incidence distribution is improved even further.

![Incidence at inducer for different vortex patterns](image)

**Fig. 2.24** Incidence at inducer for different vortex patterns
In order to create the incidence distribution at the inducer, it is necessary to design a twisted blade in the cases of free and forced vortex. The flow angle distribution required at the IGV is shown in Figure 2.25 for an assumed swirl angle of 30° at the shroud. If it is assumed that the flow angles correspond directly to the blade angles (i.e. constant distribution of blade deviation for all radius ratios), then these curves can be interpreted as the range of twist required along the span of the blade. Clearly, a high degree of twist between hub and shroud would be required for the free vortex or quadratic distribution.

Fig. 2.25 Swirl angle distribution at IGV

In addition to the requirement of a satisfactory incidence distribution, the practicalities and construction of the blades to provide it must be considered. Although the quadratic prewhirl yields a desirable incidence distribution, the twisted blade profile would be complex and expensive to manufacture. Also, the necessity of undisturbed flow with the vanes set at the 0° (axial) stagger position must be considered. For these reasons, most industrial VIGV systems employ untwisted vanes.

Despite the suggestions made by Whitfield to regulate the inducer incidence by controlling the vortex pattern developed by the VIGV’s, the incidence can also be minimised by leaning the leading edge of the inducer vane.
Rodgers\textsuperscript{[32]} demonstrated that VIGV's positioned in a radial inlet plane yield a flow angle distribution similar to that of free-vortex conditions, Figure 2.26. The resulting radial variation in axial velocity causes a relatively large reduction in the inducer incidence near the hub. Rodgers also discovered that the inlet total pressure losses for all three VIGV settings were almost equal, thus concluding that cambered vanes would not greatly improve the overall stage efficiency.

Fig. 2.26 Flow angle distribution for radial VIGV's
In their tests on a radial flow turbocharger compressor, Wallace et al.\cite{33} used two designs of untwisted vanes. One was a standard flat plate vane and the other a curved vane with a camber angle of 40°. When the curved vanes were positioned at 0° incidence a deflection of 35° of positive prewhirl was generated. A direct comparison of the compressor performance characteristic with the two different vane types positioned to provide +20° of prewhirl is given in Figure 2.27. An explanation was not offered. A likely explanation of the increased efficiency and pressure ratio exhibited by the curved vanes is that the magnitude of incidence and separation was lower, and resulted in a decrease in the inlet pressure loss.

![Fig. 2.27 Flat and 35° curved vane](image)

A similar study was conducted by Whitfield et al.\cite{34,35}. They used two different types of vanes: a flat plate vane and a curved vane giving a discharge blade angle of 20° when set to give zero incidence at inlet. The results of stage testing with the two blade designs (set to give +20° of prewhirl) are compared in Figure 2.28, and these also demonstrated the improved performance of the curved vanes over flat plate vanes. They suggested that the increase in pressure ratio using the curved vanes occurred as a

![Fig. 2.28 Flat and 20° curved vane](image)
result of the increase in stage efficiency (due to less loss and turbulence at the impeller inlet), combined with a decrease in swirl from the curved vanes.

Despite the increased efficiency of the cambered blades at higher setting angles discussed here, due to their shape they have only permitted a limited range of turning and have not permitted axial flow without significant losses. If the guide vane design is to be industrially feasible, it must satisfy all of the requirements detailed earlier in this chapter. A solution to a similar design specification was presented by Sanz et al.\cite{36} when designing a guide vane cascade for use in a wind tunnel cascade that would operate over an inlet flow angle range of 60°. The vanes had to be capable of receiving flow from either of two legs of the wind tunnel with an angle of 45° between them. The guide vane system had to achieve acceptable losses with no moving parts. Two cambered blade designs were considered, Figure 2.29, and scale models were built and tested in a smaller wind tunnel.

![Cambered blade designs](image)

Fig. 2.29 Cambered blade designs
Pressure surveys on both sides of the blades were conducted with the vanes set at air inlet angles of 0° and 45°. The averaged results of the three centre blades of the tested cascade are presented in Figure 2.30 (0°) and Figure 2.31 (45°) for the Lewis No. 3 design. The measured losses in total pressure (expressed as percent of inlet dynamic head) were 17 percent or less over a range of inlet air angles from -5° to +55° while the exit air angles remained within 1.5° of axial.

Fig. 2.30 Performance at 0° inlet air angle

Fig. 2.31 Performance at 45° inlet air angle
If the same vane shape was used in an annular cascade and pivoted about a point near its leading edge, it could possibly be used as a VIGV capable of generating a prewhirl within the range of -5° to +55°.

Serovy and Kavanagh\[37\] detailed some of the considerations that must be made by the designer when designing variable geometry blading for axial flow compressors. Although axial compressor IGV's are required to operate over smaller incidence ranges than centrifugal compressor IGV's, the basic principle of their operation and design constraints remain the same. They proposed two important points relating to the design of such blading:

- It may be that any variable geometry scheme that is really valuable in maintaining efficient compressor performance will be quite complex and that simple rotatable guide vanes will be inadequate.
- The aerodynamic design of variable geometry systems will have to be based on a detailed knowledge of blade-section turning and loss characteristics.

The first point made by Serovy and Kavanagh seems to have been an extremely valid point as many efforts have been made to design and test alternative forms of blading, such as tandem blades. These can consist of two consecutive blade rows in very close proximity to one another; the leading row remains fixed while the trailing row will pivot about its leading edge.

Okiishi et al\[38\] presented a comprehensive study of the performance of one of the simplest forms of uncambered tandem vanes, Figure 2.32. The leading 30 percent of the vane was fixed whilst the trailing 70 percent moved as a trailing edge flap.
Chapter Two: State of Art

An annular cascade of 13 vanes was tested in a wind tunnel at Mach numbers between 0.13 and 0.38. Downstream total pressure and flow angle measurements were taken at 10, 30, 50, 70, and 90 percent of the passage height (between hub and shroud). The results of the air outlet angles at blade stagger angles between 5° and 50° are given in Figure 2.33. Also shown in this figure are predicted outlet angles for a circular and parabolic arc camber line blade using Carter’s rule (see Nomenclature).
Fig. 2.33 Outlet air angle distribution

It can be seen that the radial distribution of flow angles is fairly uniform for blade angles up to 25°, the distribution also compares reasonably well with the prediction for the circular arc camber line blade. As the blade angle increases, the flow angle distribution distorts, particularly near the hub. This can be explained by consideration of the blade hub and tip clearances: as the blade angle increases, the tip clearance will decrease and the hub clearance will increase, allowing more axial leakage of flow past the blade.
Figure 2.34 shows the variation of outlet air angle with blade stagger angle at the mid-span (50 percent passage height) position. Again, a strong similarity with the Carter's rule prediction for the circular arc camber line blade is demonstrated by the results.

Despite the linearity observed for the variation of air angle with blade stagger angle, the total pressure loss coefficient follows a significantly different trend, Figure 2.35. A marked increase in loss is observed for flap angles greater than approximately 35°. When compared with the results offered by Sanz et al[36] for cambered vanes, the relative merits of each blade type can be seen. Whilst the total pressure loss coefficients at low stagger angles for uncambered tandem vanes are lower than those of cambered vanes, the inverse applies for higher vane angles (above 35°).
Beelte and Oppermann\cite{39} presented the findings of a similar study into the benefits of tandem vanes over a single row of uncambered flat plate vanes using the configuration shown in Figure 2.36. They reported a noticeable decrease in loss coefficient across the vanes along with an increase in deflection when using the tandem vanes.

Fig. 2.36 Tandem vane configuration

Cyrus\cite{40} reported on the performance of an axial compressor with tandem inlet guide vanes based on the C4 airfoil design. The guide vanes were required to operate over a range of 80°, however above 20° stagger angle the flow began to separate and at 50° pressure pulsations appeared due to the onset of stall.

Early work in this field was conducted by Linder and Jones\cite{41}, who tested three different guide vane cascades for use in axial compressors, Figure 2.37. The first design was a standard cambered blade similar to that used in tests already detailed. The second was a variable camber tandem vane, and the third a more elaborate concept called 'the venetian blind concept'.

The performance of the blades was evaluated at a turning position of 30° and at the axial flow position using wind tunnel techniques. The tests revealed that although the cambered blade delivered approximately 30° of turning with low loss when positioned at 30° stagger,
the losses and wake generation were substantially increased when it was operated in the axial position. Similarly, the venetian blind vanes revealed a loss coefficient of 0.069 in the axial position compared with 0.021 for the variable camber vanes. In addition, the variable camber vane also developed approximately 30° of turning when positioned at 30° stagger with relatively low loss.

More recently, a similar cambered tandem blade based on a concept known as a controlled diffusion airfoil (CDA) was developed and tested by Saha and Roy\textsuperscript{[42]}. The original CDA cascade consisted of a single row of cambered vanes called ‘CDA 43’. The formation of a separation pocket was observed on the suction surface of this blade. As described earlier, this phenomenon leads to high losses and inefficient cascade operation. Testing of the new tandem cascade design, ‘CDA 32-21’ shown in Figure 2.38, did not reveal such separation.

Although the tandem cascade demonstrated the ability to maintain an attached flow over a large portion of its blade surface and provide higher deflection, the resulting total pressure loss coefficient was slightly higher than that of the single blade row at higher stagger angles.
2.4 Computational Fluid Dynamics

All (internal and external) fluid flows are governed by a set of mathematical laws that can be written in the form of a set of equations known as the Navier-Stokes equations. These equations are derived from the physical laws of conservation of mass, energy, and momentum. During the 1960's and 70's, computerised techniques were developed to solve these equations for an entire flow field. Major progress has subsequently been made in technique development and other fields associated with the evolution of more accurate and efficient solvers.

Computational Fluid Dynamics (CFD) is now a powerful and widely available design tool which allows the aerodynamicist to examine complex flow fields with less dependency on experimental facilities. Virtually any conceivable geometry can be modelled, often using standard CAD (Computer Aided Design) packages linked to the CFD application. The geometry is subdivided into a large number of cells or control volumes. In each of these cells, the Navier-Stokes partial differential equations can be rewritten as algebraic equations that relate the velocity, temperature and pressure, for example, in that cell to those in all of its immediate neighbours. The resulting set of equations can then be solved iteratively until convergence is reached, yielding a complete description of the flow throughout the domain. All CFD codes are split into three main elements: (i) the pre-processor, (ii) the solver, and (iii) the post-processor.
The complete procedure for modelling a flow field is summarised by Versteeg and Malalasekera\cite{43} as follows:

i) Pre-processor
- Definition of the geometry of the region of interest: the computational domain.
- Grid generation - the sub-division of the domain into a number of smaller, non-overlapping sub-domains: a grid (or mesh) of cells (or control volumes or elements).
- Selection of the physical and chemical phenomena that need to be modelled.
- Definition of fluid properties.
- Specification of appropriate boundary conditions at cells which coincide with or touch the domain boundary.

ii) Solver
- Approximation of the unknown flow variables by means of simple functions.
- Discretisation by substitution of the approximations into the governing flow equations and subsequent mathematical manipulations.
- Solution of the algebraic equations.

iii) Post-processor
This offers visualisation of the results of the solver such as:-
- Domain geometry and grid display.
- Vector plots.
- Line and shaded contour plots.
- 2D and 3D surface plots.
- Particle tracking.
- View manipulation (translation, rotation, scaling etc.)
The use of CFD by turbomachinery manufacturers has increased significantly over the past 15 years, resulting in a shorter hardware development cycle. Combined with measurements, CFD provides a complementary tool for simulation, design, optimisation, and, most importantly, analysis of three-dimensional flows otherwise inaccessible to the engineer. Detailed measurement using standard instrumentation in rotating passages such as those in centrifugal impellers is expensive, and sometimes impossible. In these cases, CFD provides the only means of testing.

A comprehensive description and comparison of modern CFD techniques with specific reference to turbomachinery flows is offered by Lakshminarayana. He outlined the requirements for an accurate flow field solution as follows:

i) Governing equations, including turbulence transport equations; validity of approximations made.
ii) Enforcement of proper boundary conditions.
iii) Adequate grid resolution.
iv) Turbulence modelling.
v) Computer architecture, including parallel processing.
vi) Assessment of computational techniques through calibration and validation.

Numerous validations of three-dimensional viscous Navier-Stokes codes in the analysis of turbomachinery flows have been conducted and presented in recent years. The CFD analysis of the radial inlet of a centrifugal compressor presented by Flathers et al. shows particularly well how the requirements outlined above contribute to an accurate flow-field prediction.
The use of CFD in the design of inlet guide vane systems is based upon the well-proven application of 2-D and 3-D cascade performance prediction. Comprehensive descriptions of benchmark testing of both linear and annular non-rotating axial turbine cascades have been offered by Raw et al\cite{46}, and Niestroj and Came\cite{47}. One of the analyses conducted by Niestroj and Came\cite{47} was on the linear turbine cascade shown in the mesh cross-sectional view in Figure 2.39, for which well documented results of experimental investigations have been published, Gregory-Smith\cite{48}. Examples of validation of the CFD analysis are shown in Figures 2.40 and 2.41 where strong agreement between numerical and experimental results can be seen.

Fig. 2.39 Cross-sectional mesh of turbine blade (leading edge)

Fig. 2.40 Pitchwise averaged flow angles

Fig. 2.41 Static pressure distribution around blade
Porter\textsuperscript{[49]} used the Dawes\textsuperscript{[16]} code, BTOB3D, to design a centrifugal compressor inlet guide vane system incorporating cambered vanes. Apart from a small degree of separation at the vane leading edge, the vanes operated with very little separation at setting angles up to 50° (Figure 2.42). At 0° (Figure 2.43) the flow acceleration and deceleration caused by the vane leading edge blockage suggests that the losses would be particularly high in comparison with flat plate vanes.

![IGV Performance from Dawes](image)

**Fig. 2.42 Velocity vectors at setting angle = 40°**
Swain\[^{50}\] demonstrated a method of relative comparison of theoretical designs. He used the Dawes\[^{16}\] code, BTOB3D, to compare the performance of the improved 'c12' design and the old 'b1' shown in Figures 2.44 and 2.45 respectively.
Figure 2.46 shows the velocity vectors near the shroud of the ‘b1’ system, where the effect of the vane tip clearance clearly results in very little flow turning downstream of the vanes. Considerable flow separation is also evident from the vane leading edge.

The reduced tip clearance of the ‘c12’ system results in significantly higher deflection, Figure 2.47. A small degree of flow separation can be seen at the vane leading edge, but the flow re-attaches resulting in negligible deviation.
Gross separation occurs near the hub of the ‘b1’ design, Figure 2.48, which leads to flow recirculation around the vane and virtually no turning. Although the velocity vectors near the hub of the ‘c12’ design, Figure 2.49, reveal a pocket of separation on the suction surface of the blades, the downstream flow field is far less turbulent than that displayed for the old design, and again results in an increase in turning.

As well as the stationary frame of reference models that have been discussed so far, CFD is particularly invaluable to the turbomachinery designer for analysing the flow within rotating passages. Validation of such modelling has been presented in many case studies including an automotive engine cooling fan system (Coggiola and Broberg\textsuperscript{51}), and for a centrifugal compressor (Shah and Bartos\textsuperscript{52}). A more appropriate study to the research being conducted in this project was that of a
centrifugal compressor to be used in a water-treatment application. Pitkänen\textsuperscript{[53]} used CFD to design a compressor stage incorporating a backswept impeller with a vaneless diffuser and no VIGV’s. The volute was not modelled. The compressor was then manufactured and total-to-total pressure ratio and isentropic efficiency was experimentally measured for correlation with the performance predictions. The comparisons of pressure ratio and efficiency are shown in Figures 2.50 and 2.51 respectively. The performance was predicted reasonably well on the whole, and the author attributed any discrepancies in the results to his assumption of constant pressure recovery factor through the volute at varying operating conditions.

With the advent of more powerful and less costly computer processors (and a decrease in the cost of RAM), increasingly complex CFD analyses are now feasible. Recent advances in the programming of CFD solvers and pre-processors mean that entire compressor stages can now be accurately modelled. One recent advance has been the development of a numerical procedure to accommodate the interface between rotating and stationary components (such as stators and rotors). An examination of such an interface was conducted by Elmendorf et al\textsuperscript{[54]} as part of a CFD analysis of a 15-stage axial compressor.
In addition to the accuracy and versatility of the CFD solver, the ease of use and applicability to the particular study in question should also be considered. Due to their high rotational speed and complex geometries, turbomachinery flows are generally far more complex to model than others encountered in fluid dynamic practice. The flows are normally three-dimensional, and are often turbulent and separated, and in an impeller the flow is always unsteady. As a result, accurate turbomachinery flow field predictions require CFD packages that are created with this application in mind.

Two commercially available CFD solvers were evaluated for use in this study, however only one package offered the robustness and versatility to solve such flows. CFX-TASCflow and its associated pre-processing software, CFX-Turbogrid and CFX-TASCbob3d was developed specifically for turbomachinery analysis, Raw et al.\textsuperscript{46}. Also, numerous proven case studies using the software to solve complex turbomachinery flows were available in published literature. Consequently, CFX TASCflow was selected as the CFD solver to be used in this study.

2.5 Conclusion

The research discussed in this chapter has shown that a need exists to improve the understanding and the technology of industrial centrifugal compressor variable inlet guide vanes. The existing designs of VIGV’s and current theory will provide a basis for the development of an improved geometry using both experimental and theoretical research techniques.

Initially, the research aims to provide the theoretical tools to predict the behaviour of a VIGV system suitable for an industrial compressor using the existing uncambered vane form. Subsequently, the aim will be to develop an improved form of vane that will maintain low loss and high deflection over a wide range of mass flow rates, thereby giving a significantly enhanced performance. A summary of the specific hypotheses of the research is as follows:
• The performance modelling of VIGV's suitable for industrial centrifugal compressors can be improved using a combination of experimental and numerical techniques.

• The relative benefits in stage performance of improved VIGV profiles can be determined by aerodynamic examination of the VIGV system alone.

• The overall stage performance of industrial centrifugal compressors requiring a large range of mass flow can be improved using a VIGV system capable of high-swirl with low-loss.

The outcome of this research will be a usable technique for real industrial compressor environments, and confirmation that an acceptable design can be achieved that represents a rewarding improvement in performance.
3.1 Centrifugal Compressor Stage and VIGV System

The compressor stage configuration is based upon the design proposed by Swain and Connor[15] and is shown in Figure 3.1. Thirteen untwisted flat plate variable inlet guide vanes were positioned in an axial plane immediately upstream of a converging ‘swan-neck’ inlet passage. The impeller was fully machined with approximately 40° of backsweep and has 9 main and 9 splitter vanes. A constant rotational speed of 18050 rpm corresponding to a maximum total-to-total pressure ratio of approximately 2:1 was selected for both experimental and numerical performance analyses.

Sixteen variable diffuser vanes with flat and circular arc surfaces were automatically adjusted to maintain optimum efficiency and maximise the stable operating range.

Fig. 3.1 Centrifugal compressor stage arrangement
3.2 Test Rig Design

In order to validate and gain confidence in the numerical study, experimental analysis of the performance of the VIGV system was essential. Ideally, the VIGV system should be tested in its typical operational mode, i.e. at the inlet to the centrifugal compressor stage. However, the close proximity of any intrusive instrumentation to the impeller and the stage power requirements would make this option impractical with the available resources. Instead, it was considered far more desirable to design and build a dedicated experimental facility that would avoid such problems.

The requirements of the test rig were simply to draw a known mass flow rate of air (up to a target of 1kg/s) at known conditions through an instrumented VIGV system. These requirements were to be met using minimal available power supplies and within a minimal laboratory floor area. It was considered that the ideal rig design would incorporate a fan/compressor to draw the air through the rig, followed by suction ducting to connect to the VIGV system, with some form of calibrated inlet to the VIGV system.

The first conceptual design for the test rig is shown in Figure 3.2, however it was soon discovered that this was not the ideal rig design for a number of reasons.

