Development of a 1-D performance prediction technique for automotive centrifugal compressors

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

Additional Information:

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

Metadata Record: https://dspace.lboro.ac.uk/2134/7796

Publisher: © Helen M. Meese

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/
Best Copy
Available
Development Of A 1-D Performance Prediction Technique For Automotive Centrifugal Compressors

PhD Thesis

By

Helen .M. Meese

23 May 2006
Abstract

The increasing demand for improved performance in diesel and petrol engines - particularly in the motor-sport industry - has increased the need for performance enhancing devices such as the automotive turbocharger. The prediction of compressor performance in the early design stage of a turbocharger is critical and helps to ensure that the range and matching of the constituent components (impeller, diffuser and volute) is satisfactory. Although the fluid flow inside the compressor is three-dimensional, effective analysis can be carried out using one-dimensional prediction techniques. Many prediction techniques have been developed over the decades and improvements to these methods have primarily been due to a greater understanding of compressor operation. The major gain from establishing more accurate prediction techniques is the reduction of uncertainties in both design time and production costs as well as allowing existing designs of centrifugal compressors to be improved.

This thesis presents the work carried out to develop a PC-based 1-D prediction technique called CAPRICE, with focus aimed at developing new models for the vaneless diffuser and volute casing. Extensive analysis of the existing models has been undertaken relating geometric features to performance. A specially-constructed interstage test rig, designed to extract data from the components, enabled experimental data from two compressors to be gathered. The data collected was used to develop several new correlations; A new correlation for the prediction of impeller work was produced and shown to be an improvement on the Wiesner equation. Formulae for the prediction of impeller and diffuser surge were developed which enabled a more accurate prediction of surge to be made and diffuser and volute loss and recovery coefficient correlations were produced, separating the diffusion system model in CAPRICE for the first time. An equation for the prediction of the length of the log spiral path in the diffuser was also derived. The work was validated against existing 1-D and 3-D models and shown to produce excellent comparisons and overall compressor maps have been produced to demonstrate these developments throughout this work.

The resulting 1-D performance prediction technique gives the designer better control of the overall performance by allowing a greater level of adjustment to be made to the individual component geometries, previously unavailable in CAPRICE.

Key words: turbocharger, compressor, vaneless diffuser, volute, 1-D prediction, modelling
Acknowledgements

I would like to take this opportunity to thank Loughborough University for giving me the chance to undertake this PhD. It has been a 10 year battle, but I have finally succeeded.

Thanks should also be given to the following people;

To Dr Ed Swain, who set me on this path. Don’t you wish you hadn’t? Thank you for pushing me to my limits and making me a stronger person for the experience. To Ms Helen Sankey and Ms Jo Mason, and all the clerical staff, thank you for being a shoulder to cry on and an ear to bend, you are appreciated more than you know. To Mr Henk Versteeg, who helped me through the toughest stage of this PhD, submission, your constant encouragement and kind words enabled me to finish with confidence, thank you.

And to all the staff in both the Mechanical & Electrical Engineering Dept’s. You know who you are. Your continued support and encouragement, especially through the rough times have helped me beyond measure, thank you for making me the engineer I am today.

Thanks also to Garrett Engine Boosting Systems, in particular Dr’s Quentin Roberts and Bill Connors. This research would not have been possible without the use of your test facilities and access to so much wonderful information and expertise.

Finally, I would like to thank the three most important people in my life. Firstly my partner Nick Hudson, whose love and understanding and regular kicks in the posterior kept me going for 10 years - words cannot say what you mean to me. To my Father, Bill Meese, you are and always will be the greatest engineer I know, thank you for showing me the joys of machinery.

And to my mother Val Meese, your passing has left a hole in my heart. You so wanted to see me graduate and I am sorry you will not, but your love and spirit have kept me going...

This is for you.
# Table of Contents

## A NOMENCLATURE

## B TERMINOLOGY

### 1 INTRODUCTION TO TURBOCHARGING

#### 1.1 INTRODUCTION

#### 1.2 COMPRESSOR CHARACTERISTICS

#### 1.3 COMPRESSOR OPERATION

- 1.3.1 Performance Limitation

#### 1.4 KEY GEOMETRIC FEATURE & 1-D AERODYNAMIC ANALYSIS

- 1.4.1 Inlet Casing
- 1.4.2 Impeller
  - 1.4.2.1 Backsweep & The Exit Velocity Triangle
  - 1.4.2.2 Impeller Efficiency
  - 1.4.2.3 Impeller Work
  - 1.4.2.4 Slip
  - 1.4.2.5 Impeller Loss
- 1.4.3 Diffuser
  - 1.4.3.1 Vector Triangle
  - 1.4.3.2 Diffuser Efficiency
  - 1.4.3.3 Diffuser Losses
- 1.4.4 Volute

#### 1.5 CONCLUSION
2 STATE OF ART LITERATURE REVIEW

2.1 INTRODUCTION

2.2 DIFFUSERS

2.2.1 Channel diffusers
2.2.2 Curved & Vaned Diffusers
2.2.3 Summary
2.2.4 Vaneless Diffusers – Flow Distortion & Component Interaction
2.2.4 Conclusion

2.3 VOLUTES

2.3.1 Volute Interaction
2.3.2 The Work of Braembussche
2.3.3 Conclusions

2.4 FLOW LIMITATIONS

2.4.1 Surge
2.4.2 Stall
2.4.3 Choke
2.4.4 Conclusions

2.5 SUMMARY

2.6 OBJECTIVES & JUSTIFICATION FOR THE RESEARCH

2.7 STRUCTURE OF THE THESIS

3. NUMERICAL METHODS & THE PERFORMANCE PREDICTION TECHNIQUE – CAPRICE

3.1 INTRODUCTION

3.2 NUMERICAL TECHNIQUES

3.2.1 1-D Techniques
3.2.2 Computer-Based Techniques
3.2.3 3-D Techniques – Computational Fluid Dynamics
3.2.4 Conclusion

3.3 INTRODUCTION TO THE INTEGRATED DESIGN SYSTEM

3.3.1 Background to the 1-D Off-Design Prediction

Program - CAPRICE

3.3.1.1 Centrifugal Compressor Losses

3.3.1.2 Surge

3.3.1.3 Blockage

3.3.1.4 Impeller Model

3.3.1.5 Diffusion System Model

3.3.1.6 Early CAPRICE Predictions

3.3.2 CAPRICE's Iteration Procedure

3.3.3 Limitations of the CAPRICE Prediction Program

3.4 CONCLUSION

4 CAPRICE PREDICTION & AN INITIAL EXAMINATION OF THE DIFFUSION SYSTEM

4.1 INTRODUCTION

4.2 THE CAPRICE PREDICTION AND EXPERIMENTAL DATA

4.2.1 Experimental Date Extraction

4.2.2 Geometric Data Extraction

4.2.2.1 Blade Profile

4.2.2.2 Throat Area

4.2.2.3 Rake & Wrap Angles

4.2.2.4 Blade Thickness

4.2.2.5 Backsweep Angle

4.2.2.6 Inducer Inlet Angle

4.2.2.7 RMS Diameter

4.2.3 Summary
4.3 PREDICTED & EXPERIMENTAL COMPRESSOR MAPS

4.3.1 Predicted & Experimental Overall Efficiency Ratio

4.3.2 Summary

4.4 EXAMINING THE DIFFUSION SYSTEM USING AN INVERSE APPROACH

4.4.1 The Inverse CAPRICE Program

4.4.2 Diffusion System Efficiency Ratio

4.5 IMPROVING THE IMPELLER MODEL

4.5.1 Impeller Entropy Loss Coefficient

4.5.2 Impeller Work

4.5.2.1 Work & Mass Flow Function Correlation

4.5.2.2 Work & Velocity Ratio Correlation

4.5.2.3 Improved Work Correlation

4.5.3 Impeller Surge

4.5.3.1 Inducer Incidence at Surge

4.5.3.2 Impeller Exit Angle at Surge

4.5.4 Mach Number at Surge

4.5.5 Summary

4.6 CAPRICE2 PREDICTIONS

4.6.1 Overall Characteristics

4.6.2 Overall Efficiency Ratio

4.6.3 Diffusion System Efficiency Ratio

4.6.4 Impeller Work

4.6.5 Impeller Surge

4.6.6 Conclusion

4.7 SCOPE OF THE INITIAL RESEARCH

4.8 FURTHER DEVELOPMENT OF THE RESEARCH
6 OVERALL CHARACTERISTICS, PRESSURE VARIATIONS & CFD VALIDATION

6.1 INTRODUCTION

6.2 PRESSURE VARIATIONS INSIDE THE COMPRESSOR
   6.2.1 Determining the Impeller Characteristic from the Diffuser Pinch Characteristic
   6.2.2 Compressor Static Pressure Variations
   6.2.3 Swirl between Volute Exit and Downstream Ducting
   6.2.4 Conclusion

6.3 COMPARISON OF EXPERIMENTAL & CAPRICE2 DATA
   6.3.1 Overall Performance Maps
   6.3.2 Overall Efficiency Ratio
   6.3.3 Impeller Efficiency Ratio

6.4 COMPUTATIONAL ANALYSIS
   6.4.1 Comparison of CFD & Experimental Characteristic
   6.4.2 Comparison of CFD & Experimental Overall Efficiency

6.5 SUMMARY

7 CAPRICE2 MODEL VALIDATION

7.1 INTRODUCTION

7.2 IMPELLER MODEL IMPROVEMENTS
   7.2.1 Impeller Loss Coefficient
   7.2.2 Impeller Correlations
      7.2.2.1 Simple Curve-fitting Technique
7.2.2.2 Twin Efficiency Equation Technique