![Fig. 3.2 Proposed experimental facility design](image-url)
1) A mass flow measurement device was required at the inlet to the test rig. This would necessitate a straight length of ducting before the diffusing section was reached, resulting in an excessive overall test rig length. An alternative to this was to use an ‘upstream and downstream large’ measurement device which would mean that a plenum chamber could be used. The advantage of this is that the VIGV system could be directly mounted onto it just as it would normally be mounted directly onto a filter chamber.

2) Although the viewing section would allow visualisation of the flow characteristics downstream of the VIGV system, very limited flow visualisation equipment was available so it was felt that the additional manufacturing expense would be unjustifiable.

3) The air mass flow rate through the rig could be better controlled using a variable speed compressor rather than a butterfly valve at inlet to the compressor.

4) Predicted pressure losses across the mass flow measurement device and through the VIGV system when set at high stagger angles suggested that an axial fan would not be ideal to draw the required air flow rate through the rig.

As a result of these issues, the test rig shown in Figure 3.3 was designed and built to meet the requirements highlighted earlier.

![Fig. 3.3 Plan view of experimental facility](image-url)
3.3 Test Rig Components

3.3.1 Plenum Chamber

The inlet to the test rig is a 1.5m cubic plenum chamber that houses the inlet mass flow measurement device. The minimum dimensions of the chamber were dictated by BS 1042\(^{[55]}\) as the mass flow measurement device was ‘upstream and downstream large’. Although the function of the chamber was to stagnate the high velocity inflowing air, initial instrumentation testing showed that an unsteady flow regime was present within the VIGV system passage. The 2-D CFD streamlines (treated as an axisymmetrical model) shown in Figure 3.4a suggested that this unsteadiness was possibly due to the inflowing air following a jet-like flow pattern through the plenum chamber and impinging on the VIGV hub. This in turn results in turbulence and significant velocity components orthogonal to the axis of the VIGV system. Further modelling (Figure 3.4b) demonstrated that the flow regime could be improved by positioning a disc with a central hole located just downstream of the mass flow measurement device and co-axial with the VIGV section. This disc ensures that the inflowing air stagnates inside the chamber rather than flowing straight through it. The benefits of the disc are demonstrated by the resulting decrease in turbulence and hence pressure pulsations within the VIGV passage, Figure 3.5.
3.3.2 Variable Inlet Guide Vane System

The VIGV test section shown in Figure 3.6 was mounted directly onto the downstream side of the plenum chamber. Thirteen untwisted flat plate vanes can be manually adjusted to give setting angles ranging from -20° to +90°. A spherical section hub and shroud ensures minimal hub and tip blade clearance at any setting angle. Since the VIGV system was not tested directly in front of the centrifugal compressor, the inlet passage of the compressor needed to be represented by an addition to the guide vane passage. An adapter was manufactured and profiled internally to match the compressor inlet geometry. This allows the flow field to be examined at the axial plane in which the leading edge of the centrifugal impeller would normally coincide. The vanes were positioned manually to an accuracy of ±1° using the external actuating mechanism shown in Figure 3.6.

Fig. 3.6 Variable inlet guide vane system (viewed from upstream direction)
3.3.3 Compressor

The VIGV system was connected to the compressors (blowers) with a 2m straight length of ducting mounted on its downstream face. Two centrifugal blowers with a maximum speed of 6200 rpm were operated in a parallel configuration to draw the air through the test section and discharge it to atmosphere. The air mass flow rate through the rig could be directly governed by speed control of the variable speed 20kW d.c. motor which powered the blowers via a 4:1 speed increase pulley system.

3.4 Instrumentation

The function of the test rig was to experimentally determine flow field parameters within the VIGV system so that overall performance characteristics could be determined. A selection of both intrusive and non-intrusive methods of flow measurement was available, each method having its own distinct advantages and disadvantages as shown in Table 3.1

<table>
<thead>
<tr>
<th>Method</th>
<th>Advantages</th>
<th>Disadvantages</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pneumatic</td>
<td>Simple, well-proven, guaranteed accuracy in accordance with standards, 3-D measurement Reference: BS1042, Dimmock, Kassens and Rautenberg</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Slow response, inability to resolve unsteady flow but accuracy is affected by any unsteadiness, intrusive technique</td>
</tr>
<tr>
<td>Hot Wire</td>
<td>Fast response Reference: Witze</td>
<td>Difficult to use accurately in recirculating flows, inability to measure pressures, intrusive technique</td>
</tr>
<tr>
<td>Optical (PIV)</td>
<td>Fast response, PIV - high resolution snapshot of velocity vectors in 2-D flow plane, non-intrusive Reference: Grant, Gray</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>Expensive, inability to measure pressures, optical access and measurement in 2-D plane in spherical section passage is extremely complex</td>
</tr>
</tbody>
</table>

Table 3.1 Comparison of measurement techniques
Although the flow through the VIGV system was inherently unsteady to a small extent, the overall performance characteristics could be far more readily determined and compared under the assumption of a steady flow regime. Simplicity and speed of measurement of overall flow parameters was considered to be more important than high-resolution examination of the flow field (e.g. examination of flow within the boundary layer, and the use of non-intrusive techniques). With these points in mind, and the availability of existing instrumentation within the laboratory, pneumatic measurement methods were favoured.

3.4.1 Instrumentation Schedule

The test rig was instrumented in accordance with BS1042\textsuperscript{[55]}, and followed recommendations outlined by Dimmock\textsuperscript{[56]}, and Ower and Pankhurst\textsuperscript{[61]}. Atmospheric air was drawn from the laboratory through a standard BS 1042 4" nozzle or 7" orifice plate for corrected air mass flow rates of 0.5 kg/s or 1 kg/s respectively. The air mass flow rate through the rig could be accurately calculated using a static pressure tapping mounted just downstream of the orifice plate or nozzle. Since the mass flow measurement device worked on an 'upstream and downstream large' principle, no upstream tapping was necessary as the laboratory ambient conditions could be used. The test section of the rig was assumed to be adiabatic, hence total temperatures were assumed to be constant throughout and temperature measurement was not required when determining velocities.

A Kiel probe was introduced into the flow immediately upstream of the vanes at the mid-span position (as seen near the bottom of the passage in Figure 3.6) to measure VIGV inlet total pressure and account for losses in the plenum chamber.

Complete 2-dimensional flow field surveys could be conducted at two different axial planes within the VIGV passage, indicated in Figure 3.7. The first test plane was located approximately one blade chord downstream of the vanes whilst the second plane, as discussed earlier, was located in the plane in which the impeller leading edge would normally be found (the outlet plane). Twelve equi-spaced holes around the circumference of the shroud in each of these planes provide traverse positions which effectively vernier across the blade pitch since there are thirteen vanes. Dynamic
head and velocity vectors could be deduced anywhere in the flow field in these two planes from a calibrated three-hole wedge probe controlled radially by a stepper motor driven traverse unit.

Fig. 3.7 Instrumented VIGV system

Six static pressure tappings were spaced at equal axial intervals along opposite sides of the hub, and eight similarly spaced along the shroud.
3.4.2 Instrumentation Accuracy

The use of pneumatic instrumentation in flow field surveys such as this will inherently introduce a certain degree of instrument error into the results. However, the magnitude of this error can be determined and minimised by prior calibration against other instrumentation of a known accuracy.

Since the inlet nozzle and orifice plate were manufactured, installed and operated in accordance with BS 1042\textsuperscript{[35]}, a maximum mass flow rate error of \( \pm 4\% \) could be guaranteed. Calibration of the wedge probe and Kiel probe was conducted in a steady flow wind tunnel incorporating both a Venturi meter and pitot-static tube for velocity measurement. Velocity measurements using the wedge probe were verified against the pitot-static tube over a range of mass flow rates, and yaw angle measurement could also be easily verified since the flow was purely axial. Total pressure measurement using both the wedge probe and Kiel probe were also calibrated against the pitot-static tube. A fundamental requirement of the Kiel probe is not only to measure total pressure accurately, but also to do so over a wide range of probe incidence angles. This capability was assessed by measuring the response to yawing the probe in the wind tunnel axial flow. A comparison of the responses of the Kiel probe and pitot-static tube to yaw angle is shown in Figure 3.8. It can be seen that the response of the Kiel probe is unaffected by flow incidence angles up to \( \pm 25^\circ \) in any direction, whereas the pitot-static tube will only tolerate a maximum of \( \pm 5^\circ \) of incidence (pressure coefficient = difference between actual and measured total pressure divided by actual dynamic head). This feature is crucial to the accurate measurement of total pressure loss through the VIGV system as the inlet flow to the system is not purely axial (a small radial velocity component also occurs due to the flow within the plenum chamber).
3.4.3 Instrument Control and Data Acquisition

The development of a versatile automated technique to control the instrumentation and record the data was necessary so that the time taken to conduct the tests could be radically reduced, as demonstrated by Backhouse and Ivey\cite{62}. The risk of manual data measurement and input error would be also be reduced using such a technique. The radial position of the wedge probe was controlled automatically from a PC via the purpose-built stepper motor drive unit shown in Figure 3.9.

Fig. 3.9 Wedge probe traverse gear
Chapter Three: Experimental Facility

The unit could be attached to any of the yawmeter traverse holes in either of the two axial planes and automatically configured for the particular plane. Yaw angles were measured and recorded manually to an accuracy of ±1°, and all readings were sampled on a 5 second averaging basis. Continuous traversing or positioning of the probe at any of eleven equally spaced radial positions within the passage could be controlled directly from the PC. The radial positional tolerance of the probe was less than 0.5mm, and it was calibrated before each traverse by using the hub wall as a datum. A view of the test facility and instrumentation is shown in Figure 3.10.

![Fig. 3.10 Test rig and instrumentation](image)

A single Furness Controls FCO510 calibrated digital manometer was used for all pressure measurements, the pressure input source being dictated by a Furness Controls FCO91 scanning box. The manometer, scanning box and traverse gear were all interfaced to the PC for automatic control by the PC software as shown in the schematic diagram in Figure 3.11. The software was written and compiled in Microsoft QBASIC and controlled the probe positioning and data acquisition and storage so that manual errors were minimised. The software source code is provided in Appendix A.
**3.5 Test Procedure**

In order to ensure consistent relative test conditions and measurements, a standard test procedure was followed for each configuration using the same instrumentation. With the vanes positioned at the required setting angle, and the mass flow rate set, all of the VIGV section static tappings were sampled, followed by the nozzle or orifice plate pressure drop. A radial traverse in one of the circumferential probe hole positions was then conducted. Dynamic head, pressure loss through the passage, and yaw angle were logged at eleven radial positions from hub to shroud. The procedure was then repeated in all of the circumferential probe hole positions for that particular operating configuration. The test was conducted at IGV setting angles of 0°, +30° and +60° for mass flow rates of 0.5kg/s and 1kg/s. Due to the limited availability of the VIGV system used in this study, tests were conducted in the outlet plane for both mass flow rates, but were only conducted in the blade plane at 0.5kg/s.

**3.5.1 Data Reduction**

All of the test data was automatically stored by the control software in ASCII data files in a format that can be imported into a spreadsheet for further manipulation. In addition to the numerical data, a text header was automatically added to each data file to describe the test configuration and ambient conditions. The numerical data...
Chapter Three: Experimental Facility

consisted of the VIGV system hub and shroud static pressure readings, the Kiel probe reading, mass flow measurement device reading, followed by the traverse data (radial position, dynamic head, total pressure loss, and yaw angle). An example of the data file output is provided in Appendix B.
CHAPTER 4

EXPERIMENTAL RESULTS

4.1 Measurement technique validation

The instrumentation used in the test rig has been described in the previous chapter together with the individual calibration techniques to ensure that their behaviour was well understood. In addition it was considered necessary to ensure that the instrumentation gave valid readings when used in the test rig. Verification of the experimental technique was conducted using the results from each complete test configuration, as described below. This provided a simple test to ensure that the instrument and data acquisition control software and the data reduction technique were producing reliable results.

The wedge probe provided the local static and stagnation pressure along with the fluid direction. From this data, the velocity vectors could be derived as well as the local axial velocity at each span-wise position and pitch-wise location. Mass flow rate was then accurately calculated at each position, and integrated across the span and pitch to give a total mass flow rate through the VIGV system. Comparison of the mass flow rate calculated from the traverse technique were then compared with that calculated from the nozzle or orifice.

The average percentage error in mass flow rate calculation using the traverse technique based upon the nozzle or orifice mass flow rate is plotted in Figure 4.1 against vane setting angle (traverses conducted at outlet plane). The magnitude of the mass flow rate error using the traverse technique increased with vane setting angle, but also varied between tests conducted at the same configuration. The increase in error with setting angle is probably due to the limited accuracy and resolution to which the yaw angle could be measured from the wedge probe. It should also be
noted that the use of a three-hole wedge probe permits only 2-dimensional flow velocity measurement (axial and tangential components). The use of a 5-hole probe for full 3-dimensional flow measurements would also permit the measurement of the radial velocity component, however a suitable 5-hole probe was not available for this study. Consequently, the presence of a radial flow component in the outlet plane of the VIGV system could be partially attributable for the validation error. It is likely that the magnitude of any radial flow component in the VIGV system outlet flow would increase with setting angle as a result of radial equilibrium. This likelihood is therefore in some agreement with the trends shown in Figure 4.1.

Fig. 4.1 Validation of traverse technique

4.2 VIGV System Experimental Performance

4.2.1 Vane Performance

The objective of traversing in the plane immediately downstream of the vanes was to provide an understanding of the vane flow behaviour in isolation, i.e. without the aerodynamic effect of the converging spherical passage. Although aerodynamic influence from the passage was inevitable due to the test plane's location, an understanding of the aerodynamic influence of the passage shape would still be
achievable. All of the results described in this section are taken from the traverse data at this plane, and all results in the Chapter are taken from tests at a mass flow rate of 0.5kg/s.

The variation of average deflection with vane setting angle is plotted in Figure 4.2. It is clear from this figure that the converging passage shape significantly affects the aerodynamic output from the vanes, as the deflection is higher than the actual vane setting angle. This effect can be explained by consideration of conservation of angular momentum and continuity through the passage, as described in Section 2.3.1. In this section of the passage, the mean radius decreases in the flow direction at a greater rate than the orthogonal area, with a resulting increase in the mean flow angle.

![Graph showing deflection vs. vane setting angle](image)

**Fig. 4.2 Deflection vs. vane setting angle**

The aerodynamic influence of the passage shape is further highlighted in the radial distributions of pitch-wise-averaged deflection and velocity at this plane, Figure 4.3. At vane setting angles of 0° and 30° the distributions are quite linear, but at 60° a peak in both deflection and velocity can be seen at approximately 25% span. This suggests that the flow is separating from either the hub wall or from the vane suction surface near the hub. Either of these phenomena are possible due to the vane and passage design although if the passage itself was purely responsible, some separation (or reduction in velocity) near the hub at lower vane setting angles would be expected. At higher setting angles, the distribution will be affected by radial equilibrium of the flow through the passage, i.e. the balance between centrifugal forces and static
pressure across the span. This would account for the reduction in velocities near the hub at higher setting angles. It should also be noted from Figure 4.3 that the measurement of zero deflection at the IGV setting angle of 0° has demonstrated the accuracy of the angular position of the vane and the traverse technique.

Fig. 4.3 Radial distributions of deflection and velocity

Fig. 4.4 Circumferential distributions of deflection and velocity
Chapter Four: Experimental Results

The circumferential distributions of span-wise-averaged deflection and velocity in Figure 4.4 do not exhibit the same uniformity as the radial profiles. As the circumferential traverse crosses the blade wake, a noticeable change in both velocity and deflection could be expected such as that exhibited at the 60° vane setting angle. However, the trends in Figure 4.4 suggest that a distinct blade wake is not present at this axial plane at lower setting angles, a far more turbulent flow field is depicted instead. This level of flow unsteadiness was also apparent during the testing as a highly fluctuating pressure reading from the wedge probe.

The magnitude of the losses caused by this turbulence and by incidence and viscous losses was measured using a mass-averaged total pressure loss coefficient described below.

\[
\text{Pressure loss coefficient} = \frac{(P_{01} - P_{02})}{(P_{01} - P_1)}
\]

- \(P_{01}\) = total pressure at inlet to guide vane passage
- \(P_{02}\) = total pressure at exit of guide vane passage
- \(P_1\) = static pressure at inlet to guide vane passage

The significance of incidence losses when using flat plate vanes at high vane setting angles can be seen from Figure 4.5. Although the vanes successfully deflect the air at high setting angles, a high penalty is incurred by the dramatic increase in losses.

![Fig. 4.5 Pressure loss vs. vane setting angle](image-url)
4.2.2 Overall Performance

Although measurements conducted in the upstream traverse plane provided useful information regarding the performance of the vanes themselves, an assessment of the performance of the VIGV system as a whole is of equal importance. Traversing in the outlet plane provided a measurement of the combined effects of both vane and passage aerodynamic performance, and gave an indication of the flow regime that would normally enter the inducer.

Figure 4.6 shows that, despite its complex meridional shape, the converging section passage has a minimal effect on the average deflection. The slight reduction is expected and occurs as a result of the passage area and radius ratio at the inducer inlet as described earlier.

![Fig. 4.6 Deflection at VIGV system outlet vs. vane setting angle](image)

Span-wise-averaged and pitch-wise-averaged velocity and deflection distributions at the outlet plane are shown in Figures 4.7 and 4.8 respectively.
Chapter Four: Experimental Results

Fig. 4.7 Radial distributions of deflection and velocity at VIGV system outlet

Fig. 4.8 Circumferential distributions of deflection and velocity at VIGV system outlet
Chapter Four: Experimental Results

Figure 4.7 shows that the passage has little effect on the radial distributions of deflection and velocity, with the exception of the velocity profile at the 60° vane setting angle. The sharp reduction seen in absolute velocity towards the hub in Figure 4.3 is no longer evident, thus reinforcing the suggestion that it was originally caused by vane or hub separation, which has re-attached by the time the outlet plane is reached.

The circumferential distributions of deflection and velocity in Figure 4.8 also show little change in comparison with those measured further upstream. As expected, the distributions are more uniform as turbulence generated by the vanes has by now mixed out. The dissipation of turbulence was also evident as a much steadier pressure reading from the wedge probe. Absolute velocities in Figures 4.7 and 4.8 have all almost doubled due to the decrease in cross-sectional flow area from the upstream traverse plane to the outlet plane.

The aerodynamic benefits of the passage shape in terms of turbulence dissipation and separation re-attachment have been presented so far, but total pressure loss through the system is also inevitable and must be minimised. A comparison of the loss coefficient at the upstream traverse plane and the outlet plane is presented in Figure 4.9.

![Graph showing pressure loss vs. vane setting angle](image)

Fig. 4.9 Pressure loss vs. vane setting angle
The trends clearly indicate that the passage shape is of sound aerodynamic design as it is responsible for a negligible loss when the vanes are operating at low setting angles. At 60° however, the passage and vanes share an almost equal proportion of the total system loss. This is probably due to the extent of the separation pocket observed near the hub in the upstream plane, which will travel further downstream thus resulting in a high passage loss. The vorticity generated by the vane deflection also means that the effective passage length, and hence loss, will be much greater at higher setting angles.

It should be borne in mind that however negligible the inlet losses might initially appear, the effect on the overall stage performance should ultimately be considered. Any losses or flow non-uniformities at the inlet to the compressor will be amplified by the stage pressure ratio and could potentially be responsible for much greater overall stage losses.

4.2.3 Mass Flow Rate Scaling

As described in Chapter 3, the majority of the testing was conducted at a mass flow rate of 0.5 kg/s, although testing at other flow rates was necessary to validate the assumption of non-dimensionality of the pressure loss coefficient. Average outlet swirl angle and the system total pressure loss coefficient were the two main aerodynamic performance parameters used for comparison within the study, both of which should be unaffected by varying mass flow rate.

In order to validate this assumption, tests were also conducted in the outlet plane at a mass flow of 1 kg/s using the same instrumentation and test technique. Radial and circumferential distributions of deflection and velocity are not presented, as the profiles are all identical to those described earlier in this chapter (except for the magnitude of absolute velocity). The overall outcome of the testing is shown in Figure 4.10 where the total pressure loss coefficient is plotted against average outlet swirl angle (rather than vane setting angle). Only slight discrepancies in the magnitude of swirl angle and pressure loss coefficient are evident. More importantly, the trends exhibit a strong correlation suggesting that the parameters are sufficiently unaffected by mass flow rate.
Fig. 4.10 Pressure loss vs. outlet swirl angle at 50% and 100% flow rate

Examination of the Reynolds number at the inlet guide vanes revealed that the tests were being conducted around the transitional region between laminar and turbulent flow. Testing at 0.5kg/s corresponded to a Reynolds number of approximately $8 \times 10^4$, while 1kg/s was equivalent to $1.6 \times 10^5$. As shown in Figure 4.10, despite the transitional nature of the flow at these mass flow rates, the pressure loss coefficient was unaffected by the mass flow rate.

At the outlet from the VIGV system, a similar trend in Reynolds number was revealed, with a minimum of $1.5 \times 10^5$ at 0.5kg/s. However, the critical Reynolds number at this position based on flow in pipes was approximately $8 \times 10^3$.

4.3 Closure

Despite the limited scope of the test technique, the results presented in this chapter have demonstrated the applicability of the technique and provided an adequate description of the flow field within the VIGV system. The results also form a sound basis against which the output of the numerical technique can be validated and compared.
5.1 Introduction

In addition to the experimental study, one of the key focuses of the research was the development of an accurate and versatile numerical performance prediction technique. It was also imperative that the technique was both fast and reliable, as a considerable number of 3-D simulations would be conducted during the course of the research. Both 3-D and 1-D techniques were investigated, each having its own particular benefits and disadvantages. The development of the 3-D technique, and a brief summary of the 1-D technique is discussed in this chapter.

5.2 Computational Fluid Dynamics – An Overview

The design of VIGV systems has generally relied upon 1D and 2D theoretical techniques, but with the increasing versatility of Computational Fluid Dynamics (CFD) solvers and pre-processors, a 3D approach can now more readily be followed. If this approach is to be financially viable, a standard technique must be developed by which CFD models may be created and solved to an acceptable level of accuracy in as short a time space as possible.

Turbomachinery flows are normally three dimensional, and are often turbulent and separated. For this reason, accurate flow field predictions require CFD packages that are developed with this application in mind. In addition to this, the solution accuracy is also dependent on the modelling technique from pre to post processing. If CFD is to be used effectively in analysing and comparing a number of similar systems, a
Chapter Five: Numerical Techniques

standard proven pre-processing, solving and post-processing technique is required. Not only will this accelerate the whole modelling process, but it will also ensure consistent levels of accuracy due to grid topology and solution convergence.

Two such techniques were developed; initial work focused purely on the application of CFD to the VIGV system, with subsequent work being conducted on modelling the entire stage.

5.3 VIGV Simulations

5.3.1 Geometry Definition

Three aspects of turbomachinery internal design require consideration for accurate geometrical representation in CFD models; the vane shape, and the hub and shroud shapes. Although software packages such as CFX BladeGen have recently been developed to accomplish this task, no such packages were available when the research begun so a versatile geometry definition process was required for expediency.

Due to the circumferential periodicity of most turbomachines, only one blade and pitch need be modelled since periodic boundary conditions can be applied to the sides of the domain. This means that a single blade must be fully defined in 3-dimensional co-ordinates, whilst the hub and shroud (which are axisymmetric) need only be defined as 2-dimensional co-ordinates. The geometry definition system was required to be both versatile and fast so that virtually any conceivable passage and vane shape could be modelled accurately.

The hub and shroud co-ordinates (in almost all cases) followed the design rules proposed by Swain[50] that are outlined in Section 2.3.1. These rules can be interpreted as a series of trigonometrical relationships between axial and radial co-ordinates at any point along the passage. The capability to modify the passage shape within the constraints of these rules was required with minimal user input. From the equations outlined below relating to the parametric VIGV system geometry in Figure
5.1, it was possible to automatically generate any passage shape based upon the rules by entering only the area ratio and radius ratio.

**Fig. 5.1 Parametric VIGV system geometry**

![Parametric VIGV system geometry diagram](image)

Area Ratio \( (A_R) = \frac{R_{s1}^2 - R_{h1}^2}{R_{s2}^2 - R_{h2}^2} \)

Radius Ratio \( (R_R) = \frac{R_{s1} + R_{h1}}{R_{s2} + R_{h2}} \)

Where \( R_{h2} \) and \( R_{s2} \) are dictated by the inducer hub and tip radii (35mm and 112.8mm respectively)

From simultaneous equations, it can then be seen that:

\[
R_{s1} = \frac{(A_R \left( R_{s2}^2 - R_{h2}^2 \right)) + \left( R_R \left( R_{s2} + R_{h2} \right) \right)^2}{2R_R \left( R_{s2} + R_{h2} \right)}
\]

and

\[
R_{h1} = R_R \left( R_{s2} + R_{h2} \right) - R_{s1}
\]

also,

\[
L = 2 \sqrt{R_{s1}^2 - \left( \frac{R_{s1} + R_{s2}}{2} \right)^2}
\]

and

\[
R_h = \frac{R_{h2}^2 + L^2 - R_{h1}^2}{2 \left( R_{h1} - R_{h2} \right)}
\]

From the design rules, \( R_s = R_{s1} \)

Hence all geometrical parameters can be deduced from the area ratio and radius ratio.
Implementation of the equations into a spreadsheet enabled fast automated co-ordinate generation with the versatility to change and view the meridional passage shape before being exported to the mesh generator.

Blade co-ordinates were generated in a similar fashion using trigonometrical relationships that permitted any vane shape to be generated at any setting angle. A series of calculations are executed automatically in a spreadsheet in the following sequence:

1. Since the blade is untwisted, the profile need only be provided at the hub and shroud of the blade (the mesh generator will linearly interpolate to generate the rest of the vane profile). Since the hub and shroud are both spherical the blade hub and tip must follow a circular arc, which means that the vane chord can be measured in terms of the angle subtended by this arc (Figure 5.2). If a vane design is selected that does not have a constant pitch/chord ratio across its span, the spreadsheet can be adjusted by entering differing hub and tip chord range.

2. The vane is now approximated as a 2-dimensional flat plate with a chord and a span aligned in the axial direction. The vane thickness and profile is added next by simply adding 2-dimensional profile co-ordinates into the spreadsheet assuming the vane is set at 0° (axially aligned). Twisted or untwisted, cambered, profiled or flat plate vanes with chamfers can be added to make the vane a 3-dimensional entity.

3. Trigonometrical manipulation of the hub and shroud co-ordinates allows the vane to be rotated about its centre-line shown in Figure 5.2, 5.3 and 5.4. The only user input required at this stage is the vane setting angle (θ). Again, a meridional view is provided by the spreadsheet before the data is exported to the mesh generator.
5.3.2 Mesh Generation

The pre-processor used in these investigations, CFX TurboGrid, imported the coordinates from ASCII data files (exported from the spreadsheet) and automatically generated the vertices which define the domain. A standard H-type grid template was used in which the blade consisted of a section of 'blocked-off' cells. CFX TurboGrid is a semi-automatic mesh generator in which the user can position a number of 'construction lines' that will dictate the complete grid layout. Care was required at this stage to minimise resulting cell skewness and aspect ratios, and to determine the exact distribution and number of cells. The template would automatically refine the grid at the hub and shroud, and around the vane as these were the areas of main concern in the simulations.

Initially, three templates (IGV setting angles of 0°, +30°, +60°) were developed and tested so that they could be stored and re-used to generate future grids. This significantly reduced the time required to create high quality grids and meant that any changes to the vane or passage shape could be easily accommodated. Figures 5.5 a, b, and c show mid-span radial slices of the grids created from the templates with the vane set at 0°, +30° and +60° respectively.
Fig. 5.5a Mid-span radial grid slice at IGV setting angle = 0°

Fig. 5.5b Mid-span radial grid slice at IGV setting angle = +30°

Fig. 5.5c Mid-span radial grid slice at IGV setting angle = +60°
Accurate geometrical representation of such blade rows in CFD simulations is crucial if repeatable and accurate results are required. One important feature in the modelling of blade rows that has been explored extensively in recent years is blade tip clearance. Modelling blade tip clearance is now considered essential in rotating blade rows, and its significance in stationary blades at high stagger angles should not be overlooked.

As described in Section 3.3, a spherical section hub and shroud minimised the blade hub and tip clearance in the inlet guide vane system used in this research. Although the vane hub and tip clearance measured from the test facility was less than 1mm, or 1% blade height at any setting angle, initial investigations were conducted into its effect on the outlet flow field from the VIGV system. Two identical grids (grid A and grid B) were created with a vane setting angle of +60°. The blade in grid A (Figure 5.6a) was assumed to be solid from the passage hub to shroud (no blade clearance). The blade ‘block-off’ in Grid B (Figure 5.6b) was modified to mimic the clearance at blade hub and shroud including the stem that attached the blade to its spindle.
The effect of clearance modelling is demonstrated by the results of the two simulations, which have been plotted as pitchwise-averaged VIGV system outlet swirl angles in Figure 5.7. The effect is particularly significant near the shroud at the outlet to the VIGV system, and it was considered necessary to include clearance modelling in all future VIGV meshes.

In addition to accurate geometrical representation and grid quality, the grid resolution is another important factor in practical CFD simulations. If too few mesh cells are used in the simulation, the accuracy of the results may be compromised. Conversely, if too many cells are used, the demands on computing resources and solving time can be increased with little or no effect on the solution accuracy. It is therefore usual to conduct a grid dependency study at the outset of the CFD analyses to ensure that an optimum grid resolution is used throughout.

Two grid sizes and numerous cell distribution patterns were investigated in order to assess grid sensitivity. Initially, a mesh of approximately 100,000 cells was created and tested, followed by a 150,000 cell mesh with a similar cell distribution pattern. A decrease in total pressure loss of approximately 10% was noted with the increase in mesh density, but the distribution and magnitude of swirl angles and absolute velocities at the outlet plane remained essentially unchanged. Although a higher solution accuracy in terms of pressure loss would possibly have been achievable with a larger number of cells, the increase in solving time of approximately 100% from 100,000 to 150,000 cells meant that the smaller grid offered a far more acceptable
solution turnaround time for a practical design procedure. A grid density of approximately 100,000 cells was therefore considered to provide sufficient accuracy and resolution to be adopted as the standard for all future working grids. Grids of this density also proved to be of an acceptable quality in terms of cell skew angle and aspect ratio in order to allow accurate and stable solving.

5.3.3 Solving

The application of appropriate and accurate boundary conditions to any simulation is critical to the solving speed and solution accuracy, Figure 5.8. In this case, the most robust known boundary conditions were found to be inlet total pressure and either outlet average static pressure (for low mass flow rate studies), or outlet mass flow rate (for high mass flow rate studies). (A spanwise inlet velocity profile was also applied in a test case, but was found to have a negligible effect on the final flow field and increased the pre-processing and convergence time.) CFX-TASCflow solves the Reynolds stress averaged Navier Stokes equations, and is purpose written along with the pre-processor, CFX TurboGrid, for the modelling of complex turbomachinery geometries. A second order modified linear profile skew discretisation scheme is employed as this offers a combination of high accuracy along with robustness. Turbulence was modelled using the $k$-$\varepsilon$ Kato-Lauder\textsuperscript{[63]} model.

![Fig. 5.8 VIGV grid with shroud and one periodic face removed](image-url)
With each simulation starting from an initial guess of a uniform flow field, convergence (reduction of r.m.s. residuals to a maximum of $10^{-4}$) was typically achieved within approximately 100 iterations or 12 hours of CPU time on a Sun UE2-2200.

A rotated view of the complete VIGV system (13 vanes and passages) with the shroud removed is shown in Figure 5.9.

Fig. 5.9 Rotated mesh showing complete VIGV system
5.4 Stage Simulations

The technique developed for numerically modelling the VIGV system was subsequently adapted to incorporate the whole stage; i.e. the impeller and diffuser vanes in addition to the inlet guide vane system. Using the associated pre-processing software, CFX Turbogrid and CFX TASCbob, it was possible to construct the numerical mesh for each component of the stage individually and then numerically 'glue' them together.

5.4.1 Geometry Definition

Although the VIGV geometry used in the stage investigations was identical to that described in sections 3.2 and 5.3, it was necessary to move the axial location of the outlet plane upstream. This was necessary for two reasons:

1) When modelling the VIGV system in isolation, the hub extension used in the experimental facility was also modelled for performance comparison. When operating in the complete stage, this extension is replaced by the rotating inducer hub.

2) Ideally the VIGV outlet / impeller inlet plane should be located at the mechanical connection between the two devices as this where the rotating impeller hub meets the stationary VIGV system hub. However, using this plane as the numerical interface between the components would lead to solution convergence problems due to the proximity of the vane leading edges to the inlet boundary of the impeller. Conversely, if the numerical interface was moved too far upstream of the impeller leading edge, the solution accuracy would be compromised by the modelling of the stationary VIGV system hub as a rotating surface.

The impeller geometry was already available, and had been generated using a suite of 1-D preliminary design and 2-D aerodynamic analysis software described in detail by Came and Robinson\textsuperscript{[64]}. 
Since the diffuser was a variable geometry system, a versatile geometry definition technique similar to that used for the VIGV's was required. The technique followed the same principle outlined in section 5.3 so that any vane profile could be applied and manipulated to generate the co-ordinates at any vane setting angle. The spreadsheet was further modified so that the vane co-ordinates could be generated given either a setting angle (relative to the radial direction) or a diffuser throat area, Figure 5.10.

Fig. 5.10 Variable diffuser vane geometry

The hub and shroud of the diffuser model used in the numerical studies were extended radially beyond the point where they would normally meet the volute. This was necessary to compensate for the absence of the volute, as the flow field at exit from the diffuser would be otherwise be undeveloped and cause simulation convergence problems. In addition to this extension, the shroud wall was modified so that the diffuser passage converged slightly to prevent separation at the walls and aid convergence. The complete stage meridional passage is shown in Figure 5.11 with the modifications detailed in this section.
5.4.2 Mesh Generation

The VIGV mesh described earlier was slightly modified to account for the reduced length of the system, but maintained the same topology. As well as modifying the existing grids, two additional grids at IGV setting angles of +20° and +40° were created for use in the stage simulations.

The impeller mesh was based on an available template designed to apply an H-type grid to an impeller incorporating one set of splitter vanes. Tip clearance was modelled in the impeller, with a total grid size of approximately 72,000 cells. Figure 5.12 shows two computational domains (one rotated) consisting of the impeller main vane, splitter vane and hub (shroud removed for visualisation).

Modelling the vaneless space between the impeller outlet and the diffuser inlet was possible by splitting the impeller grid into two separate grids. The vaneless annulus that can be seen downstream of the impeller trailing edge was automatically removed and re-saved as the vaneless space grid so that a different set of boundary conditions could be applied to it.
The diffuser vane mesh was created using a similar H-type grid template to that applied to the VIGV’s. No hub and tip clearance was modelled in the diffuser vane grid due to the high skewness of the cells within the vane, which would lead to simulation convergence and accuracy problems. Grids and templates were created for three different diffuser vane setting angles, each with a mesh density of approximately 64,000 cells. In accordance with the VIGV and VVD operating algorithm, the diffuser vane angles selected for modelling were 62.2°, 68.3°, and 71°. These settings corresponded with VIGV setting angles of 0°, +20°, +40°, and +60° respectively (a diffuser setting angle of 71° was used at VIGV settings of +40° and +60°).

Figures 5.13 a, b, and c show the blade shape and hub of the diffuser meshes at setting angles of 62.2°, 68.3°, and 71° respectively. At lower setting angles, it was necessary to extend the diffuser mesh radially inwards due to the proximity of the vane leading edge to the mesh inflow boundary. Consequently, the outer radius of the vaneless space mesh was reduced to compensate for the change and ensure that the two meshes did not overlap.
Fig. 5.13a  Grid slice at hub for diffuser vane setting angle = 62.2°

Fig. 5.13b  Grid slice at hub for diffuser vane setting angle = 68.3°

Fig. 5.13c  Grid slice at hub for diffuser vane setting angle = 71°
5.4.3 Grid Interfaces

Once the grids had been created, it was necessary to numerically ‘glue’ them all together to form one complete model of the stage incorporating the VIGV’s and the variable vaned diffuser. Two types of numerical interface were employed in the model using the ‘General Grid Interface’ (GGI) feature within CFX-TASCflow.

At the interface between the impeller and vaneless space grids, a ‘General Connection Interface’ could be used as the grids were stationary relative to one-another so no sliding interface was necessary. Although this type of interface allows adjacent grids without perfect node-to-node alignment to be linked, this was not actually necessary in this case as the two grids had originally been created as one complete grid.

The interfaces between the VIGV and impeller, and the vaneless space and diffuser posed the additional complexity of a change of relative motion. For this reason, a ‘Stage Interface’ was used in which a rotating and stationary grid can be linked together numerically. Using this type of interface, two or more blade passages can be solved simultaneously with circumferential averaging between rotating and stationary regions. Steady state solutions can then be obtained in each reference frame. The stage interface accounts for the mixing loss caused by the relative motion between components. The interface mixing is assumed to be sufficiently large to compensate for changes in pitch between components, such that any upstream velocity profile has mixed out prior to entering the downstream component, TASCflow3D documentation[65].

Grid dependency studies of each of the grid components indicated that a total mesh size of approximately 225,000 nodes gave the optimum performance without unnecessary demands on computing resources. One complete vane passage through each of the components attached together was modelled, but the entire system is shown in Figure 5.14 (inlet guide vanes set at 60° to axial, diffuser vanes set at 71° to radial).
Fig. 5.14 Rotated mesh showing complete compressor stage
5.4.4 Solving

Two operating points on each constant VIGV angle characteristic were modelled using ambient total pressure and temperature for the inlet boundary condition (inlet to the VIGV system) and mass flow rate for the outlet boundary condition (outlet of the diffuser). Turbulence was modelled using the standard k-ε model.

The physical boundary conditions applied to the model are outlined in Table 5.1 below. All walls were assumed to be smooth and adiabatic, the impeller and vaneless space grids were defined to be rotating at the same speed, whilst the VIGV and diffuser grids were defined as stationary.

<table>
<thead>
<tr>
<th>Boundary Condition</th>
<th>Attachment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stationary wall - relative frame</td>
<td>Default, all internal objects (inlet guide vane, impeller main and splitter vane, diffuser vane)</td>
</tr>
<tr>
<td>Counter rotating wall - relative frame</td>
<td>Impeller shroud, vaneless space hub and shroud</td>
</tr>
<tr>
<td>Rotational periodicity</td>
<td>Radial sides of all grids</td>
</tr>
<tr>
<td>General connection interface</td>
<td>Impeller grid $\rightarrow$ vaneless space grid</td>
</tr>
<tr>
<td>Stage interface</td>
<td>VIGV grid $\rightarrow$ impeller grid</td>
</tr>
<tr>
<td>Vaneless space grid $\rightarrow$ diffuser grid</td>
<td></td>
</tr>
<tr>
<td>Inflow</td>
<td>VIGV inlet</td>
</tr>
<tr>
<td>Outflow</td>
<td>Diffuser outlet</td>
</tr>
</tbody>
</table>

Table 5.1 Boundary conditions applied to full stage model

Simulation convergence from an initial guess was achieved by starting at low speed with a corresponding mass flow rate and progressively increasing the speed after approximately 10 time steps at each speed. Maintaining a constant grid density and topology throughout the various configurations enabled any converged flow field for a
given configuration to be used as an initial guess for the next configuration to be modelled. This technique minimised solving time which was typically 10 to 14 CPU hours using a twin 336 MHz processor SUN Ultra Enterprise 3500 server.

5.5 1-D Stage performance predictions

The 3-D numerical technique was capable of producing detailed descriptions of the flow field within the compressor, so that the performance of the aerodynamic components of the stage could be analysed. Using this technique to predict the overall stage operating characteristics was however extremely inappropriate and inefficient due to the computing resources and the time required to solve the model.