7.2.3 Impeller Work

7.2.3.1 Comparison of CFD and Experimental Work

7.2.3.2 Improved Work Model

7.2.4 Impeller Surge Correlations

7.2.5 Inducer and Diffuser Pinch Mach Number at Surge

7.3 THE DIFFUSION SYSTEM

7.3.1 Comparison Of CFD And Experimental Component Efficiency

7.3.2 Separating the Diffusion System Components

7.3.3 Diffusion System Surge

7.3.4 Volute Mach Number at Surge

7.4 SUMMARY

7.5 REVIEW OF PRESENTED WORK

8 DEVELOPING THE DIFFUSER AND VOLUTE LOSS MODEL

8.1 INTRODUCTION

8.2 PROPORTIONING LOSSES

8.3 EXPERIMENTAL LOSSES IN THE DIFFUSER AND VOLUTE

8.3.1 Experimental Diffuser Loss Coefficient

8.3.1.1 Comparison of CFD & Experimental Loss Coefficient

8.3.2 Experimental Volute Loss Coefficient

8.3.3 Conclusion

8.4 VARIATION OF DIFFUSER EXIT GAS ANGLE WITH DIFFUSER GEOMETRY

8.4.1 The Log Spiral Path

8.4.2 Variations in Diffuser Width

8.4.3 Conclusion
8.5 ASSESSING DIFFERENT TYPES OF LOSS MODELLING TECHNIQUES

8.5.1 Runstadler Equivalent Model
   8.5.1.1 Diffuser Equivalent Geometry
   8.5.1.2 Comparison of CFD & Experimental Blockage
   8.5.1.3 Volute Equivalent Geometry
   8.5.1.4 Summary

8.5.2 Simple Pipe Loss Models
   8.5.2.1 Pipe Loss of the Vaneless Diffuser
   8.5.2.2 Pipe Loss of the Volute
   8.5.2.3 Volute Pipe Model with Helical Flow
   8.5.2.4 Conclusion

8.5.3 Rogers Vaneless Diffuser Loss Model

8.5.4 Vaneless Diffuser Empirical Model
   8.5.4.1 Correlation of Experimental Data with Geometry and Shape Function
   8.5.4.2 Determining the Speed Function
   8.5.4.3 Conclusion