Stage performance characteristics could be predicted with in a fraction of the time and resources by using an existing 1-D prediction technique. A modified version of the technique created by Swain and Connor\textsuperscript{15} was used as it was developed for exactly this type of variable geometry centrifugal compressor stage. It was based on one-dimensional methods and empirical correlations, and originally used a simple VIGV performance loss model, which was subsequently replaced with the models derived from the analyses described in this study.
CHAPTER 6

COMPUTATIONAL RESULTS

6.1 Validation of Experimental and Computational Techniques

CFD is an extremely valuable tool to improve the understanding of a fluid behaviour and can provide an insight into the most complex fluid problems. However, the use of CFD in the design and development of turbomachinery should be complemented wherever possible by comparable experimental data. Once validated, a standardised numerical technique can then be used in the design process with a high degree of confidence, and extrapolated to configurations that have not been experimentally tested. The numerical technique used in this research was initially validated over a number of test configurations and boundary conditions.

As described in Chapter 3, the test rig was capable of drawing an air mass flow rate of up to 1kg/s through the VIGV system. In normal operation however, mass flow rates of up to 6kg/s are possible through compressor stages using this size of VIGV system. Although the majority of the experimental testing was conducted at a mass flow rate of 0.5 kg/s, it was also considered necessary to conduct some tests at a higher value in order to assess the implications of mass flow rate on other aerodynamic parameters. The validity of the CFD predictions at varying mass flow rates could also be examined so that confidence could be gained in final predictions at the actual design mass flow rate for which only limited experimental data was available.

Experimental tests were conducted at mass flow rates of 0.5kg/s and 1kg/s with the inlet guide vanes set at angles of 0°, +30°, and +60°. Numerical simulations were conducted using equivalent boundary conditions: prescribed mass flow rate at outlet and total pressure and temperature at inlet. Although the majority of the experimental testing was conducted at 0.5 kg/s, the standard CFD simulation operating...
configuration of 1kg/s was selected for two reasons. Firstly, the specification of accurate boundary conditions was difficult as a static pressure outlet boundary condition (rather than a mass flow rate) was sometimes necessary to aid convergence at 0.5 kg/s mass flow rate simulations. Also, convergence time was significantly reduced using the 1kg/s boundary condition.

The aerodynamic performance parameters selected for comparison were those that were most relevant to the analysis and development technique itself. Since the ultimate objective of VIGV system development is to improve the performance of the whole compressor stage, attention was focussed on the VIGV system outlet flow field, as this is where interaction with the inducer leading edge would normally occur. Any distributions of aerodynamic parameters measured at this plane could be considered to be the inlet conditions to the centrifugal impeller. Two such parameters were selected for the validation technique at this plane: absolute velocity magnitude and swirl angle.

6.1.1 Absolute Velocity

Figure 6.1 shows the pitch-wise-averaged variation of absolute velocity at the outlet plane across the span for vane setting angles of 0°, +30° and +60°. The numerically predicted trends compare very well with the experimental results across the span. Discrepancies in the velocity near the hub and shroud walls can be attributed to inaccurate modelling of wall losses and vane clearance effects. Experimental measurements were not taken at close proximity to the walls, as the end-wall effects of the probe would result in misleading velocity readings. Although the experimental results reveal a similar non-uniformity at the outlet plane at the lower setting angles, the vane wake cannot be seen as distinctly as it can in the computational prediction.

6.1.2 Swirl Angle

The pitch-wise averaged distribution of swirl angles at the outlet plane remained uniform at all setting angles, again exhibiting a close correlation between experimental and numerical results, Figure 6.2. Discrepancies in the swirl angles near
the hub are apparent, and probably due to boundary layer development occurring within this region.

Fig. 6.1 Spanwise experimental and theoretical velocity profiles

Fig. 6.2 Spanwise experimental and theoretical deflection profiles
6.1.3 Pressure Loss Coefficient

In addition to the flow characteristics at the outlet plane, the performance of the VIGV system must also be assessed in terms of its efficiency. This has been calculated and presented as a non-dimensional pressure loss coefficient described earlier. Figure 6.3 shows the pressure loss coefficient at both flow rates, for both experimental and numerical results.

![Graph showing pressure loss coefficient vs IGV setting angle](image)

**Fig. 6.3 Experimental and theoretical pressure loss vs setting angle**

Identical trends are quite clearly seen, although the absolute values of the loss coefficient differ slightly. This is probably due to the inaccuracy of the numerical loss models within the flow field and the approximation of smooth wall losses employed by the CFD solver. A simulation was also conducted at 3kg/s (corresponding to a typical stage mass flow rate with the VIGV's set at +60°) to test the applicability of the loss coefficient. The results of this simulation proved that the loss coefficient was equally valid at mass flow rates corresponding to Reynolds numbers beyond the transitional values seen in the experimental tests.
The object of this exercise has been to examine the capability of the CFD technique described in Chapter 5 to predict a reliable and repeatable description of the flow within the VIGV system. In addition to the quantitative comparison of numerical and experimental data presented here, a qualitative comparison is equally important, i.e. the spatial distribution of scalars, and the performance trends with changing boundary conditions.

This initial validation process proved that the numerical predictions were sufficiently accurate for the technique to be used with confidence in the design and development process. Although some discrepancies are evident in the absolute values of the compared data, the trends are sufficiently comparable to enable potential design modifications to be accurately evaluated and compared relative to one another.

6.2 VIGV System Computational Performance

As explained in Chapter 1, an ideal centrifugal compressor VIGV system will be capable of inducing a high range of inlet swirl angles at low loss. Figure 6.4 shows the loss trend depicted in Figure 6.3 plotted against swirl angle rather than guide vane setting angle. It can readily be seen that swirl angles up to approximately ±60° are achievable using this system if increased losses are tolerated.
Chapter Six: Computational Results

In order to develop the VIGV system design and reduce the losses associated with high vane setting angles, an appreciation of the aerodynamic behaviour of the vanes and the passage was necessary. The use of more complex instrumentation and measurement systems such as optical techniques could have provided an extremely detailed view of the actual performance. However, efficient use was made of limited resources to validate the numerical technique using the most appropriate parameters. Although not providing a comprehensive view, the experimental technique gave an adequate understanding of the overall VIGV system aerodynamic behaviour. Having been validated by the experimental data, the numerical technique was therefore considered to be the most suitable approach to examine the flow field in detail.

The overall aerodynamic performance of the VIGV system was assessed in terms of the magnitude and distribution of outlet swirl angles, and the resulting total pressure loss, Figure 6.4. Using the CFD technique however, it was possible to examine the flow in considerably more detail anywhere within the system.
6.2.1 Vane Performance

Mid-span velocity vector plots at setting angles of 0°, +30°, and +60° are shown in Figures 6.5 a, b, and c respectively. As demonstrated in the experimental results, the deviation at mid-span is very low. Although the vane would ideally have a rounded leading edge profile, a straight chamfer is machined on the leading and trailing edge in order to reduce production costs. Incidence losses at the 0° setting angle are minimised by the leading edge chamfer, and the absence of the usual bulbous vane leading edge profile that is typical of vanes that are required to operate efficiently over high incidence ranges.

As the setting angle is increased, incidence losses and the tendency of the flow to separate from the leading edge of the suction surface also increase. The velocity vector plots support the trend depicted in Figure 6.4 relating the total pressure loss to swirl angle. It can be seen from both Figures 6.4 and 6.5 that at setting angles higher than 60°, the vanes would behave more as a throttling device rather than a swirl inducing device as the additional pressure loss penalty would exceed the benefit of the additional turning.

Figures 6.6 a, b, and c show the velocity vector plots around the vane at a setting angle of +60° at 10% span, 50% span, and 90% span respectively. Separation from the suction surface of the vanes is more apparent near the hub and results in more noticeable flow deviation from the vane trailing edge. The increase in deviation towards the passage walls is attributable to both boundary layer development and vane hub and tip clearance. Although vane hub and tip clearance is minimised by the spherical passage design of this VIGV system, some flow leakage over the vane hub and tip is inevitable.
Fig. 6.5 Mid-span velocity vectors

(a) Setting angle = 0°

(b) Setting angle = +30°

(c) Setting angle = +60°
Chapter Six: Computational Results

Fig. 6.6 Velocity vectors across span at setting angle = 46°

(a) 10% span
(b) 50% span
(c) 90% span
6.2.2 Overall Performance

From the near-hub absolute velocity contour plots in Figures 6.7 a, b, and c, it can be seen that the flow downstream of the vanes is dominated by the development of the separation pocket on the suction surface of the vanes at high setting angles. At 30°, Figure 6.7 b, a small area of separation on the suction surface of the vane can be seen, and the vane wake has not fully dissipated by the time the outlet plane is reached. However, at 60° (Figure 6.7 c), the separation pocket on the suction surface of the vanes is significantly enlarged but the vane wake is rapidly mixed out and does not extend far downstream. This is because the flow will follow a higher swirl angle at higher vane setting angles, and travel further before the outlet plane is reached. It will then have travelled sufficient distance so that the wake is no longer distinct. Non-uniformities at the inducer such as this will result in a decrease of the overall stage efficiency.

Absolute velocity contours at 50% span and 90% span at a vane setting angle of +60° are shown in Figures 6.8 a and b respectively. It can be seen from these figures that the severity of the separation pocket downstream of the vanes diminishes across the span towards the shroud.

Examination of the flow regime in a meridional plane revealed a small region of low total pressure near the hub surface immediately downstream of the vanes (when set at +60°), Figure 6.9. This is due to the toroidal shape of the passages, which results in a small degree of flow detachment and boundary layer growth on the hub surface. This confirms the suggestion made in Section 4.2.1 that the flow separates from the hub wall due to radial equilibrium at high vane setting angles. The resulting boundary layer extends throughout the passage, and will ultimately enter the impeller.

Figure 6.9 also shows that the flow follows the aerodynamic stream surfaces on the whole, so radial velocity components at the outlet plane are negligible and measurement in this plane is not necessary.
Chapter Six: Computational Results

(a) Setting angle = $0^\circ$  
(b) Setting angle = $+30^\circ$  
(c) Setting angle = $+60^\circ$

Fig. 6.7 Near-hub absolute velocity contours
Chapter Six: Computational Results

(a) 50% span

(b) 90% span

Fig. 6.8 velocity contours across span at setting angle = +60°
6.2.3 Aerodynamic Losses

In order to thoroughly assess the aerodynamic performance of the system, it was necessary to determine the source of total pressure loss throughout the passage. Further development or re-designing of the system would be greatly facilitated if losses could be attributed to particular areas within the passage.
Examination of mass-averaged total pressure across quasi-orthogonal surfaces throughout the passage, Figure 6.10, shows how the loss develops through the passage at the +60° vane setting angle. The overall implication of the hub wall separation and resulting boundary layer development at high setting angles can be seen here. At lower setting angles, the passage losses are negligible in comparison with the vane incidence and separation losses. At higher setting angles however, as suggested in Chapter 4, it can be seen that the passage downstream of the vanes is responsible for the same proportion of the system losses as the vanes themselves at high vane setting angles.

A reduction in total pressure loss coefficient (or increase in total pressure) is apparent at approximately 25% passage length. This is because the total pressures were calculated across quasi-orthogonal grid surfaces, and the axial distance taken from the mean radius stream-surface. As shown in the mesh views in Chapter 5, the grid distorts as the leading edge of the vane is approached, which means that the actual grid surface is in fact far from orthogonal.
Clearly, the two main areas of concern regarding the aerodynamic performance of the VIGV system are vane incidence loss leading to separation, and the separation occurring at the hub wall downstream of the vanes.

6.3 Closure

The computational predictions have shown a strong agreement with the experimental results and have enabled a more detail examination of the flow regime within the VIGV system. Sufficient confidence has also been gained in the computational technique for it to be used in the design process of potential new vanes and passage shapes.
7.1 Introduction

The experimental and computational analyses of the existing VIGV system suggested that there was scope for potential improvement to its overall aerodynamic performance. A set of aerodynamic design rules outlined by Casey\textsuperscript{[66]} were followed in an attempt to optimise the overall VIGV system design. The guidelines that are applicable to this study are outlined below with a brief description of each criterion and how it can be addressed.

- Avoid poor incidence onto blading

  In almost all turbomachines, the blade angle at its leading will be designed to provide optimum incidence and minimise losses over the required operating range of the machine. Cambered, curved and twisted vanes are often employed to satisfy this criterion, but are not so applicable in VIGV systems for a number of reasons:

  i) Variable inlet guide vanes are required to operate over a very high range of setting angles with low loss and deviation. In order to reduce incidence losses at high setting angles, PVD (Prescribed Velocity Distribution) studies have shown that a profiled vane would have an extremely bulbous leading edge. Although this would result in an acceptable level of incidence at high setting angles, the vanes' performance at low setting angles would be compromised as losses would be considerably increased due to the increase in blockage.

  ii) The addition of a fixed camber to the blade design will result in lower incidence losses at the deflection for which the blade has been designed. In the case of VIGV systems, however, the vanes are required to operate with minimal loss at 0° deflection as this corresponds to the design duty of the
compressor. If a highly cambered vane is positioned to produce 0° deflection, the losses will inevitably be much higher than at its designed setting angle.

iii) Profiled vanes represent a considerable increase in VIGV system manufacturing costs. Unless the vanes could operate with comparable or lower losses at the compressor design duty, then the long-term benefits in stage efficiency would be insufficient to justify the additional VIGV system cost.

- **Reduce friction on wetted surfaces**
  Frictional losses caused by the vanes and the passage walls are unavoidable, but can be minimised by opting for the smallest number of vanes with the shortest possible chord length to achieve the desired turning. A large passage area (low hub/tip ratio) will also reduce the severity of the passage wall losses and reduce velocities.

- **Avoid kinetic energy loss**
  Spanwise flow anywhere within the VIGV passage should be minimised in order to avoid any unnecessary kinetic energy loss. This includes radial velocities induced by boundary layer separation at the hub wall and by secondary flows caused by blade loading. Vane hub and tip clearance should also be minimised in order to avoid leakage jets.

- **Avoid flow separation**
  Mixing losses are incurred when separated flow leaving the vanes eventually mixes with the main flow. The vanes should therefore be aerodynamically designed to avoid such separation pockets and vane wakes so that the associated losses are minimised. Separation from the suction surface of the inlet guide vanes is clearly evident at high setting angles, as is the dramatic increase in the system total pressure loss.

- **Provide a uniform distribution of flow onto downstream blade rows**
  In multiple blade row machines, the outlet flow regime of one blade row is the inlet flow regime into the next one downstream. This means that an irregular flow pattern at the outlet of a blade row will probably result in a performance penalty in
the downstream one. Blade rows should therefore not be designed in isolation, and the implications of the outlet flow on downstream components should be considered.

As discussed in the previous chapter, the presence of any pitch-wise or span-wise non-uniformity in the VIGV system outlet flow can have a detrimental effect on the performance of the compressor. In addition to the design of the vanes, their axial location is also critical in this respect, as their proximity to the inducer is an influential factor in the uniformity of the inducer inlet flow.

Having assessed the performance of the existing VIGV system, the areas of potential improvement have been highlighted by this set of design guidelines. Using the now-proven computational technique to provide fast and accurate comparative performance predictions, parametric studies were conducted on a number of potential vane and passage re-designs.

Whilst it was accepted that the computational technique could not be guaranteed to provide an accurate absolute performance prediction (such as the magnitude of total pressure loss for a given design configuration), using a standardised technique enabled accurate relative comparisons from one design to another.

7.2 Passage Design

It has already been shown that the passage shares an equal proportion of the total system losses with the vanes at high setting angles. It has also been determined that the rate of convergence of the hub wall is responsible for a small degree of flow separation which probably contributes significantly to the passage losses. In addition to the effect on the system loss, the passage shape also affects the magnitude and distribution of velocities and swirl angles at the inducer inlet.

Clearly, the toroidal shape of the VIGV system passage is itself the root of a considerable proportion of the losses as it lengthens the meridional flow distance and results in adverse radial pressure gradients throughout the passage. Despite this
effect, the current parametric shape of the passage offers a distinct performance improvement over cylindrical designs, as detailed in Chapter 2. The use of a spherical surface hub and shroud profile in conjunction with circular inner and outer vane profiles resulted in constant blade clearance at hub and tip for any vane setting angle. The comparison of passage designs shown in Figure 7.1 demonstrates how the blade tip clearance at 0° setting angle can be minimised using a spherical shroud. Similarly, the blade hub clearance will be minimised at higher setting angles by the spherical passage hub. This spherical vane passage naturally leads to a swan-neck converging section passage design between the vanes and the impeller leading edge.

![Diagram of Cylindrical and Spherical Passage Designs](image)

**Fig. 7.1 Comparison of Cylindrical and Spherical Section Passages**

Using the geometric arrangement suggested by Swain\(^ {50} \), the VIGV system passage shape could be defined by two geometrical parameters; radius ratio and area ratio. These are the ratios of mid-span radius and total flow area from the inlet of the VIGV
system to the inlet to the inducer. Through conservation of angular momentum and continuity, the theoretical swirl angle at the inducer could be predicted based upon the vane setting angle (assuming zero deviation) and the radius and area ratios. A high radius ratio would tend to increase the turning due to conservation of angular momentum, whereas a high area ratio would tend to decrease the turning due to continuity. Swain\textsuperscript{[50]} suggested that the average outlet swirl angle could be approximated as a function of the passage geometrical parameters described in Section 5.3 by the following relationship:

\[
\text{Average Outlet Swirl Angle} = \tan^{-1} \left( \tan \theta \left( \frac{\text{radius ratio}}{\text{area ratio}} \right) \right)
\]

This assumes that there is zero deviation from the inlet guide vanes, and that the flow regime is unaffected by viscous losses at the passage walls. The outcome of the relationship over a range of conceptual passage geometries is shown in Figure 7.2. As expected, the trends indicate that an ideal passage shape will have a low area ratio and high radius ratio.

---

![Fig. 7.2 Effect of passage shape on outlet swirl angle](image-url)
Although this simplistic approach has provided an approximation of the effect of the passage geometry in terms of swirl angle, the associated losses have not been considered. These losses can be predicted to an acceptable degree of accuracy using 3-D CFD predictions. Using the automatic grid generation technique described in Section 5.3, a parametric study was conducted by varying the radius ratio and area ratio of the passage in series of simulations. Four new passage designs were analysed, with radius ratio/area ratio varying from 0.7 to 2.1. The meridional passage shapes are shown in Figure 7.3, each having a common outlet plane (impeller inlet plane). The implication of radius ratio on passage length can be seen, and in the case of the ‘d3’ design this leads to an unacceptably large VIGV system which would not only lead to high inlet losses, but also excessive manufacturing costs.

The results of the simulations are presented in Tables 7.1 and 7.2 for the range of geometries shown in Figure 7.3. The data are presented both in terms of a fixed vane setting angle (varying outlet flow angle) in Table 7.1, and a fixed outlet flow angle (varying vane setting angle) in Table 7.2 which allows a more meaningful comparison of the passage performance. (Pressure loss coefficient was not used to compare the designs in this investigation, as the varying inlet areas of the passage shapes resulted in a varying inlet static pressure. This in turn would result in the calculation of misleading loss coefficients.)
Table 7.1 Vane Setting Angle = 60°

Table 7.2 Average Outlet Swirl Angle = 60°

It can be seen that although they generate the most swirl, the d1 and d3 designs which have a high radius ratio/area ratio, also generate the highest pressure loss. This is mainly due to viscous losses caused by the increased length of the inlet passage at higher radius ratios, and is also due to the degree of separation that occurs on the hub wall as the passage converges. When presented in terms of a fixed outlet swirl angle, it can be seen that the c12 design, which has a much lower radius ratio/area ratio, is in fact the best inlet passage design based upon these design constraints.