8.5.5 The Van Den Braembussche Volute Model

8.5.6 Volute Empirical Model
   8.5.6.1 Sub-dividing the Volute
   8.5.6.2 Conclusion

8.6 SUMMARY

9 ASSESSING THE ACCURACY OF THE NEW CORRELATIONS AND CAPRICE3 – THE IMPROVED PREDICTION TECHNIQUE

9.1 INTRODUCTION

9.2 CAPRICE3 PROGRAM STRUCTURE
A. Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Area</td>
<td>m²</td>
</tr>
<tr>
<td>AR</td>
<td>Area Ratio</td>
<td>A₄/A₃</td>
</tr>
<tr>
<td>A/r</td>
<td>Volute Area to radius ratio</td>
<td></td>
</tr>
<tr>
<td>AS</td>
<td>Aspect Ratio</td>
<td>b/(π.D)</td>
</tr>
<tr>
<td>B</td>
<td>Blockage Factor</td>
<td>1 - (ū / U)</td>
</tr>
<tr>
<td>b</td>
<td>Width of diffuser passage</td>
<td>m</td>
</tr>
<tr>
<td>Cr</td>
<td>Friction factor</td>
<td></td>
</tr>
<tr>
<td>Ch₁₀⁻¹</td>
<td>Inlet enthalpy loss coefficient</td>
<td></td>
</tr>
<tr>
<td>Cₚ</td>
<td>Specific Heat at constant pressure</td>
<td>J/kgK</td>
</tr>
<tr>
<td>Cₚₘₐₚ</td>
<td>Mean specific heat</td>
<td>J/kgK</td>
</tr>
<tr>
<td>Cₚᵲᵢ</td>
<td>Ideal Pressure Recovery</td>
<td>1-(1/AR²)</td>
</tr>
<tr>
<td>Cₚᵲᵢ</td>
<td>Static Pressure Recovery coefficient</td>
<td>(P₄ - P₃) / (P₀₃ - P₃)</td>
</tr>
<tr>
<td>Cₚᵲᵢ/Cₚᵲᵢ</td>
<td>Effectiveness</td>
<td></td>
</tr>
<tr>
<td>Cr</td>
<td>Contraction ratio</td>
<td></td>
</tr>
<tr>
<td>ε</td>
<td>Deflection</td>
<td>m</td>
</tr>
<tr>
<td>D</td>
<td>Diffuser Diameter</td>
<td>m</td>
</tr>
<tr>
<td>DR</td>
<td>Diameter Ratio</td>
<td>D₄/D₃</td>
</tr>
<tr>
<td>Dₑₐᵥ</td>
<td>Equivalent Pipe Diameter</td>
<td>√(4.A/π)</td>
</tr>
<tr>
<td>g</td>
<td>Gravitational constant</td>
<td>m/s²</td>
</tr>
<tr>
<td>h</td>
<td>Enthalpy</td>
<td>J/kgK</td>
</tr>
<tr>
<td>Δh</td>
<td>Enthalpy change</td>
<td>J/kgK</td>
</tr>
<tr>
<td>hᵣ</td>
<td>head lost to friction</td>
<td>Pa</td>
</tr>
<tr>
<td>i</td>
<td>Incidence angle</td>
<td>deg</td>
</tr>
<tr>
<td>KR</td>
<td>Relative Roughness</td>
<td></td>
</tr>
<tr>
<td>K</td>
<td>Constant of free vortex</td>
<td></td>
</tr>
<tr>
<td>Symbol</td>
<td>Definition</td>
<td></td>
</tr>
<tr>
<td>--------</td>
<td>------------</td>
<td></td>
</tr>
<tr>
<td>$k$</td>
<td>total density loss coefficient</td>
<td></td>
</tr>
<tr>
<td>KE</td>
<td>Kinetic energy</td>
<td></td>
</tr>
<tr>
<td>$L_{eqv}$</td>
<td>Equivalent Pipe Length</td>
<td></td>
</tr>
<tr>
<td>$l$</td>
<td>pipe length</td>
<td></td>
</tr>
<tr>
<td>$l$</td>
<td>width of volute passage</td>
<td></td>
</tr>
<tr>
<td>L</td>
<td>Radial Length Of Diffuser $r_4 - r_3$</td>
<td></td>
</tr>
<tr>
<td>LWR</td>
<td>Length to Width Ratio $L_{eqv}/b$</td>
<td></td>
</tr>
<tr>
<td>N</td>
<td>Rotational speed rev/sec</td>
<td></td>
</tr>
<tr>
<td>NS</td>
<td>specific speed $N \sqrt{\frac{V}{(N^2 D^2)^{3/4}}}$</td>
<td></td>
</tr>
<tr>
<td>$m$</td>
<td>Mass flow rate kg/s</td>
<td></td>
</tr>
<tr>
<td>M</td>
<td>Mach Number</td>
<td></td>
</tr>
<tr>
<td>$m$</td>
<td>mean hydraulic depth $A/P = ((\pi/4)D^2)/(\pi D) = D/4$ m</td>
<td></td>
</tr>
<tr>
<td>PE</td>
<td>Potential energy J</td>
<td></td>
</tr>
<tr>
<td>$P_0$</td>
<td>Stagnation Pressure Pa</td>
<td></td>
</tr>
<tr>
<td>P</td>
<td>Static Pressure Pa</td>
<td></td>
</tr>
<tr>
<td>$\dot{Q}$</td>
<td>rate of heat transfer W</td>
<td></td>
</tr>
<tr>
<td>Q</td>
<td>Heat Transfer J</td>
<td></td>
</tr>
<tr>
<td>$r$</td>
<td>Radius m</td>
<td></td>
</tr>
<tr>
<td>R</td>
<td>Gas Constant kJ/kgK</td>
<td></td>
</tr>
<tr>
<td>$R_c$</td>
<td>Pressure ratio</td>
<td></td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds Number</td>
<td></td>
</tr>
<tr>
<td>S</td>
<td>Log Spiral Flow Path m</td>
<td></td>
</tr>
<tr>
<td>$s$</td>
<td>Entropy kJ/kgK</td>
<td></td>
</tr>
<tr>
<td>$T_0$</td>
<td>Stagnation Temperature K</td>
<td></td>
</tr>
<tr>
<td>T</td>
<td>Static Temperature K</td>
<td></td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
<td>Unit</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
<td>------</td>
</tr>
<tr>
<td>U</td>
<td>Blade Speed</td>
<td>m/s</td>
</tr>
<tr>
<td>(\dot{u})</td>
<td>mean velocity</td>
<td>m/s</td>
</tr>
<tr>
<td>U</td>
<td>velocity</td>
<td>m/s</td>
</tr>
<tr>
<td>V</td>
<td>Absolute Velocity of fluid</td>
<td>m/s</td>
</tr>
<tr>
<td>(V)</td>
<td>volumetric flow rate</td>
<td>m³/s</td>
</tr>
<tr>
<td>(V_w/V_r)</td>
<td>Velocity Ratio</td>
<td></td>
</tr>
<tr>
<td>W</td>
<td>Relative Velocity of fluid</td>
<td>m/s</td>
</tr>
<tr>
<td>w</td>
<td>width</td>
<td>m</td>
</tr>
<tr>
<td>W</td>
<td>Work done</td>
<td>J</td>
</tr>
<tr>
<td>WDR</td>
<td>Width To Diameter Ratio</td>
<td>by/D₃</td>
</tr>
<tr>
<td>Z or n</td>
<td>Number of blades</td>
<td></td>
</tr>
<tr>
<td>z</td>
<td>elevation</td>
<td></td>
</tr>
<tr>
<td>U/NT</td>
<td>Blade Speed Parameter</td>
<td></td>
</tr>
<tr>
<td>(2\theta_{eqv})</td>
<td>Equivalent Divergence Angle</td>
<td>deg</td>
</tr>
<tr>
<td>(\Delta P)</td>
<td>Pressure Difference</td>
<td>Pa</td>
</tr>
<tr>
<td>(\Delta T)</td>
<td>Temperature difference</td>
<td>K</td>
</tr>
<tr>
<td>(\alpha)</td>
<td>Flow Angle to the Radial</td>
<td>deg</td>
</tr>
<tr>
<td>(\beta)</td>
<td>Backsweep</td>
<td>deg</td>
</tr>
<tr>
<td>(\delta)</td>
<td>Boundary Layer</td>
<td></td>
</tr>
<tr>
<td>(\phi)</td>
<td>Azimuth angle</td>
<td>deg</td>
</tr>
<tr>
<td>(\gamma)</td>
<td>ratio of specific heats = (C_p/C_v)</td>
<td></td>
</tr>
<tr>
<td>(\theta)</td>
<td>Flow Angle To The Tangential ((90^\circ - \alpha))</td>
<td>deg</td>
</tr>
<tr>
<td>(\eta)</td>
<td>Efficiency</td>
<td></td>
</tr>
<tr>
<td>(\mu)</td>
<td>Viscosity</td>
<td>kg/ms</td>
</tr>
<tr>
<td>(\rho)</td>
<td>Density</td>
<td>kg/m³</td>
</tr>
</tbody>
</table>
\( \sigma \) Slip factor

\( \zeta \) Stagnation Pressure Loss Coefficient \( \frac{(P_{03} - P_{04})}{(P_{03} - P_{3})} \)

**Subscripts**

- 0 Stagnation condition
- t-t or T-T Total to Total
- t-s or T-S Total to Static
- 1 Impeller inlet
- 2 Impeller exit
- 3 Diffuser inlet
- 4 Diffuser exit
- 5 Volute tongue
- 6 Volute throat
- 7 Volute exit
- h hub
- ind inducer
- r Radial component
- ref reference value typically peak of range
- t tip
- T throat
- ton tongue
- s surge
- s isentropic change
- w Tangential component
- c choke
- \( \theta \) tangential component in diffuser and volute
B. Terminology

aspect ratio  
backsweep angle  
blockage factor  
choke  
deflection  
dynamic head  
flow coefficient  
hub  
hub-tip-ratio  
incidence  
inducer  
impeller  
Mach number
<table>
<thead>
<tr>
<th>Term</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>meridional</td>
<td>A plane cutting a turbomachine through a diameter and the longitudinal axis following the blade form</td>
</tr>
<tr>
<td>profile</td>
<td>Associated with a blade shape, e.g. losses, co-ordinates</td>
</tr>
<tr>
<td>radius ratio</td>
<td>The radius ratio of two datum sections, e.g. radius of diffuser inlet section to impeller outer radius is the vaneless diffuser radius ratio</td>
</tr>
<tr>
<td>rake angle</td>
<td>Angle blade tip makes with radial in an axial direction</td>
</tr>
<tr>
<td>separation</td>
<td>When a fluid flowing along a surface ceases to be parallel to the surface but flows over a near-stagnant bubble, or eddy, or around another stream of fluid</td>
</tr>
<tr>
<td>shroud</td>
<td>Surface defining the outer diameter of a turbomachine flow annulus</td>
</tr>
<tr>
<td>squareness</td>
<td>Ratio of inducer tip to outer diameter</td>
</tr>
<tr>
<td>slip factor</td>
<td>Whirl velocity as a proportion of blade speed</td>
</tr>
<tr>
<td>stagnation</td>
<td>Total - conditions that would ensue if a moving fluid was brought isentropically to rest</td>
</tr>
<tr>
<td>stall</td>
<td>The condition of operation (usually defined in terms of incidence) of an aerofoil, or cascade, at which the fluid deflection begins to fall rapidly and/or losses increase rapidly. Typically defined as the operating point where losses are twice the minimum</td>
</tr>
<tr>
<td>static conditions</td>
<td>Properties of a moving fluid as they would be obtained using instruments moving with the flow</td>
</tr>
<tr>
<td>surge</td>
<td>Unstable operating region caused by adverse pressure gradients. Cycle of events creates a reversal of flow and a reduction of the adverse pressure gradient, followed by a re-establishment of forward flow leading to a repetition of the sequence</td>
</tr>
<tr>
<td>temp.rise coef.</td>
<td>Non-dimensional increase in temperature across a compressor stage - specific work $\frac{\Delta H}{U^2}$ - work coefficient, or loading coefficient</td>
</tr>
<tr>
<td>throat area</td>
<td>Generally the smallest cross-sectional area in a blade row</td>
</tr>
<tr>
<td>throat opening</td>
<td>Opening - the 2D distance associated with the throat area</td>
</tr>
<tr>
<td>Term</td>
<td>Definition</td>
</tr>
<tr>
<td>------------</td>
<td>---------------------------------------------------------------------------</td>
</tr>
<tr>
<td>tip</td>
<td>the outermost section of the blade or vane</td>
</tr>
<tr>
<td>vane</td>
<td>alternative name for blade, but typically for radial impellers</td>
</tr>
<tr>
<td>volute</td>
<td>scroll shape widely used as a collector after a radial compressor, or flow director at inlet to a centripetal turbine</td>
</tr>
<tr>
<td>whirl velocity</td>
<td>tangential component of the absolute velocity</td>
</tr>
<tr>
<td>wrap angle</td>
<td>Angle at which blade bends positively or negatively away from a datum point</td>
</tr>
</tbody>
</table>
1. Introduction to Turbocharging