7.2.1 Re-designed passage geometry

Although a spherical section hub and shroud is required in order to minimise blade clearance at all setting angles, it necessitates a converging section passage to the impeller inlet plane. The separation pocket seen in the meridional view in Figure 6.9 was due to the toroidal shape of the passage, which resulted in a boundary layer detachment. The CFD analysis has demonstrated that the swan-neck design may not necessarily be the most efficient method of reducing the flow area. In an attempt to eliminate the separation pocket and reduce the passage losses, a conical section hub was investigated in place of the converging spherical section, shown diagrammatically in Figure 7.4.
Although the magnitude of the separation pocket was reduced, the new hub did not appear to affect the overall system loss at all. It is possible, however, that the implications of the improvement in flow regime in terms of total pressure loss were not accurately predicted, as the technique validation had already shown that this was the case in some of the previous simulations.

The overall outcome of the investigations into re-designing the passage shape is that the existing geometry is in fact probably the optimum shape if vane hub and tip clearance is to be minimised. If a different impeller inlet diameter is used, the technique presented here will enable a new optimum inlet passage geometry to be determined rapidly and accurately as long as its parametric design remains the same.
7.3 Vane Design

The existing VIGV system employs 13 untwisted flat plate vanes, each with a pitch/chord ratio of 1. Without modifying the existing passage shape or vane spindle housings and actuating mechanism, the scope for research into new configurations entailed examination of parameters such as the vane chord and pitch, and the shape and type of vane.

In addition to the mechanical constraints of the VIGV assembly, manufacturing constraints were also considered so that it would be possible for a new prototype set of vanes to be manufactured, installed, and experimentally tested. The net outcome of these constraints was that the vane must be made from flat plate mild steel, and no profiling other than simple leading and trailing edge chamfers would be possible.

Relative assessments of various inlet guide vane designs are presented in this and subsequent chapters at a setting angle of +60°. The designs were assessed by examination of the average deflection produced at the impeller inlet plane and the total pressure loss coefficient. As both of these aerodynamic parameters varied for each design, a means of direct comparison of the relative merits of each design was required.

The equivalence curve shown in Figure 7.5 was derived from the relationships between inlet prewhirl and loss on the overall stage performance, shown below. Comparison of the implications of the various inlet guide vane designs on the stage performance was therefore possible using this relationship between prewhirl and loss.

Effect of prewhirl: 

\[ P_{03} = P_{01} \left( \frac{\eta \Delta T}{T_{01}} + 1 \right)^{\frac{C_p}{\gamma - 1}} \]

where: \( \Delta T = (u_2 V_{w2} - u_1 V_{w1})/C_p \)

Effect of inlet loss coefficient: 

\[ P_{03} = R_C (P_{01} - (Loss Coef)(P_{01} - P_1)) \]

Nominal stage performance parameters were derived for use in these calculations as follows: \( P_{01} = 101235 \text{Pa}, \eta = 0.8, T_{01} = 298 \text{K}, u_2 = 329.9 \text{m/s}, u_1 = 157.4 \text{m/s}, V_{w2} = 306.3 \text{m/s}, R_C = 2, P_1 = 99318 \text{Pa} \)
7.3.1 Pitch/Chord Study

The use of well known 2-D blade performance prediction rules such as Howell’s correlation are inapplicable at high incidence angles, therefore selection of an optimum flat plate vane design depended mainly on empirical data from successive CFD simulations. Using the computational technique, the effect of pitch/chord ratio of flat plate vanes based on the existing shape was assessed. The implications of varying both pitch (by varying the number of vanes) and chord (by varying the total arc angle of the vane) were investigated separately.
7.3.1.1 Varying Chord

Vane chords of 17°, 22°, 27°, and 32° (shown in Figure 7.6) were selected for analysis. A vane chord greater than 32° was not considered as the torque induced on such a vane by the pressure difference across its faces would be unacceptably high.

![Diagram of vane chords](image)

**Fig. 7.6 Varying flat plate vane chord (17° to 32°)**

The overall results of this investigation are shown in Figure 7.7 (the system was modelled with 13 vanes at a setting angle of +60° in each case). It can be seen that the 27° chord vane which had been selected by Swain[50] exhibits a good compromise of minimal deviation and pressure loss at the 60° setting angle. When compared with the equivalence curve in Figure 7.5, the gradient of the trend depicted in Figure 7.7 shows that the 27° is in fact the optimum vane chord. This means that as the vane chord is increased further, there would be a net detrimental effect on the stage performance as the gradient of the curve becomes steeper than that in Figure 7.5.
7.3.1.2 Varying Pitch

The pitch/chord ratio can also be varied by changing the pitch (or number of vanes). A reduction in the pitch/chord ratio by reducing the number of vanes (and hence spindles and housings) is more attractive to the manufacturer than reducing the chord of the vanes as the potential savings in manufacturing cost are much greater. Odd numbers (from 5 to 17) of $27^\circ$ chord flat plate vanes were analysed at a setting angle of $+60^\circ$.

Figure 7.8 shows the effect of varying pitch on the overall VIGV system performance. Based on the equivalence curve in Figure 7.5, the trend suggests that 13 to 17 vanes would provide the optimum pitch.
The aim of the pitch/chord study was to determine whether or not the number and the chord of the vanes could be adjusted to deliver an improved VIGV system performance. By reducing the number of vanes or reducing the chord, the total frictional losses across the vane surfaces is reduced at the expense of fluid deflection. The study described here has examined the effect of varying either pitch or chord separately, and has determined that the existing vane configuration in fact delivers the optimal performance. However, rather than considering the two parameters separately, the results from the study should be used to consider the effect of the combined pitch/chord ratio. This means that the performance of the existing vane configuration could also be achieved using another vane configuration with the same pitch/chord ratio.

A pitch/chord ratio of 1 was selected for the design of the existing vanes, this means that when set at ± 90°, the vanes will fully close off the flow passage with no overlap between adjacent vanes. With this in mind, it can be seen that the following configurations also share a pitch/chord ratio of approximately 1: 17 x 21° chord vanes, 15 x 24° chord vanes, 11 x 32° chord vanes. As mentioned earlier, it is...
desirable to reduce the number of vanes in such a machine in order to reduce manufacturing costs, so the 32° chord vane configuration was selected for further analysis.

As forecast by the pitch/chord ratio study detailed above, Figure 7.9 confirms that the performance of a vane configuration consisting of 11 vanes of 32° chord is comparable with the original 13 x 27° chord vanes.

![Performance comparison of pitch/chord=1 vane configurations](image)

The overall performance of the two configurations is comparable as increasing the vane chord compensates for reducing the number of vanes, both in terms of losses and average deflection. If the increased torque to which the longer chord vanes will be subjected is within an acceptable limit, the 11 x 32° chord configuration is a very suitable alternative, and represents an advantage in reduced manufacturing costs.
7.3.2 Vane Shape Study

The existing vane shape is of constant pitch/chord ratio across its span, which in an annular cascade means that its leading and trailing edges diverge from the hub to the shroud. The edges are in fact radial lines sharing the same centre-point, which lies at the point where the axes of vane rotation coincides with the axis of symmetry of the passage, as shown in Figure 5.2 earlier. Constant chord vanes (increasing pitch/chord ratio from hub to tip) are often used in annular cascades, and could also be used in this case.

A comparison of the geometries of the constant chord, and constant pitch/chord ratio vanes is shown in the meridional passage view in Figure 7.10. Two possible constant chord vane configurations are proposed here, Vane A has a 27° chord at the hub, whilst Vane B has a 27° chord at the shroud. The vanes were modelled merely as an adaptation of the existing vanes a reduced shroud chord (Vane A), or an extended hub chord (Vane B). A significant difference was anticipated between the aerodynamic performance of constant chord and constant pitch/chord vanes due to their geometrical configuration at high setting angles. An implication of using constant chord vanes in an annular cascade is that the orthogonal clearance length between adjacent vanes varies considerably across the span when positioned at high setting angles. This varying clearance occurs due to the increasing pitch between the vanes from hub to tip, as illustrated in Figure 7.11.

Since the vane could no longer be defined by a common chord angle at its tip and hub, the geometries were generated using a modified version of the spreadsheet described in Section 5.3. The resulting vane configurations are shown in the computational meshes in Figure 7.12.
Chapter Seven: VIGV System Development

Fig. 7.10 Meridional view of constant chord vane configurations

Fig. 7.11 Illustration of orthogonal clearance between adjacent vanes
Fig. 7.12 Computational meshes

(a) Original constant pitch/chord vanes

(b) Constant chord Vane (A)

(c) Constant chord Vane (B)
Cascades operating in the mode shown in Figure 7.11 for Vane A will inevitably give rise to high levels of deviation due to the clearance between the vanes. Vane B, however, was not expected to exhibit the same deficit in aerodynamic performance due to its larger chord. The extended chord means that there will be a significant overlap between adjacent vanes at the hub, and a geometrical layout identical to that of the original vanes at the shroud.

The implications of the geometrical configuration shown in Figure 7.11 are highlighted by the overall performance results in Figure 7.13. As expected, Vane A performed poorly as a result of the high levels of deviation caused by its short chord. Vane B, however performed similarly to the original vanes, producing marginally more swirl with slightly higher losses. Comparison of these results with the equivalence curve in Figure 7.5 indicates that Vane B would in fact result in a improved stage performance with no extra manufacturing or assembly costs.

![Fig. 7.13 Performance comparison of constant chord and original vanes](image-url)
The increase in swirl generated by Vane B is attributable to the elimination of the separation pocket on the suction surface of the vanes across the lower half of the span, Figure 7.14. The resulting deviation near the hub is therefore significantly reduced, as shown in Figure 7.15, leading to an increase in the average outlet swirl angle. (Figures 7.14 and 7.15 are shown at a vane setting angle of +60°).

The results of this investigation have shown that Vane B could be a suitable alternative to the original constant pitch/chord vanes if increased turning is favoured at the expense of increased loss.

7.3.3 Vane Type Study

As a result of the research into vane design presented so far, it has been re-iterated that the most significant implication of using flat plate vanes at high setting angles is the inherent incidence loss. It would appear therefore that the most appropriate solution to this problem is the use of profiled vanes. Sanz et al\[36] reported their findings on a non-accelerating guide vane cascade that could operate over an inlet flow angle range of 60° using profiled vanes. Although the loss and deviation exhibited by the vanes at high setting angles were acceptably low, at 0° setting (or incidence) angle the loss was approximately double that determined from the analyses of the existing flat plate VIGV's. This fact, combined with the manufacturing constraints outlined earlier in Section 7.3, highlights the need to find an alternative solution to this problem.

Another simple approach to reducing the incidence loss caused by flat plate vanes at high setting angles is to use flat plate tandem vanes (with a variable trailing edge flap), Figure 7.16. These vanes consist of a fixed axially-aligned leading section, about which a trailing section pivots, thus minimising incidence loss at any setting angle. Okiishi et al\[38] analysed the performance of an annular cascade of such vanes operating as inlet guide vanes to an axial compressor. Although a sharp increase in loss was observed at setting angles higher than 35°, the losses at 0° were comparable with those of the original flat plate VIGV's.
Chapter Seven: VIGV System Development

Fig. 7.14 Near-hub absolute velocity vector contours (Vane B)

Fig. 7.15 Velocity vectors at 10% span (Vane B)
The results of the tests are presented in Chapter 2, and were sufficiently encouraging to prompt further investigation into the performance of the VIGV system incorporating tandem vanes.
8.1 Introduction

The objective of the vane re-design study was to optimise the VIGV system within the imposed manufacturing constraints. The existing experimental and numerical test techniques could be used to fully evaluate the potential VIGV system performance improvement using tandem vanes. The numerical technique was used as a 'postdiction' tool in the analysis of the flat plate inlet guide vanes, i.e. the aerodynamic performance was predicted after it had been experimentally determined. In the case of the tandem vanes, however, the now-proven numerical technique could be used as a performance prediction tool to evaluate potential designs and select the optimum one for experimental performance confirmation.

8.2 Tandem Vane Design

As mentioned in the previous chapter, the new vane design had to comply with the manufacturing and installation constraints. Again, the vanes had to be made from flat plate mild steel, and no profiling other than simple leading and trailing edge chamfers would be possible. An equally significant design constraint was that the existing VIGV system assembly must be used to accommodate the new vanes with no permanent modification. This meant that the existing vane spindles must be used and that the existing hub and shroud passage shape would dictate the vane inner and outer profile.

The tandem vane concept shown in the previous chapter, which actually consist of two blade rows, was used as a basis for the development of the tandem inlet guide
vanes. In operation, the leading section of the vane remains in fixed alignment with the axial inlet flow, which means that it must be rigidly attached to the hub and shroud walls. The trailing section of the vane must pivot using the existing assembly, and its leading edge must lie on the axis of vane rotation. The axial clearance between its leading edge and the trailing edge of the leading section must be minimised to prevent flow leakage when the vanes are set at high setting angles.

This means that as well as minimising the axial clearance between the vanes, the leading and trailing edges must be so designed that there is sufficient freedom of movement to allow the trailing vane to be rotated through high setting angles. Although these requirements would ideally be satisfied using the circular profiled leading and trailing edges shown in Figure 7.15, this was not possible due to the manufacturing constraints listed above. Instead, the geometrical arrangement shown in Figure 8.1 using straight chamfers would have to be used. This design offered sufficient rotational freedom for the trailing vane to be pivoted by ±60°, with a minimal axial clearance between the leading and trailing vanes. Using this configuration, the axial clearance between the two vanes is 1mm with the trailing vane at 0° setting angle.

![Diagram of tandem vane configuration using straight chamfers](image)

Fig. 8.1 Tandem vane configuration using straight chamfers

The chamfer applied to the trailing edge of the original flat plate vanes would also be suitable for the trailing edge of the trailing tandem vane. However, since the leading edge of the leading tandem vane would no longer be required to operate over a range of incidence angles, a sharper chamfer could be applied in order to reduce incidence.
losses further. A comparison of the sectional views of the original flat plate vanes and the new tandem vane concept is shown in Figure 8.2.

Based on the assumption that the tandem vanes will perform similarly to the original flat plate vanes, it was felt the chord of the trailing tandem vane should be similar to the original. Since the leading vane has no effect on the induced deflection, it was considered that only a short chord would be necessary to achieve the desired effect of reducing the incidence loss. A chord angle of 10° was selected as this provided a sufficient chord length at the vane tip to project forward of the spindle mechanism, as shown in Figure 8.3. This was necessary so that the leading vane could be attached to the stationary passage shroud wall rather than the moving spindle mechanism. The trailing vane would be the same as the existing flat plate vanes, but with the leading 5° of chord removed so that the vane rotational centreline becomes the new leading edge of the trailing vane.

Since the numerical predictions had to be conducted before any experimental testing could take place, the next stage in the development of the tandem vanes was to implement the new vane profile and configuration into the existing numerical technique.
8.2.1 Numerical Technique

The geometry definition technique outlined in Section 5.3 was developed from the outset so that it could be easily modified to accommodate new vane designs. Modification of the spreadsheet to produce the tandem vane profile was possible if the tandem vanes were assumed to be solid, i.e. no gap between the leading and trailing vanes. The necessary changes were then as follows:

1. Extending the vane leading edge forward (from $-5^\circ$ from the vane centreline to $-10^\circ$ for the leading edge of the leading vane)

2. Extending or reducing the total vane chord according to the particular trailing vane chord being investigated.

3. Fixing the leading $10^\circ$ of vane chord to a setting angle of $0^\circ$ so that it remained in this axial position regardless of the overall setting angle input by the user.
Chapter Eight: Tandem Vane Development

The graphical output from the modified spreadsheet is shown in Figure 8.4 (the trailing vane chord is 22° in this case). In the position shown (setting angle = 0°), the mid-span profile of the tandem vane appears as a single solid vane shown in Figure 8.5 a. Profiles at +30° and +60° are shown in Figures 8.5 b and c respectively, where it can be seen that no axial clearance between the two vanes is present in the numerical model.

Fig. 8.4 Tandem vane spreadsheet graphical output

(a) Setting angle = 0°  (b) Setting angle = +30°  (c) Setting angle = +60°

Fig. 8.5 Mid-span section through tandem vane

The geometrical accuracy of the numerical vane model was obviously slightly compromised by the absence of the clearance between the vanes. Despite this, accurately modelling the aerodynamic effect of the clearance was not anticipated to be sufficiently necessary to warrant a total change of the proven geometry, mesh generation, and solving techniques.
Although the vane design had been modified, the overall geometrical configuration and size of the computational domain remained unchanged. Only minor changes were therefore required to the existing grid topology templates that had been created for the original flat plate vanes. Changes in the vane hub and tip clearance were accounted for by 'blocking-off' additional cells at both the hub and tip of the leading vane, and in the area of the trailing vane stem at the spindle. A view of the tandem vane computational mesh (with an additional rotated mesh) is shown in Figure 8.6 a, b, and c at 0°, +30° and +60° respectively.

In order to ensure consistency of prediction accuracy so that the tandem vanes could be fairly compared with the original vanes, the boundary conditions and solving technique were identical to those used in the original flat plate vane simulations.

8.2.2 Trailing Vane Chord Selection

Since the parametric shape of the tandem vane had been outlined, an optimal trailing vane chord was the only remaining geometric variable to be determined. The most appropriate technique to determine this chord was a computational pitch/chord study similar to that conducted on the original flat plate vanes. Although the configuration selected for experimental testing must have 13 vanes so that the existing VIGV assembly could be used, a pitch study was conducted as well as a chord study for completeness of the research.
Fig. 8.6 Tandem vane computational meshes
8.2.2.1 Varying Chord

Since the chord of the leading tandem vane was fixed at 10° and the trailing vane was the only moving part, the system adopted for measuring the tandem vane chord was to measure the chord of the trailing vane only. Vane chords of 17°, 22° and 27° (shown in Figure 8.7) were selected for analysis at a setting angle of +60°.

![Fig. 8.7 Varying tandem vane chord (17° to 27°)]

The results of the varying chord study are presented in Figure 8.8 along with those from the original flat plate vane chord study. The potential performance improvement offered by the tandem vanes is immediately apparent from the results of the chord study. For any particular vane chord angle, the average outlet swirl produced by the tandem vanes is comparable with the flat plate vanes, whilst the pressure loss is reduced to less than 50%. Comparison of the tandem vane performance trend with the equivalence curve in Figure 7.5 suggests that a vane chord of 27° would provide the optimal performance.
8.2.2.2 Varying Pitch

Again, odd numbers of vanes from 5 to 17 were selected for analysis at a setting angle of +60°. Based on the outcome of the chord study, a tandem vane chord of 22° was selected for comparison with the flat plate vanes of 27° chord. The results of the pitch study are presented in Figure 8.9, where the trends depict a similar performance comparison to Figure 8.8 above. Of particular note is the similarity in overall performance between 13 and 17 tandem vanes. This suggests that any deviation at the vanes has been sufficiently reduced with 13 tandem vanes that the addition of more vanes has little effect on the overall performance. Also, the increase in viscous losses associated with the increase in wetted surface area caused by the greater number of vanes is balanced by the reduction in loss due to flow separation at the vanes.
The pitch chord study strongly supported the decision to investigate in detail the potential performance benefits of tandem vanes. The objective of the study was, however, to determine a suitable tandem vane chord angle for further experimental investigation. The trends in the previous two figures show that the 22° chord and 27° chord tandem vanes are both potential candidates. Although the 27° chord vane appears to perform better than the smaller chord, it was felt that the torque induced in the vane stem could be too high for the vane to be used in possible future full-scale testing. Both the 27° tandem vane and the original flat plate vanes share the same chord, but in the case of the tandem vane, the whole moving chord is downstream of the vane rotational centreline, and the vane stem is half the length of the original vanes. This means that the torque in the vane stem resulting from the pressure differential across the vane at high setting angles will be considerably higher than that in the original flat plate vanes.

The 22° chord tandem vane was therefore selected for further numerical investigation, and ultimately experimental evaluation.
8.3 Tandem (22° Chord) Vane Computational Performance

Before any new vanes were manufactured for experimental testing, a more detailed computational analysis of the 22° chord tandem vanes was conducted. So far, a comparative assessment of the tandem vane performance had been conducted at a setting angle of +60° as this was clearly the most demanding vane operating configuration. However, the performance of the VIGV system at lower setting angles should also be examined, as it must not be compromised at the expense of any improvement at +60°.

The flow visualisations presented in this section have been arranged identically to those in Section 6.2 for direct comparison of the two vane designs.

8.3.1 Vane Performance

Mid-span velocity vector plots at setting angles of 0°, +30°, and +60° are shown in Figures 8.10 a, b, and c respectively. Deviation at this radial plane is slightly higher than the flat plate vanes due to the decrease in effective vane chord. Leading edge separation has been significantly reduced at all setting angles by the axial alignment of the leading vane section, and by the re-designed leading edge chamfer. The subsequent separation from the suction surface of the vane at the higher setting angles has also diminished as a result of the vane re-design.

Figures 8.11 a, b, and c show the velocity vector plots around the vane at a setting angle of +60° at 10% span, 50% span, and 90% span respectively. Although flow deviation and separation within the passage is again very prominent near the hub, a marked improvement in the flow regime is evident. Towards the vane tip, separation is barely identifiable from the computational predictions.

Although the magnitude of the deviation has increased slightly, the velocity vector plots have demonstrated a considerable improvement in the flow regime around the inlet guide vanes.
Fig. 8.10 Mid-span velocity vectors

(a) Setting angle = 0°

(b) Setting angle = +30°

(c) Setting angle = +60°
Fig 8.11 Velocity vectors across span at setting angle = +60°
8.3.2 Overall Performance

Near-hub absolute velocity contour plots at setting angles of 0°, +30°, and +60° are shown in Figures 8.12 a, b, and c respectively. As described in Figures 8.10 and 8.11, the magnitude of the separation pocket on the suction surface of the vanes at high setting angles has significantly diminished. Despite the decrease in separation, dissipation of the vane wake through the passage is similar to that of the flat plate vanes.

Absolute velocity contours at 50% span and 90% span at a vane setting angle of +60° are shown in Figures 8.13 a and b respectively. Again, the merits of the tandem vane design are apparent in the absence of any noticeable separation across the vane span. The increase in deviation can, however, be seen in the velocity contours by the angle of the vane wake as it leaves the trailing edge of the vanes.

8.3.3 Aerodynamic Losses

The overall performance of the tandem vanes is compared with that of the original flat plate vanes in Figure 8.14. The predicted performance improvement at any setting angle is clearly demonstrated in the trends shown in this figure. With the vanes set at their axial (0°) position, any additional viscous loss caused by the increase in total surface area of the tandem vanes is compensated by the reduction in incidence loss from the re-designed leading edge chamfer. This effect was further investigated by re-modelling the tandem vanes at the 0° setting angle with the leading edge chamfer used from the flat plate vanes, the results of which showed a noticeable increase in the pressure loss coefficient. At higher setting angles, the reduction in the severity of the separation pocket results in a dramatic decrease in the overall VIGV system losses. The reduction in the effective vane chord is also noticeable as a slight reduction in the average outlet swirl angle as the vanes approach the +60° setting angle.
(a) Setting angle = 0°  (b) Setting angle = +30°  (c) Setting angle = +60°

Fig. 8.12 Near-hub absolute velocity contours
Fig. 8.13 Velocity contours across span at setting angle = +60°

(a) 50% span

(b) 90% span
Chapter Eight: Tandem Vane Development

The numerical technique had forecast a potentially rewarding improvement in the overall performance of the VIGV system if tandem inlet guide vanes were used in place of the original flat plate vanes. Since the tandem vane concept had been developed in accordance with the design constraints outlined earlier, a new set of vanes could be manufactured and experimentally tested so that the predictions could be validated.
8.4 Vane Manufacture and Installation

The tandem vanes were laser-cut from 6mm mild steel plate as per the profile design in Figure 8.15. Leading and trailing edge chamfers and the locating holes in the vane stem were subsequently added, and a smooth surface finish was ensured on all of the vane faces.

The trailing vanes were installed using the existing spindle mechanisms, whilst the leading vanes were aligned at 0° setting angle and glued in place at the hub and shroud. Shims were placed between the leading and trailing sections during assembly to ensure an axial clearance of 1mm so that the trailing vane would be free to rotate up to angles of ±60°. The VIGV assembly was then re-installed complete with instrumentation exactly as it was in the original tests.
8.5 Experimental Results

Again, experimental tests were conducted at mass flow rates of 0.5kg/s and 1kg/s with the tandem inlet guide vanes set at angles of 0°, +30°, and +60°. Due to time limitations, experimental testing of the tandem vanes was only conducted at the outlet plane rather than the upstream plane. Based on the outcome of the original testing, it was felt that sufficient flow field data could be obtained from this plane to conduct an adequate technique validation and provide an overall performance assessment of the new vanes.

Figure 8.16 shows the average deflection across the range of setting angles for both the flat plate and tandem vanes. The trends are comparable apart from the 'zero error', where approximately 3° of swirl is generated from vanes which should be set at 0°. The tandem vanes produce marginally less swirl at higher setting angles due to their shorter chord.

Fig. 8.16 Experimental deflection at VIGV system outlet
Figures 8.17 and 8.18 show the radial and circumferential distributions of deflection and velocity. Comparison with the results from the flat plate vanes in Figures 4.7 and 4.8 reveals that the vane re-design has little effect on the outlet flow field.

![Graph showing radial distribution of deflection and velocity at VIGV system outlet](image)

**Fig. 8.17** Radial distribution of deflection and velocity at VIGV system outlet

![Graph showing radial distribution of deflection and velocity at VIGV system outlet](image)

**Fig. 8.18** Radial distribution of deflection and velocity at VIGV system outlet
By far the most noticeable difference in performance between the two types of vanes is the overall total pressure loss coefficient through the VIGV system. This is plotted against average outlet swirl angle in Figure 8.19 to provide an overall performance comparison of the two vane designs. As shown in Figure 8.16, the deflection produced by the two types of vanes is comparable, the main difference being the clear decrease in pressure loss coefficient at the +60° setting angle. The apparent increase in loss at the lower setting angles was however, both unexpected and extremely undesirable. Consequently, it raises doubts about the validity of the experimental tests.

![Graph showing overall experimental performance of tandem and flat plate vanes](image)

**Fig. 8.19** Overall experimental performance of tandem and flat plate vanes

### 8.6 Validation of Experimental and Computational Results

Although minor modifications to both the experimental facility and the numerical technique had been necessary in order to accommodate the new tandem vanes, they had on the whole remained unchanged so that a fair comparison could be made between the two vane designs. Despite this standardisation of investigation techniques, it was still felt that the techniques should be re-validated for the tandem vane case.
Absolute velocity, swirl angle and pressure loss coefficient at the outlet plane were again compared in the validation process.

8.6.1 Absolute Velocity

Figure 8.20 shows the pitch-wise averaged variation of absolute velocity at the outlet plane across the span for vane setting angles of $0^\circ$, $+30^\circ$, and $+60^\circ$. An acceptable correlation is exhibited on the whole, although the velocities tend to be slightly under-predicted towards the hub at each setting angle.

8.6.2 Swirl Angle

Figure 8.21 shows that the computational and experimental trends of pitch-wise averaged swirl angle are in general agreement. The actual difference in results between the two sets of results is however somewhat questionable, particularly at the higher setting angles. The swirl appears on the whole to be under-predicted in each case, with the difference peaking at more than $10^\circ$ at the mid-span position for the $+60^\circ$ setting angle.

Closer examination of the computational technique revealed no apparent flaws or potential sources of error other than the assumption that the vanes were solid. If the axial clearance between the vanes had been modelled accurately, the static pressure difference across the two surfaces of the vane would draw air through the gap towards the suction surface, resulting in a further decrease in the predicted swirl. Comparison of the flat plate and tandem vane experimental swirl angle distribution suggests that the two types of vane provide a very similar level of deflection. This suggestion is somewhat contradicted by the results presented in the previous section as the tandem vane performance prediction clearly shows a higher level of deviation than the flat plate vanes as a result of the shorter effective vane chord. Consideration of the 2-D aerodynamic implications of the geometry of the two annular cascades also suggests that the deviation should be higher for the tandem vanes. An unwrapped view of the two geometries is shown in Figure 8.22, where the increase in orthogonal clearance between the tandem vanes at a setting angle of $+60^\circ$ can be seen.
Chapter Eight: Tandem Vane Development

Fig. 8.20 Spanwise experimental and theoretical velocity profiles

Fig. 8.21 Spanwise experimental and theoretical deflection profiles
The outcome of these considerations was that the most likely source of the discrepancy was the experimental technique. The $0^\circ$ setting angle experimental swirl results shown in Figure 8.21 suggest that the tandem vanes were in fact misaligned either during installation or during testing. Following the experimental testing, the test rig was again stripped down so that the alignment of the vanes could be re-assessed. The alignment error varied from vane to vane and was found to be between $0^\circ$ and $+3^\circ$, thus contributing to the error in Figure 8.21.
8.6.3 Pressure Loss Coefficient

The computational results in Figure 8.23 show that the tandem vanes reduce the pressure loss at higher setting angles at the expense of deflection. However, the experimental results suggest that deflection is only very marginally compromised. Although a reduction in pressure loss coefficient is evident in the experimental results, it is not as distinct as the computational results.

![Graph showing pressure loss coefficient and average outlet swirl angle](image)

Fig. 8.23 Experimental and theoretical overall performance comparison

Despite these discrepancies in the results at the VIGV system outlet, both techniques have demonstrated that the tandem vanes offer a potentially rewarding increase in the overall stage efficiency.
CHAPTER 9

STAGE PERFORMANCE

9.1 Introduction

The research presented so far has focused on the development of the inlet guide vane system by examining its performance in isolation. This was a far more efficient development technique than repeatedly examining the full stage as it significantly reduced the time and the experimental and computational resources required to conduct the analyses. Although this technique allowed rapid development of the design, the effect of the VIGV system on the overall compressor stage performance was ultimately the main concern.

The relative merits of each potential VIGV system design have so far been determined through the swirl/loss coefficient and through a qualitative assessment of the aerodynamic performance within the inlet passage. The investigations have been made under the assumption that the swirl/loss coefficient provided an accurate view of the relative performance of the VIGV systems in terms of their effect on the overall stage performance. In reality, the implications of the nature of the VIGV system outlet flow field should also be considered, as its interaction with the inducer leading edge could significantly affect the overall stage performance.

It was therefore considered necessary to examine the implications of the 3-D flow interaction at the inducer leading edge by conducting full stage analyses of the existing and new VIGV system designs. Examination of the flow interaction was possible using a simple mathematical approach and more detailed 3-D computational predictions. The first technique was based on the calculation of velocity triangles from the VIGV system outlet flow field and the impeller geometrical and operational
Chapter Nine: Stage Performance

characteristics. The 3-D computational technique, described in Section 5.4, was considerably more versatile and informative than using velocity triangles.

Validation of the computational technique was possible using overall stage performance data from experimental tests conducted on the complete stage (compressor, diffuser and flat plate VIGV system). The applicability of the swirl/loss coefficient in assessing the overall performance implications of the VIGV system could also be examined by implementing it in the 1-D performance prediction technique described in Section 5.5.

9.2 VIGV — Impeller Interaction

As described in Section 2.1.2, variable inlet guide vane systems can extend the stable operating range of a centrifugal compressor by regulating the incidence onto the leading edge of the inducer. The magnitude and direction of the incidence angle is dictated by the gas inlet relative velocity angle ([x]), and the impeller blade inlet angle. Ideally, the VIGV system operating schedule will result in the velocity triangles shown in Figure 9.1. Three compressor mass flow rates corresponding to VIGV setting angles of 0°, +30°, and +60° are shown here (assuming zero VIGV deviation), with the impeller operating at a constant rotational speed. Increasing the prewhirl as the mass flow rate reduces results in a constant relative inlet gas angle ([x]). This means that an optimal level of incidence onto the inducer can be maintained even at low flow rates, which reduces the susceptibility of the impeller to stall near its leading edge.

![Impeller inlet velocity triangles at varying VIGV setting angles](image_url)

Fig. 9.1 Impeller inlet velocity triangles at varying VIGV setting angles
Using a combination of data available from the VIGV system tests and stage performance testing, the actual inlet velocity triangles can be determined at each VIGV setting angle. The relationship between VIGV system setting angle and compressor mass flow rate is taken from the actual stage operating schedule. Absolute velocity vectors deduced from the VIGV system testing can then be combined with typical mass flow rates and the impeller rotational speed from the stage testing to produce the inlet velocity triangles and hence the inlet relative gas angle. Figures 9.2 a, b, and c show the spanwise distributions of inlet relative gas angle at VIGV setting angles of 0°, +30°, and +60° respectively.

With the inlet guide vanes set at their axial (0° setting angle) position for the highest mass flow rates, no swirl distribution exists at the inducer leading edge. The aerodynamic effect of the VIGV system passage on the distribution of velocities does however influence the inlet velocity triangles and hence the incidence matching. Figure 9.2a shows that the incidence is very well matched at the maximum flow rate across the span. As the flow rate is decreased without adjusting the inlet guide vanes, the increase in incidence can clearly be seen. As the guide vanes are imparting no swirl in this configuration, the difference in vane design has no effect on the inlet velocity triangles.

At a guide vane setting angle of +30°, Figure 9.2b, the incidence is still fairly well matched despite the further decrease in mass flow rate. The increase in swirl angle near the hub using the tandem vanes is evident as a slight decrease in the relative flow angle.

If inducer stall occurs at a specific value of incidence, increasing the prewhirl will move this incidence to lower mass flow rates. Although the incidence across the span at the +60° setting angle is significantly increased, this is not unreasonable as this operating configuration represents a turndown in mass flow rate of approximately 63%. The distribution shown in Figure 9.2c indicates that the inlet guide vanes are imparting insufficient swirl across the majority of the span. Towards the hub, the relative inlet flow angle is too low, not as a result of high inlet swirl, but high absolute velocity (see Figures 6.1, 6.2, 8.20, 8.21).
The distributions of relative gas inlet angle presented here are sufficiently informative to demonstrate that the VIGV system operating schedule is suitable in terms of impeller incidence regulation. They show that the tandem inlet guide vanes should have a negligible effect on the interaction at the impeller leading edge. Also, it can be seen that further modification of the vane shape to improve the incidence matching at low mass flow rates (high vane setting angles) would have a detrimental effect on the performance at higher mass flow rates.
Chapter Nine: Stage Performance

Fig. 9.2a Impeller inlet flow angles at IGV setting angle = 0°

Fig. 9.2b Impeller inlet flow angles at IGV setting angle = +30°

Fig. 9.2c Impeller inlet flow angles at IGV setting angle = +60°
9.3 Flat Plate Inlet Guide Vane Stage Performance Investigations

The results presented in the previous section have demonstrated the suitability of the VIGV system in terms of impeller incidence matching, but further implications on the stage performance characteristics have not been examined. These implications can be explored using experimental and numerical stage analysis techniques where applicable. A brief description of the compressor stage used in each of the stage investigations is offered in Chapter 3.

The duty considered in this study was a design pressure ratio of 2:1 at a rotational speed of 18050 rpm. Inlet guide vanes were set at 0°, +20°, +40°, and +60° with the diffuser vanes set at angles ranging from 62.2° to 71°. The resulting mass flow rate ranged from 1.88 to 5.2 kg/s, with a peak polytropic efficiency of 85%. Inlet conditions were standardised to 298K and 101.3 kPa. The operating conditions detailed here were applied in both the experimental and numerical studies.

3-D computational predictions were conducted using the technique described in Section 5.4, with boundary conditions based on the operating configurations described above. These boundary conditions were also implemented in the 1-D performance prediction technique described in Section 5.5.

In addition to the computational techniques, experimental performance data was also available for the complete stage incorporating the flat plate VIGV system. These stage tests were carried out using the compressor manufacturer's test facility. Mass flow rate was measured in accordance with BS1042 [55] at the outlet from the compressor discharge ducting. Inlet and outlet pressures and temperatures were measured at four circumferential locations upstream and downstream of the compressor.
9.3.1 3-D Stage Computational Results – Flat Plate Vanes

Although velocity triangles are adequate for examining the flow interaction between the VIGV system and the inducer leading edge, the aerodynamic implications of the interaction reach further downstream into the impeller vane passage. Examining the flow field in detail within a rotating impeller is extremely difficult using conventional experimental instrumentation techniques. CFD, however, has the capability to examine the aerodynamic performance anywhere within the passage in considerable detail with relative ease.

All stream surface plots presented in this section are shown in an ‘unwrapped’ form to enable visualisation of the complex 3-D shape in a 2-D plane.

Detailed examination of the impeller leading edge incidence matching is possible using velocity vector plots. Figures 9.3 a, b, c, and d show the relative velocity vectors on a stream surface near the impeller tip (approximately 90% span) at flow rates from 5.2 kg/s (IGV=0°) to 1.9 kg/s (IGV=60°). As expected, the velocity vectors coincide almost tangentially with the leading edge of the impeller vane until very low mass flow rates are reached.

There is no significant separation at the inducer leading edge in any of the figures, although the development of re-circulation pockets can be seen on the suction surface of the vanes towards the impeller outlet at the low flow rate. This re-circulation results in increased impeller loss and less desirable diffuser inlet conditions.

The aerodynamic behaviour described by the velocity vector plots is also illustrated by the contours of relative Mach number through the impeller. Figures 9.4 a, b, c, and d show the relative Mach number contours at the 90% span stream surface, again at flow rates from 5.2 kg/s (IGV=0°) to 1.9 kg/s (IGV=60°).

High velocity around the intervanes at the high mass flow rate indicates that the impeller would be most likely to choke around this region.
Relative velocity vectors and contour of relative Mach number at mid-span and on a stream surface near the impeller hub (approximately 10% span), are shown for the same operating configurations in Appendix C. Contours of circumferentially mass-averaged relative Mach numbers are also shown in a meridional view for each of the operating configurations. Again, these plots demonstrate the capability of the stage to achieve an extremely large turndown in flow at constant rotational speed.
Fig. 9.3a Relative velocity vectors at 90% span
Flat plate VIGV's @ 0°, Mass flow rate = 5.2 kg/s
Fig. 9.3b Relative velocity vectors at 90% span
Flat plate VIGV's @ +20°, Mass flow rate = 4.3 kg/s
Fig. 9.3c Relative velocity vectors at 90% span
Flat plate VIGV's @ +40°, Mass flow rate = 2.9 kg/s
Fig. 9.3d Relative velocity vectors at 90% span
Flat plate VIGV's @ +60°, Mass flow rate = 1.9 kg/s
Fig. 9.4a Relative Mach number contours at 90% span
Flat plate VIGV's @ 0°, Mass flow rate = 5.2 kg/s
Fig. 9.4b Relative Mach number contours at 90% span
Flat plate VIGV's @ +20°, Mass flow rate = 4.3 kg/s
Fig. 9.4c Relative Mach number contours at 90% span

Flat plate VIGV's @ +40°, Mass flow rate = 2.9 kg/s
Fig. 9.4d Relative Mach number contours at 90% span
Flat plate VIGV's @ +60°, Mass flow rate = 1.9 kg/s
9.3.2 Stage Experimental Results – Flat Plate Vanes

A comparison of the CFD and experimentally measured total-to-total pressure ratio is shown in Figure 9.5 for each of the stage operating configurations. Although the experimental results were measured just downstream of the volute, the volute was not numerically modelled. It was considered in this case where a vaned diffuser was used that the volute would not significantly influence the stage performance characteristics. Good agreement between numerical and experimental results can be seen towards the surge end of each operating curve.

![Fig. 9.5 Experimental and 3-D numerical stage performance predictions](image)

Total-to-total polytropic efficiency is also plotted in Figure 9.5 for each of the stage operating configurations. Underprediction of both pressure ratio and efficiency near the choke end of the curves is evident with good agreement in the surge region. Calculation of the work input at each of these points has also shown good agreement.
between experimental and numerical methods. This suggests that the slight underprediction of efficiency was responsible for the discrepancy in pressure ratio predictions.

9.3.3 1-D Stage Computational Results – Flat Plate Vanes

With the flat plate VIGV swirl/loss model implemented in its algorithm, the 1-D numerical prediction technique was also compared with the experimental stage performance results. A comparison of the overall stage pressure ratio and polytropic efficiency is presented in Figure 9.6.