Synopsis

Chapter one introduces the research project and discusses the relevance of 1-D performance prediction at the initial design stage. An introduction to the principles of turbocharging is presented, in particular the operation and construction of the automotive turbocharger compressor and its constituent components. Included in this section is a review of the fluid mechanic and thermodynamic theory required in understanding the compressor performance characteristics and the relationships between compressor performance and geometry is presented in terms of a 1-D numerical approach.
1.1 Introduction

The development of new technologies and the growing need to improve performance in diesel and petrol engines - particularly in the motor-sport industry - has increased the need for performance enhancing devices such as the automotive super- and turbo-charger.

The relatively small size, weight and low cost of these devices, combined with their high performance, durability and reliability have made them an important part of modern engine design, and it is recognised that further development, especially for passenger vehicle applications, would provide benefits to a wider automotive market. Moreover, as ever increasing pressures are placed on engine manufacturers to conform to stringent emissions regulations, turbochargers are increasingly being viewed as a means of reducing environmental pollution; today's turbocharged diesel engines producing 50% less NOx and CO₂ emissions than conventional engines\(^1,2\).

Supercharging is a technique whereby air required in the combustion cycle of an engine is compressed to a higher than ambient pressure than that normally available to a naturally aspirated engine. The compressed air allows more fuel to be combusted, which increases the power output of the engine for a fixed engine size, resulting in improved engine performance and efficiency\(^3\). Two typical automotive superchargers, which utilise different forms of compression are shown in figure 1-1.
The turbocharger is a specific form of supercharging and was developed during the 1920’s as a result of the increasing need for improved power-to-weight ratios of medium-sized diesel engines such as marine propulsion engines and power generation units.

Figure 1-2  A typical automotive turbocharger [2]

Their popularity quickly spread to the automotive sector (a typical automotive design being shown in figure 1-2) where manufacturers were able to combine high engine performance with fuel economy using smaller turbocharged engines. The basic size of the turbocharger is governed by the quantity of air required by the engine, which is determined by the speed, boost pressure and volume of the cylinders.

Most of the recent developments in super- and turbocharger manufacture have been in the high performance market with significant developments taking place in the field of centrifugal compressor design. The supercharger compressor is typically driven by the engine crankshaft, but its main limitation is that the driving power it requires is restricted by the crankshaft speed. However, supercharging is an attractive option in small automotive engines where engine response is the main requirement and fuel efficiency is of secondary importance. Unlike the supercharger, the turbocharger compressor is driven by a turbine via a connecting shaft. The exhaust gases from the engine are directed into the turbine and then expanded to atmosphere.
Turbochargers are constructed in a wide range of configurations, using both axial and radial turbines and radial compressor arrangements\(^{[4-7]}\) and figure 1-3 shows a cut-away of a typical automotive turbocharger with a centrifugal compressor (consisting of a compressor wheel or 'impeller' followed by a diffuser and volute casing) and radial turbine.

![Typical automotive turbocharger configuration](image)

**Figure 1-3**  typical automotive turbocharger configuration \([18]\)

The operating conditions of the centrifugal compressor have a considerable influence on its design, particularly in small automotive types where it is important to retain a wide operating range, whilst maintaining compactness and cost effectiveness\(^{[5]}\). Traditional methods of manufacture and design applied by turbocharger manufacturers have relied upon re-sizing existing compressor units to establish new designs; both the turbine and compressor rotors being developed for a specific application which are then fitted to existing centre casings and bearing housings. Although this method has proved adequate over the years, large-scale experimental prototype testing is still required and a good deal of experience is necessary to ensure the required performance characteristic is achieved. This technique does not necessarily utilise the best aspects of each component and therefore, the overall performance achieved is not necessarily the optimum of the compressor. With the increased use of computer-based design systems, the development of new compressors and improvements to their performance can be rapidly achieved without extensive testing. Even
so, further development of these systems is still required as complete reliance on them is not yet achievable.