Since the empirical loss model derived from experimental testing of the VIGV system was used in the numerical technique, excellent correlation can be seen for both pressure ratio and efficiency. These results suggest that the 1-D performance prediction technique could be used with a high degree of confidence to give accurate results for further similar designs based on this configuration.
Chapter Nine: Stage Performance

Fig. 9.6 Experimental and 1-D numerical stage performance predictions

The 1-D numerical technique was a far more efficient method of evaluating the implications of potential VIGV system designs on the overall stage performance. Rather than repeatedly modelling the entire stage using the 3-D numerical technique, modelling of the VIGV system in isolation (not the complete stage) was required at varying setting angles. Assuming the interaction between the VIGV system and the impeller remained similar, the VIGV system swirl and loss characteristics could then be fed into the 1-D technique to conduct the stage performance prediction.
Chapter Nine: Stage Performance

9.4 Tandem Inlet Guide Vane Stage Performance Investigations

Having proven the 3-D numerical stage performance prediction technique, it could be adapted to accommodate the tandem vane design and used with confidence to predict the full implications of the new inlet guide vanes. Identical boundary conditions were applied so that the results could be directly compared with the flat plate inlet guide vane performance predictions. Experimental stage performance data was not available for the tandem inlet guide vanes and no future experimental testing was anticipated, so validation could only be conducted against the results of the 1-D technique. No modifications to the 1-D technique were necessary other than substituting the tandem inlet guide vane loss model for the flat plate model.

9.4.1 3-D Stage Computational Results – Tandem Vanes

As shown in the 1-D interaction study in Section 9.2, the tandem inlet guide vanes have a negligible effect on the impeller aerodynamic performance compared with the flat plate vanes. Relative velocity vector plots at the 90% span stream surface are shown in Figures 9.7 a, b, c, and d for comparison with the equivalent flat plate velocity vectors in Figures 9.3 a, b, c, and d. As expected, no noticeable difference exists between the two inlet guide vane designs in terms of the aerodynamic interaction or flow field through the impeller passage.

A similar story is revealed by the contours of relative Mach number, Figures 9.8 a, b, c, and d, which remain essentially unchanged from those describing the flow with flat plate inlet guide vanes.

Relative velocity vectors at the mid-span and 10% span stream surfaces are shown in Appendix C. The relative Mach number contour plots corresponding to those presented for the flat plate inlet guide vanes can also be found in Appendix C. No difference in aerodynamic performance between the two inlet guide vane designs can be seen in these views.
Fig. 9.7a Relative velocity vectors at 90% span
Tandem VIGV's @ 0°, Mass flow rate = 5.2 kg/s
Fig. 9.7b Relative velocity vectors at 90% span
Tandem VIGV's @ +20°, Mass flow rate = 4.3 kg/s
Fig. 9.7c Relative velocity vectors at 90% span
Tandem VIGV's @ +40°, Mass flow rate = 2.9 kg/s
Fig. 9.7d Relative velocity vectors at 90% span
Tandem VIGV's @ +60°, Mass flow rate = 1.9 kg/s
Fig. 9.8a Relative Mach number contours at 90% span
Tandem VIGV's @ 0°, Mass flow rate = 5.2 kg/s
Fig. 9.8b Relative Mach number contours at 90% span
Tandem VIGV's @ +20°, Mass flow rate = 4.3 kg/s
Fig. 9.8c Relative Mach number contours at 90% span
Tandem VIGV's @ +40°, Mass flow rate = 2.9 kg/s
Fig. 9.8d Relative Mach number contours at 90% span
Tandem VIGV’s @ +60°, Mass flow rate = 1.9 kg/s
The absence of any change in the impeller flow field using the tandem vanes was on the whole expected and desirable, as the inlet guide vane system, impeller and diffuser had originally been designed to complement one another aerodynamically. The intended outcome of using the tandem vanes was to produce a similar impeller inlet flow field with a decrease in the inlet loss to the impeller, and hence an increase in the overall stage pressure ratio and efficiency at low mass flow rates.
9.4.2 1-D Stage Computational Results – Tandem Vanes

The results of the 1-D prediction technique are plotted in Figure 9.9 along with the overall performance characteristics predicted by the 3-D numerical technique. Again, good agreement between the results of the two techniques can be seen, as the tandem vane empirical loss model was used in the 1-D technique.

![Figure 9.9 1-D and 3-D numerical stage performance predictions (tandem vanes)](image)

The mass flow rate boundary conditions selected for the choke points predicted using the 3-D technique were higher than the maximum mass flow rates in the 1-D predictions. Although this means that these points appear as anomalies, if the 1-D performance lines were extended to higher mass flow rates, good correlation would again be exhibited.
9.5 Comparison of flat plate and tandem VIGV stage performance

The lack of experimental stage performance data using the tandem inlet guide vanes meant that numerical prediction data had to be used to compare the two designs. The investigations presented in this chapter have shown that the 1-D numerical technique was the most appropriate prediction technique for evaluating the implications of the inlet guide vane design on the overall stage performance. A comparison of the stage performance predictions using the two VIGV designs is shown in Figure 9.10.

![Fig. 9.10 Comparison of flat plate and tandem inlet guide vane stage performance](image)
As expected, the tandem inlet guide vanes produced no noticeable increase in efficiency or pressure ratio at low setting angles. However, as the mass flow rate is reduced and the guide vane setting angle increased, the improved performance of the tandem vanes becomes particularly apparent as an increase in the stage pressure ratio. At a VIGV setting angle of +60° the increase in pressure ratio is most noticeable, and an associated increase in polytropic efficiency of as much as 3% was predicted.

The effect of the entire VIGV system on the stage performance was also investigated numerically by simple removing the whole system from the inlet to the impeller, and operating the impeller and VVD at the 0° VIGV configuration. Although the impeller inlet velocity profile differed due to the absence of the converging section passage, no change occurred in pressure ratio or efficiency. This suggests that any additional loss caused by the VIGV system inlet ducting is compensated by the velocity profile produced by this geometry.

9.6 Closure

The 1-D and 3-D numerical performance predictions have demonstrated the potential improvement in stage pressure ratio and efficiency offered by the tandem inlet guide vanes. The increase in efficiency results from the decrease in inlet losses caused by the guide vanes at high setting angles. Combined with a slight decrease in the inlet prewhirl generated by the tandem vanes, this increase in efficiency contributes to the increase in stage pressure ratio.
The aerodynamic performance characteristics of an existing centrifugal compressor VIGV design have been analysed. An experimental test facility has been purpose built to accommodate and test the VIGV system using semi-automatically controlled pneumatic instrumentation. The experimental data has been complemented by numerical data from a state of art 3-D computational prediction technique. Together, these techniques have enabled a comprehensive and efficient analysis of the VIGV system, which has led to potential improvements in its aerodynamic design. The overall conclusions drawn from the research are listed below.

- Centrifugal compressor VIGV system testing can be conducted in isolation (without the complete stage), and at low mass flow rates. This means that low pressure ratio blowers that do not require a high electrical power supply can be used to draw air through the system.
- Experimental examination of the distribution of aerodynamic parameters such as swirl, absolute velocity, and total pressure at the axial plane in which the impeller leading edge lies provides sufficient data to determine the relative merits of any particular VIGV system design.
- Using a semi-automatic mesh generation system, high quality CFD meshes can be created efficiently and accurately, and used with confidence to model the performance of an entire centrifugal compressor stage incorporating variable inlet guide vanes and variable vaned diffuser.
- The physical flow field produced at the outlet of the VIGV system can be predicted accurately using CFD, providing that all geometrical aspects of the numerical model such as blade hub and tip clearance have been precisely modelled. Realistic inflow and outflow boundary conditions must also be applied.
• CFD has the capability to predict trends of loss for relative comparison between different VIGV system designs, however an accurate prediction of the magnitude of the loss is more difficult due to approximations used in the numerical loss models.

• Sufficient confidence has been gained in the 3-D numerical performance prediction technique for it to be used as the first step in the design process of new inlet guide vanes.

• Experimental validation should be conducted wherever possible for each new design evaluated to ensure that any assumptions made in the physical or fluid boundary conditions of the model are appropriate.

• The use of a spherical section passage eliminates blade hub and tip clearance with negligible additional system loss. Increasing the radius ratio results in an increase in the average swirl produced by the VIGV system at the expense of additional total pressure loss.

• The optimum pitch/chord ratio of vanes used in this type of passage is approximately 1. If constant chord vanes are used, then this should be the vane type (B).

• If flat plate vanes are used, then they should be either constant pitch/chord vanes or constant chord vanes (in accordance with the recommendation made above) in order to achieve the best overall VIGV system performance.

• Tandem inlet guide vanes will produce similar swirl and loss to the flat plate vanes at low setting angles. At high setting angles however, the tandem vanes produced similar swirl with a significantly reduced loss coefficient compared with the flat plate vanes.

• Tandem vanes produce spanwise and pitchwise distributions of velocity and swirl at the impeller leading edge that are essentially unchanged from those produced by the flat plate vanes.

• The most likely explanation for the decrease in loss using tandem vanes is the reduction in the severity of the separation pocket on the suction surface of the vanes, particularly near the hub.

• Tandem inlet guide vanes with a movable section pitch/chord ratio of 1 would provide similar swirl to the flat plate vanes, with less than half the total pressure loss at high setting angles.
Chapter Ten: Conclusions

- The VIGV operating schedule ensures the highest achievable overall stage efficiency by maintaining desirable levels of incidence across the span of the impeller leading edge at all mass flow rates and setting angles.
- Incidence matching prevents the flow from detaching from the suction surface of the impeller vanes until very low mass flow rates are reached.
- The combination of a variable geometry vaned diffuser and variable inlet guide vanes with a backswept impeller enables flow turndown of as much as 63% from the choke point at constant rotational speed.
- The validity of CFD techniques to predict the performance of such a compressor has been proven, with strong agreement being exhibited with experimental data.
- Having implemented the inlet loss model derived from the VIGV system analyses into the 1-D numerical prediction technique, the technique can be used to provide an extremely rapid and accurate prediction of the implications of the VIGV system performance on the whole stage.
- Although the tandem inlet guide vanes do not noticeably affect the aerodynamic performance of the impeller compared with the flat plate vanes, they result in a rise in the stage pressure ratio at low mass flow rates. This increase is approximately equivalent to the decrease in inlet loss from the tandem vanes multiplied by the original stage pressure ratio.
- The analyses have shown that an increase in polytropic efficiency of as much as 3% is possible at high setting angles using the re-designed tandem inlet guide vanes.

10.1 Contribution

The project has combined state of the art experimental and theoretical analysis techniques to comprehensively analyse and develop a variable inlet guide vane system. The application of CFD to turbomachinery development has until recently been fairly impractical and treated with scepticism. The research conducted in this project has, however, demonstrated its applicability if a structured development procedure is followed. To date, no literature has been found detailing the application of both experimental and theoretical analysis techniques to the design of a centrifugal
compressor variable inlet guide vane system. Tandem vanes have not normally been used in centrifugal compressor variable inlet guide vane systems, but the research has shown that they offer improved performance when compared with the original flat plate vanes. The overall outcome of this research is a usable VIGV design technique for real industrial compressor environments, and confirmation that an acceptable design can be achieved that represents a rewarding improvement in performance. In addition, the following contributions have been made in the field of turbomachinery research (references are made to the publication list in Section 10.3):

- The development of a fast and versatile technique to develop a high quality VIGV CFD mesh for any geometrical configuration (Ref. 1)
- Use of CFD to analyse and develop a centrifugal compressor VIGV system design (Ref. 1).
- Validation of CFD results for varying flow and operational configurations using a purpose built experimental test facility (Ref. 1).
- Implementation of tandem vanes in a centrifugal compressor VIGV system, resulting an improvement in the system performance (Ref. 2 and 4).
- Parametric analysis of the effects of varying inlet passage geometry on the VIGV system performance (Ref. 3).
- Understanding of VIGV to inducer flow interaction to enable improved impeller design.
- Proof of a 3-D numerical stage performance prediction technique for a variable geometry centrifugal compressor stage (Ref. 4).

10.2 Further Work

During the course of the research, a number of areas in which further work could be done to complement the study presented here have arisen. They were not conducted in this research project due to limitations in resources and time.

Further analyses of the tandem vane concept could be conducted, particularly with respect to the $27^\circ$ chord tandem vane as this was predicted to offer improved overall
performance compared with the $22^\circ$ chord. Detailed stress analyses could then be conducted to determine the suitability of the vanes in normal operation. Experimental testing of the VIGV system incorporating these vanes would represent little extra effort as the test facility would readily accommodate such vanes and the existing instrumentation could again be re-calibrated and re-used. The analyses should also be extended to 1-D and 3-D numerical stage performance predictions for completion. Another candidate for these analyses is the constant chord flat plate vane (B), which appears to be worth further evaluation.

It was originally intended that experimental compressor stage testing would be conducted using the tandem inlet guide vanes, however this was not possible within the duration of the research. Experimental stage performance data for either the $22^\circ$ or $27^\circ$ chord tandem vanes, or the constant chord flat plate vane (B), would be invaluable in validating the 1-D and 3-D numerical stage prediction techniques.

10.3 Publications


REFERENCES


References


<table>
<thead>
<tr>
<th>Reference</th>
<th>Description</th>
</tr>
</thead>
</table>
References


Also used but not referenced in this thesis:


DECLARE SUB travio ()
DECLARE SUB probadjust ()
DECLARE SUB hubtoshub ()
DECLARE SUB shrtohub ()
DECLARE SUB setplane ()
DECLARE SUB probreset ()
DECLARE SUB changeplane ()
DECLARE SUB probout ()
DECLARE SUB probin ()
DECLARE SUB move ()
DECLARE SUB adjust ()
DECLARE SUB travop ()
DECLARE SUB probpos ()
DECLARE SUB trav ()
DECLARE SUB io ()
DECLARE SUB start ()
DIM SHARED dat AS STRING
DIM SHARED nam AS STRING
DIM SHARED namnew AS STRING
DIM SHARED head AS STRING
DIM SHARED s AS INTEGER
DIM SHARED pO1 AS SINGLE
DIM SHARED t01 AS SINGLE
DIM SHARED m AS SINGLE
DIM SHARED igvang AS SINGLE
DIM SHARED p AS INTEGER
DIM SHARED ps AS SINGLE
DIM SHARED newpos AS INTEGER
DIM SHARED d AS SINGLE
DIM SHARED plane AS STRING
DIM SHARED pl AS STRING
DIM SHARED newplane AS STRING
DIM SHARED y AS INTEGER
DIM SHARED scale AS INTEGER
DIM SHARED steps AS INTEGER
DIM SHARED hub AS SINGLE
DIM SHARED inc AS SINGLE
DIM SHARED posn AS SINGLE
DIM SHARED newposn AS SINGLE
DIM SHARED delta AS SINGLE
DIM SHARED radius AS SINGLE
DIM SHARED rad AS SINGLE
COLOR 15, 1
p01 = 760: t01 = 22: m = .5
10 CLS
y = 30
REM Opening title screen
PRINT
PRINT
PRINT
PRINT "VIGV Test Rig Data Acquisition and Control Software"
PRINT "Options available:-
PRINT "1) Enter atmospheric conditions and test details"
PRINT "2) Read/write data from transducer to file"
PRINT "3) Traverse unit operations"
PRINT "0) Exit program"
PRINT
INPUT Please make selection: ", s
IF s = 1 THEN CALL start
IF s = 2 THEN CALL io
IF s = 3 THEN CALL travop
IF s = 0 THEN GOTO 100
IF s = 99 THEN GOTO 10
GOTO 10
SUB changeplane

REM Change probe plane by adjusting hub point and scale
OPEN "c:\utils\mech\posn.dat" FOR INPUT AS #3
INPUT #3, p, plane$
CLOSE #3
200 PRINT "Probe is currently in "; plane$; " plane."
PRINT
INPUT "Select traverse plane: (B)lade or (O)utlet "; pl$
IF pl$ = "B" OR pl$ = "b" THEN
    newplane$ = "blade"
    IF newplane$ = plane$ THEN GOTO 250
    IF newplane$ <> plane$ AND newplane$ = "blade" THEN
        plane$ = newplane$
        CALL setplane
        IF p = 1 THEN
            newposn = hub + delta
        ELSE newposn = ((p - 1) * inc) + (inc / 2) + delta
        END IF
        steps = INT(ABS(newposn - posn) * 200)
        CALL probin
        GOTO 250
    END IF
ELSEIF pl$ = 000 OR pl$ = "o" THEN
    newplane$ = "outlet"
    IF newplane$ = plane$ THEN GOTO 250
    IF newplane$ <> plane$ AND newplane$ = "outlet" THEN
        plane$ = newplane$
        CALL setplane
        IF p = 1 THEN
            newposn = hub + delta
        ELSE newposn = ((p - 1) * inc) + (inc / 2) + delta
        END IF
        steps = INT(ABS(newposn - posn) * 200)
        CALL probout
    END IF
ENDIF
END IF
GOTO 250
END IF
GOTO 200
250 PRINT
PRINT *       Probe is set for "; plane$; " plane."
PRINT
INPUT *       Hit return to continue.", r
OPEN "c:\utils\mech\posn.dat" FOR OUTPUT AS #3
PRINT #3, p, plane$
CLOSE #3
END SUB

SUB hubtoshr

REM hub to shroud traverse
PRINT *       Moving probe to hub ...
newpos = 1
IF newpos = p THEN GOTO 350
REM Move probe in
    newposn = hub + delta
    steps = INT(ABS(newposn - posn) * 200)
    CALL probin
350 REM Start traverse
    newpos = 11
    newposn = ((newpos - 1) * inc) + (inc / 2) + delta
    steps = INT(ABS(newposn - posn) * 200)
    y = 30
    z = 0
360 steps = steps - z
PRINT
PRINT *       Set traverse speed using + or - keys whilst traversing,"
PRINT *       and press Esc key to interrupt traverse."
PRINT
INPUT *       Press return to continue", r
FOR z = 1 TO steps
    FOR x = 1 TO y: NEXT
    OUT &H322, 4:
    FOR x = 1 TO y: NEXT
    OUT &H322, 5:
KEY$ = INKEY$
IF KEY$ = CHR$(45) THEN y = y + 10
IF KEY$ = CHR$(61) THEN y = y - 10
IF y < 30 THEN y = 30
IF KEY$ = CHR$(27) THEN GOTO 360
NEXT z
OPEN "C: \utils\mech\posn.dat" FOR OUTPUT AS #3
PRINT #3, newpos, plane
CLOSE #3
posn = newposn
y = 30
END SUB

SUB io
REM Read data from pressure transducer and write to file
CLS
PRINT
PRINT
IF nam <> "" THEN
PRINT " Enter filename for data storage (default = "; nam;");
INPUT "", namnew
IF namnew <> "" THEN
nam = namnew
END IF
ELSE
INPUT " Enter filename for data storage: ", nam
END IF
OPEN "C:\data\" + nam FOR APPEND AS #2
PRINT
INPUT " Enter text header for file: ", head
PRINT #2, head
PRINT #2, pO1, tO1, m
PRINT
410 PRINT " Logging data (hold escape key and press store
button to stop)."
dat = "Logging ..."
OPEN "com1:9600,n,8,1,rb9" FOR INPUT AS #1
PRINT
207
DO
PRINT " "; dat
PRINT #2, dat
INPUT #1, nul, dat, nul
LOOP WHILE INKEY$ <> CHR$(27)
CLOSE #1
INPUT " Enter text comment for file: ", head
IF head = "q" THEN GOTO 420
PRINT #2, head
GOTO 410
420 CLOSE #2
s = 99
END SUB

SUB move

REM Developmental traverse unit control subroutine
y = 35
PRINT
PRINT " If traverse unit is not at mid-span position (50mm), use"
PRINT " < or > keys to move traverse unit forward or backwards,"
PRINT " + or - keys to speed up or slow down,"
PRINT " and press Esc key (twice) to stop."
510 KEY$ = INKEY$
IF KEY$ = CHR$(44) GOTO 520
IF KEY$ = CHR$(46) GOTO 530
IF KEY$ = CHR$(27) GOTO 540
GOTO 510
520 DO
FOR x = 1 TO y: NEXT
OUT &H322, 0: REM sets port C
 FOR x = 1 TO y: NEXT
OUT &H322, 1:
 KEY$ = INKEY$
 IF KEY$ = CHR$(45) THEN y = y + 20
 IF KEY$ = CHR$(61) THEN y = y - 20
 IF y < 30 THEN y = 30
 IF KEY$ = CHR$(46) GOTO 530
 IF KEY$ = CHR$(27) GOTO 510
LOOP
Appendix A: Control Software

REM backwards
OUT &H323, 128:
530 DO
FOR x = 1 TO y: NEXT
OUT &H322, 4:
    FOR x = 1 TO y: NEXT
OUT &H322, 5:
    KEY$ = INKEY$
    IF KEY$ = CHR$(45) THEN y = y + 20
    IF KEY$ = CHR$(61) THEN y = y - 20
    IF y < 30 THEN y = 30
    IF KEY$ = CHR$(44) GOTO 520
    IF KEY$ = CHR$(27) GOTO 510
LOOP
540 y = 30
END SUB

SUB probadjust

REM Manual probe radial position calibration
PRINT
PRINT "Turn off motor drive, and manually wind in probe until hub is"
INPUT "reached. Then turn on motor drive and hit return.", r
steps = 200
CALL probout
END SUB

SUB probin

REM Move probe in
PRINT
PRINT "Moving probe ..."
FOR z = 1 TO steps
    FOR x = 1 TO y: NEXT
    OUT &H322, 0:
    FOR x = 1 TO y: NEXT
    OUT &H322, 1:
    NEXT z
posn = newposn
END SUB
SUB probout

REM Move probe out
PRINT
PRINT " Moving probe ..."
FOR z = 1 TO steps
FOR x = 1 TO y: NEXT
OUT &H322, 4:
FOR x = 1 TO y: NEXT
OUT &H322, 5:
NEXT z
posn = newposn
END SUB

SUB probpos

REM Probe automatic radial positioning and data-logging routine
REM control register = &H323
600 CLS
PRINT
PRINT
605 INPUT "Would you like to data-log whilst traversing ({Y)/N)? ", a$
IF a$ = "Y" OR a$ = "y" OR a$ = "" THEN GOTO 606
IF a$ = "N" OR a$ = "n" THEN GOTO 608
GOTO 605
606 PRINT
IF nam <> "" THEN
PRINT " Enter filename for data storage (default = "; nam; "): ";
INPUT "", namnew
IF namnew <> "" THEN
nam = namnew
Appendix A: Control Software

END IF
ELSE
INPUT * Enter filename for data storage: *, nam
END IF
OPEN "c:\data\" + nam FOR APPEND AS #2
PRINT
INPUT * Enter text header for file (default = Traverse Data): *, head
IF head = "" THEN head = "Traverse Data"
PRINT #2, head
PRINT #2, p01, t01, m
PRINT #2, * Traversing in *, plane$; " plane."
PRINT #2, * Radius Pol-Po2 Dyn_Head Yaw_angle"

608 PRINT
PRINT * Probe is set for *, plane$; " plane."
PRINT
PRINT * Available probe positions: -
PRINT
PRINT * 1) Hub", , hub + radius, "mm"
PRINT * 2) "", , ((1) * inc) + (inc / 2) + radius, "mm"
PRINT * 3) "", , ((2) * inc) + (inc / 2) + radius, "mm"
PRINT * 4) "", , ((3) * inc) + (inc / 2) + radius, "mm"
PRINT * 5) "", , ((4) * inc) + (inc / 2) + radius, "mm"
PRINT * 6) Mid-span", (5) * inc) + (inc / 2) + radius, "mm"
PRINT * 7) "", , ((6) * inc) + (inc / 2) + radius, "mm"
PRINT * 8) "", , ((7) * inc) + (inc / 2) + radius, "mm"
PRINT * 9) "", , ((8) * inc) + (inc / 2) + radius, "mm"
PRINT * 10) "", , ((9) * inc) + (inc / 2) + radius, "mm"
PRINT * 11) Shroud", (10) * inc) + (inc / 2) + radius, "mm"
PRINT
OPEN "c:\utilities\mech\posn.dat" FOR INPUT AS #3
INPUT #3, p, plane$
CLOSE #3
PRINT * Current probe position: *, p
PRINT
610 INPUT * Enter new probe position (press 0 to quit): *, newpos
IF newpos = 0 THEN
   newpos = p
   GOTO 690
END IF
IF newpos < 1 OR newpos > 11 THEN GOTO 610
IF newpos = p THEN GOTO 685
IF newpos = 1 THEN
    newposn = hub + delta
    ELSE newposn = ((newpos - 1) * inc) + (inc / 2) + delta
END IF
    steps = INT(ABS(newposn - posn) * 200)
    ' PRINT newposn, posn, steps
    IF newpos > p THEN CALL probout
    IF newpos < p THEN CALL probin
685 IF a$ = "Y" OR a$ = "Y*" OR a$ THEN CALL travio
OPEN "c:\utils\mech\posn.dat" FOR OUTPUT AS #3
PRINT #3, newpos, plane$
CLOSE #3
GOTO 608
690 IF a$ = "Y" OR a$ = "Y*" OR a$ = "" THEN CLOSE #2
END SUB

SUB probreset

PRINT "Resetting traverse unit."
newpos = 1
IF newpos = p THEN GOTO 700
    newposn = ((newpos - 1) * inc) + (inc / 2) + delta
    steps = INT(ABS(newposn - posn) * 200)
REM PRINT newposn, posn, steps
    IF newpos > p THEN CALL probout
    IF newpos < p THEN CALL probin
700 CALL probadjust
OPEN "c:\utils\mech\posn.dat" FOR OUTPUT AS #3
PRINT #3, newpos, plane$
CLOSE #3
END SUB

SUB setplane

REM Initialise geometrical parameters when traverse plane is changed
IF plane$ = "blade" THEN
    inc = 7.682
hub = 7.16
delta = 0
radius = 112.8
END IF

IF plane$ = "outlet" THEN
  inc = 7.1
  hub = 3.55
  delta = 6.4
  radius = 35
END IF
END SUB

SUB shrtohub

REM shroud to hub traverse
PRINT "Moving probe to shroud ..."
newpos = 11
IF newpos = p THEN GOTO 850
REM Move probe out
newposn = ((newpos - 1) * inc) + (inc / 2) + delta
steps = INT(ABS(newposn - posn) * 200)
CALL probout
850 REM Start traverse
newpos = 1
newposn = hub + delta
steps = INT(ABS(newposn - posn) * 200)
y = 30
z = 0
860 steps = steps - z
PRINT
PRINT
PRINT "Set speed using + or - keys whilst traversing, and"
PRINT "press Esc key to interrupt traverse."
PRINT
INPUT "Press return to continue.", r
FOR z = 1 TO steps
FOR x = 1 TO y: NEXT
OUT &H322, 0:
FOR x = 1 TO y: NEXT
OUT &H322, 1:

213
Appendix A: Control Software

KEY$ = INKEY$
IF KEY$ = CHR$(45) THEN y = y + 10
IF KEY$ = CHR$(61) THEN y = y - 10
IF y < 30 THEN y = 30
IF KEY$ = CHR$(27) THEN GOTO 860

NEXT z
OPEN "C:\utils\mech\posn.dat" FOR OUTPUT AS #3
PRINT #3, newpos, plane
CLOSE #3
posn = newposn
y = 30
END SUB

SUB start
REM Initial data input screen
910 CLS
PRINT
PRINT
PRINT "Enter atmospheric pressure (default=*; pOl; "mm Hg): ";
INPUT "", pnew
PRINT "Enter atmospheric temperature (default=*; tOl; "°C): ";
INPUT "", tnew
PRINT "Enter mass flow rate (default=*; m; "kg/s): ";
INPUT "", mnew
IF pnew = 0 THEN pnew = pOl
IF tnew = 0 THEN tnew = tOl
IF mnew = 0 THEN mnew = m
pOl = pnew; tOl = tnew; m = mnew
PRINT
REM Iterate to find static conditions

REM Iterate to find static conditions

t0 = tOl + 273.15
p0 = pOl * 133.3
den = p0 / (t0 * 287.1)
d = .1016
a = 3.14159 * (d ^ 2) / 4
FOR l = 1 TO 6
V = m / (den * a)
t = t0 - (V ^ 2 / 2010)
ps = p0 / ((t0 / t) ^ 3.5)
Appendix A: Control Software

den = ps / (t * 287.1)
PRINT , ps, t, den, V
NEXT 1
deltap = ((m / (.99 * 1 * 1 * (3.14159 / 4) * d * d)) ^ 2) / (2 * den)
mmwater = deltap / 9.81
PRINT
PRINT Mass flow rate = " , m, "kg/s"
PRINT Stagnation pressure = " , p0, "Pa"
PRINT Stagnation temperature = " , t0, "K"
PRINT Static pressure = " , ps, "Pa"
PRINT Static temperature = "t, "K"
PRINT Density = " , den, "kg/m^3"
PRINT Velocity = " , V, "m/s"
PRINT delta P = " , deltap, "Pa"
PRINT U-tube delta P = " , mmwater, "mm"
PRINT
915 INPUT Would you like to do another calculation? (Y/(N))", a$
IF a$ = "Y" OR a$ = "y" THEN GOTO 910
IF a$ = "N" OR a$ = "n" OR a$ = "" THEN GOTO 920
GOTO 915
920 s = 99
END SUB

SUB travio
REM Aerodynamic data input during traverse operation
rad = radius + posn - delta
PRINT
PRINT Radius Pol - Po2 Dyn. Head Yaw angle"
OPEN "com1:9600,n,8,1,RB9" FOR INPUT AS #1
PRINT
PRINT " ; rad; " ;
INPUT #1, nul, tot, nul
PRINT tot; " ;
INPUT #1, nul, dat, nul
PRINT dat; " ;
INPUT "; yaw
PRINT #2, rad, tot, dat, yaw
CLOSE #1
SUB travop

REM Traverse options control screen
1000 CLS
OUT &H323, 128: REM All Ports set as outputs
OPEN "c:\utils\mech\posn.dat" FOR INPUT AS #3
INPUT #3, p, plane$
CLOSE #3
CALL setplane
IF p = 1 THEN
    posn = hub + delta
ELSE posn = ((p - 1) * inc) + (inc / 2) + delta
END IF
REM PRINT p, posn
PRINT
PRINT
PRINT
PRINT "Options available: -"
PRINT "1) Set probe position."
PRINT "2) Reset traverse unit."
PRINT "3) Continuous traverse from shroud to hub."
PRINT "4) Continuous traverse from hub to shroud."
PRINT "5) Change probe traverse plane."
PRINT "0) Return to main menu."
PRINT
PRINT "Probe is set for "; plane$; " plane, at position "; p; ";"
PRINT
INPUT "Please make selection: ", z
PRINT
IF z = 1 THEN CALL probpos
IF z = 2 THEN CALL probreset
IF z = 3 THEN CALL shrtohub
IF \( z = 4 \) THEN CALL hubtoshr
IF \( z = 5 \) THEN CALL changeplane
IF \( z = 0 \) THEN GOTO 1080
GOTO 1000
1080 \( s = 99 \)
END SUB
### Appendix B: Control Software Output Data File

#### igv=0°

<table>
<thead>
<tr>
<th>Radius</th>
<th>Po1-Po2</th>
<th>Dyn_Head</th>
<th>Yaw_angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>38.55</td>
<td>49.66</td>
<td>0346.1</td>
<td>0</td>
</tr>
<tr>
<td>45.65</td>
<td>50.47</td>
<td>0324.6</td>
<td>0</td>
</tr>
<tr>
<td>52.75</td>
<td>42.06</td>
<td>0353.7</td>
<td>3</td>
</tr>
<tr>
<td>59.85</td>
<td>35.93</td>
<td>0360.7</td>
<td>4</td>
</tr>
<tr>
<td>66.95</td>
<td>42.11</td>
<td>0354.8</td>
<td>3</td>
</tr>
<tr>
<td>74.05</td>
<td>39.18</td>
<td>0364.0</td>
<td>4</td>
</tr>
<tr>
<td>81.15</td>
<td>39.52</td>
<td>0356.9</td>
<td>4</td>
</tr>
<tr>
<td>88.25</td>
<td>39.67</td>
<td>0362.9</td>
<td>4</td>
</tr>
<tr>
<td>95.35</td>
<td>32.41</td>
<td>0355.9</td>
<td>2</td>
</tr>
<tr>
<td>102.45</td>
<td>33.11</td>
<td>0326.9</td>
<td>0</td>
</tr>
<tr>
<td>109.55</td>
<td>26.57</td>
<td>0342.7</td>
<td>0</td>
</tr>
</tbody>
</table>

#### Traverse Data

Traversing in outlet plane.

#### igv=30°

<table>
<thead>
<tr>
<th>Radius</th>
<th>Po1-Po2</th>
<th>Dyn_Head</th>
<th>Yaw_angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>38.55</td>
<td>143.14</td>
<td>0659.2</td>
<td>28</td>
</tr>
<tr>
<td>45.65</td>
<td>107.32</td>
<td>0571.2</td>
<td>32</td>
</tr>
<tr>
<td>52.75</td>
<td>73.62</td>
<td>0534.7</td>
<td>38</td>
</tr>
<tr>
<td>59.85</td>
<td>59.63</td>
<td>0500.0</td>
<td>39</td>
</tr>
<tr>
<td>66.95</td>
<td>51.69</td>
<td>0474.5</td>
<td>39</td>
</tr>
<tr>
<td>74.05</td>
<td>51.58</td>
<td>0451.2</td>
<td>38</td>
</tr>
<tr>
<td>81.15</td>
<td>51.19</td>
<td>0429.4</td>
<td>36</td>
</tr>
<tr>
<td>88.25</td>
<td>46.88</td>
<td>0415.6</td>
<td>34</td>
</tr>
<tr>
<td>95.35</td>
<td>43.88</td>
<td>0401.1</td>
<td>33</td>
</tr>
<tr>
<td>102.45</td>
<td>50.52</td>
<td>0359.4</td>
<td>31</td>
</tr>
<tr>
<td>109.55</td>
<td>52.23</td>
<td>0341.6</td>
<td>30</td>
</tr>
</tbody>
</table>
igv=60ø
762.15  22  1
Logging ...
-2423.2  Hub Static Tapping 1
-3244.7  Hub Static Tapping 2
-4079.6  Hub Static Tapping 3
-1896.2  Shroud Static Tapping 1
-2066.8  Shroud Static Tapping 2
-2211.4  Shroud Static Tapping 3
-2293.3  Shroud Static Tapping 4
1577.7  Mass Flow Device Delta P

Traverse Data
762.15  22  1
Traversing in outlet plane.
Radius  Pol-Po2  Dyn_Head  Yaw_angle
38.55  1071.2  2251.6  46
45.65  907.3  2169.4  52
52.75  562.7  2146.9  66
59.85  187.46  2101.1  74
66.95  39.87  1865.3  76
74.05  38.44  1568.1  76
81.15  65.23  1318.7  74
88.25  78.06  1144.2  71
95.35  71.51  1022.0  68
102.45  61.49  0895.0  65
109.55  99.2  0743.5  61
Relative velocity vectors at 50% span
Flat plate VIGV's @ 0°, Mass flow rate = 5.2 kg/s
Relative velocity vectors at 50% span
Flat plate VIGV's @ +20°, Mass flow rate = 4.3 kg/s
Relative velocity vectors at 50% span
Flat plate VIGV's @ +40°, Mass flow rate = 2.9 kg/s
Relative velocity vectors at 50% span

Flat plate VIGV's @ +60°, Mass flow rate = 1.9 kg/s
Relative velocity vectors at 10% span
Flap plate VIGV's @ 0°, Mass flow rate = 5.2 kg/s
Relative velocity vectors at 10% span

Flat plate VIGV's @ +20°, Mass flow rate = 4.3 kg/s
Relative velocity vectors at 10% span

Flat plate VIGV's @ +40°, Mass flow rate = 2.9 kg/s
Relative velocity vectors at 10% span

Flat plate VIGV's @ +60°, Mass flow rate = 1.9 kg/s
Relative Mach number contours at 50% span

Flat plate VIGV's @ 0°, Mass flow rate = 5.2 kg/s
Relative Mach number contours at 50% span
Flat plate VIGV's @ +20°, Mass flow rate = 4.3 kg/s
Relative Mach number contours at 50% span
Flat plate VIGV's @ +40°, Mass flow rate = 2.9 kg/s
Relative Mach number contours at 50% span
Flat plate VIGV's @ +60°, Mass flow rate = 1.9 kg/s
Relative Mach number contours at 10% span
Flat plate VIGV’s @ 0°, Mass flow rate = 5.2 kg/s
Relative Mach number contours at 10% span
Flat plate VIGV's @ +20º, Mass flow rate = 4.3 kg/s
Relative Mach number contours at 10% span
Flat plate VIGV's @ +40°, Mass flow rate = 2.89 kg/s
Relative Mach number contours at 10% span
Flat plate VIGV's @ +60°, Mass flow rate = 1.9 kg/s
### Appendix C: Stage Performance

**Relative Mach number contours through meridional plane**

Flat plate VIGV's @ 0°, Mass flow rate = 5.2 kg/s
Appendix C: Stage Performance

Relative Mach number contours through meridional plane
Flat plate VIGV's @ +20°, Mass flow rate = 4.3 kg/s
Relative Mach number contours through meridional plane
Flat plate VIGV's @ +40°, Mass flow rate = 2.9 kg/s
Relative Mach number contours through meridional plane
Flat plate VIGV's @ +60°, Mass flow rate = 1.9 kg/s
Relative velocity vectors at 50% span
Tandem VIGV's @ 0°, Mass flow rate = 5.2 kg/s
Relative velocity vectors at 50% span
Tandem VIGV's @ +20°, Mass flow rate = 4.3 kg/s
Relative velocity vectors at 50% span
Tandem VIGV's @ +40°, Mass flow rate = 2.9 kg/s
Relative velocity vectors at 50% span
Tandem VIGV's @ +60°, Mass flow rate = 1.9 kg/s
Relative velocity vectors at 10% span
Tandem VIGV's @ 0°, Mass flow rate = 5.2 kg/s
Relative velocity vectors at 10% span
Tandem VIGV's @ +20°, Mass flow rate = 4.3 kg/s
Relative velocity vectors at 10% span
Tandem VIGV's @ +40°, Mass flow rate = 2.9 kg/s
Relative velocity vectors at 10% span
Tandem VIGV's @ +60°, Mass flow rate = 1.9 kg/s
Relative Mach number contours at 50% span
Tandem VIGV's @ 0°, Mass flow rate = 5.2 kg/s
Relative Mach number contours at 50% span
Tandem VIGV’s @ +20°, Mass flow rate = 4.3 kg/s
Relative Mach number contours at 50% span
Tandem VIGV's @ +40°, Mass flow rate = 2.9 kg/s
Relative Mach number contours at 50% span
Tandem VIGV's @ +60°, Mass flow rate = 1.9 kg/s
Relative Mach number contours at 10% span
Tandem VIGV's @ 0°, Mass flow rate = 5.2 kg/s
Relative Mach number contours at 10% span
Tandem VIGV's @ +20°, Mass flow rate = 4.3 kg/s
Relative Mach number contours at 10% span
Tandem VIGV's @ +40°, Mass flow rate = 2.89 kg/s
Relative Mach number contours at 10% span
Tandem VIGV’s @ +60°, Mass flow rate = 1.9 kg/s
Relative Mach number contours through meridional plane

Tandem VIGV's @ 0°, Mass flow rate = 5.2 kg/s
Relative Mach number contours through meridional plane
Tandem VIGV's @ +20°, Mass flow rate = 4.3 kg/s
Relative Mach number contours through meridional plane
Tandem VIGV's @ +40°, Mass flow rate = 2.9 kg/s
Appendix C: Stage Performance

Relative Mach number contours through meridional plane

Tandem VIGV's @ +60°, Mass flow rate = 1.9 kg/s