The prediction of compressor performance in the early design stage of a turbocharger is a critical element in the manufacturing process and helps to ensure that the range and matching of the constituent components is satisfactory. Performance prediction gives the designer an idea of the required compressor geometry and the engineer an understanding of the operating characteristics. Although the fluid flow inside the compressor is three-dimensional, effective analysis can be carried out using one-dimensional techniques[3-9]. A 1-D or Mean-line numerical technique utilises the co-ordinate direction in which the flow is being analysed and assumes uniform flow conditions across any given cross-sectional area. This type of approach does not account for the viscous effects in the flow, however, the simplicity of such techniques make them extremely popular[9-14]. These methods are generally based on the entry and exit conditions of the fluid using equations of state and continuity along with the assumptions of perfect or semi-perfect fluids[15,16]. From these equations, it is possible to determine the fluid's pressure, temperature and velocity with a reasonable degree of accuracy and enable the compressor characteristic to be produced with the minimum of geometric data. Many prediction techniques have been developed over the past forty years and improvements in these methods have primarily been due to technological advancements and a greater understanding of compressor operation. There are however, few generic techniques available to predict performance. Methods which have been developed are generally for specific applications produced 'in-house' by turbomachinery companies and are not generally applied to other forms of centrifugal compressor.

Two other significant issues arise with these 1-D methods; Firstly, they are largely based upon 1-D flow assumptions and so require considerable amounts of empirical data to enable accurate predictions to be produced. In many circumstances, the available data are limited, especially during the initial design phase of the compressor. Secondly, the overall performance of a centrifugal compressor is significantly affected by the geometry of individual components, especially in terms of stability, range and losses. Despite the apparent geometric simplicity of components such as the vaneless diffuser and volute casing, the complex 3-D flow profiles mean it is difficult to accurately predict their performance characteristics. It is therefore common to see many prediction methods
lumping these components together as a single ‘diffusion system’. However, as will be demonstrated, the centrifugal compressor is strongly affected by their capability to transform the energy of the fluid into pressure and therefore the vaneless diffuser and volute require particular consideration.

A comprehensive prediction technique, which enables the designer to examine the performance of all the individual components within the compressor at the initial design stage, would result in a device designed to its optimum specification. The major gain from establishing a more accurate prediction technique would enable fewer uncertainties in both design time and production costs and would allow existing designs of centrifugal compressors to be improved.

The principal incentive therefore, in undertaking this research is to provide the designer with a simple but accurate tool for the performance prediction of automotive turbocharger compressors and their constituent components at the initial design stage.

The aims of this research are:

1. To increase understanding of the aerodynamic behaviour of the centrifugal compressor, particularly the diffusion system components, namely the vaneless diffuser & volute.

2. To investigate the effects of geometry on the overall performance of the compressor and to study the interaction of the interstage components and their effect on performance.

3. To improve existing performance prediction techniques by developing a more accurate prediction method, incorporating findings from experimental investigations and by using 3-D modelling techniques such as Computational Fluid Dynamics to support these developments.

4. To draw up a series of design rules building on presently used methods, to improve performance and design, thus enabling new designs to be assessed before production.
1.2 Centrifugal Compressor Operation

Firstly, it is appropriate to identify the common components that make up a typical turbocharger centrifugal compressor. Figure 1-4 shows a cut-away of the compressor which consists of three main components.

1. Impeller
2. Diffuser, which comes in two main forms, vaned (shown in figure 1-3) and vaneless
3. Collector or Volute

![Figure 1-4](image)  
A typical centrifugal compressor [18]
A typical meridional cross-section of a compressor, with the principle dimensions and location of the three main components, is shown in figure 1-5.

![Diagram of a centrifugal compressor](image)

**Figure 1-5 Meridional view of centrifugal compressor**

Automotive turbocharger compressors of the type used throughout this research are generally designed without inlet guide vanes (IGV's). It is therefore assumed that the air enters the inlet casing and impeller axially as shown in figure 1-4. The inlet casing directs the air into the compressor, where the velocity of the air is increased as it approaches the impeller inducer, so reducing the static pressure. Energy is transferred to the air by the rotation of the impeller as it passes over the vanes, imparting swirl to the air. The static pressure rises due to the rotational acceleration. The air leaves the impeller exit at a high velocity and passes into the stationary diffuser. The diffuser converts the air’s high velocity into pressure by diffusing it through an increase in cross-sectional area, reducing the velocity to approximately that of its initial inlet velocity. The air then enters the volute, which collects the air around the circumference of the diffuser. A small amount of diffusion occurs in the volute before the air exits the compressor.
With a general outline of the compressor's operation it is possible to further describe the overall characteristic of the compressor graphically, by applying fundamental thermo- and aerodynamic principles.

1.3 Compressor Characteristics

Many thermodynamic problems associated with devices such as turbines and compressors can be solved by applying a rather idealised steady-state process\(^{[15]}\) to the fluid flow, providing the designer with a general understanding of performance of the compressor and its components. It could be argued that the compressor is not actually a steady-state device, however, if one considers a 'snap-shot' in time, then the flow conditions can be conveniently analysed in this way. Under this assumption the conservation equations of mass, momentum and energy\(^{[15,17]}\) can be applied and will be referred to throughout this work. It is assumed that the process of compression is adiabatic i.e. there is no heat transfer between the fluid and the surroundings and that air is considered as a semi-perfect gas.

1.3.1 Performance Limitation

To understand the behaviour of a specific compressor stage it is convenient to describe it in non-dimensional terms, enabling experimental data from different sources and different sized machines to be compared quickly and accurately\(^{[5]}\). If a specific compressor is assumed to be running with a semi-perfect fluid then the non-dimensional groups used to analyse the compressor characteristic can be written:

\[
\frac{P_{02}}{P_{01}} = \zeta \left( \frac{m \sqrt{T_{01}}}{P_{01}}, \frac{N}{\sqrt{T_{01}}} \right)
\]

Equation (1-1)

These parameters can be presented by plotting the mass flow rate parameter, \((m \sqrt{T_{01}})/P_{01}\) against the pressure ratio parameter \(P_{02}/P_{01}\), for a range of non-dimensional operating speeds \(N/\sqrt{T_{01}}\) known as a 'performance characteristic map'. A typical performance map for a centrifugal compressor is shown in figure 1-6.
To fully understand the operating range and limitations of the centrifugal compressor, this rather complex graphical representation can be simplified if one considers a single theoretical speed line as shown in figure 1-7. Consider the mass flow rate of the air to be adjusted by a valve placed down stream of the compressor. When the valve is shut the mass flow is zero, but there will be a small amount of pressure rise produced by the action of the air trapped between the vanes of the impeller at point A. As the valve opens, theoretically the mass flow and pressure ratio should increase up to a maximum point at ‘B’. The section of the curve between ‘A’ and ‘B’ is only theoretical and is not an acceptable area of operation. It is a region of severely unstable flow caused by adverse pressure gradients, which result in a phenomenon known as ‘Surge’.

Figure 1-6 Typical compressor performance characteristic map [18]
Surge is described as a sudden reversal of the flow leading to a drop in pressure and repeated pulsations of air through the compressor. Surge occurs for example, if the compressor operates on the positive slope of the curve say, at point ‘D’, a momentary decrease in mass flow causes a decrease in the pressure rise; therefore, the pressure on the down-stream side of the compressor at that moment is larger. This causes the flow to reverse back into the compressor, which reduces the down-stream pressure rapidly. When the pressure on the down-stream side falls far enough the compressor returns to its previous operating position and the cycle of events begins again. As long as the operating point remains on the negative side of the curve for example E, then stable operation will occur.

There is a second region, which limits the compressors operation, known as ‘Choke’ between points ‘B’ and ‘C’, for example at point ‘E’. The choke region is where the fluid flow through the compressor approaches maximum capacity and is a function of geometry. This phenomenon usually occurs when the flow approaches a Mach number = 1 at any minimum cross-sectional area in the compressor particularly in the bladed sections of the compressor like the inducer and vaned diffuser throats. As mass flow increases the pressure begins to decrease which leads to an increase in velocity at a constant speed. Eventually a point is reached where no further increase in mass flow can be gained and the compressor is said to have choked. It is possible to increase the mass flow rate by increasing the rotational speed of the impeller, but this is limited to the maximum speed the shaft can achieve. Any further increase in the mass flow rate will result in a decrease in the pressure ratio falling to point ‘C’.

This then, defines the envelope within which the compressor can operate. The shape of this curve varies from low to high speeds changing the range of mass flow rates the compressor
can operate over, as shown in figure 1-6 and it is this range and the limitation of surge and choke which the designer seeks to determine.

Contours of constant isentropic efficiency can also be superimposed onto the compressor map (shown as hashed lines in figure 1-6) to show where the best overall efficiency is for a given mass flow rate and pressure ratio. The isentropic efficiency of a compressor can be defined as a ‘total to total’ efficiency \( \eta_{cTT} \) and can be expressed as\(^{[5,17]}\):

\[
\eta_{cTT} = \frac{h_{07s} - h_{01}}{h_{07} - h_{01}} = \frac{T_{07s} - T_{01}}{T_{07} - T_{01}} \quad \text{eqn. (1-2)}
\]

By applying the appropriate thermodynamic assumptions relating pressure and temperature\(^{[15,17]}\) a definition of efficiency can be derived for a ‘total to total’ process which will be used throughout this work.

\[
\eta_{cTT} = \frac{T_{01} \left( \left( \frac{T_{07s}}{T_{01}} \right) - 1 \right)}{T_{07} - T_{01}} = \frac{T_{01} \left( \left( \frac{P_{07}}{P_{01}} \right)^{\frac{r-1}{r}} - 1 \right)}{\Delta T} \quad \text{eqn. (1-3)}
\]

1.4 Key Geometric Features & 1-D Aerodynamic Analysis

The complexities of modelling the flow inside the centrifugal compressor cannot be understated. New 2- and 3-D techniques for analysing flow and determining new geometries are constantly being refined and developed, and levels of performance and efficiency that were once considered unattainable are now well within the designers grasp. Yet many of these complex modelling techniques still rely upon simple one-dimensional flow models derived from fundamental thermodynamic and fluid mechanic principles. In the case of the centrifugal compressor there exists a series of ‘tried and tested’ formulae based on a combination of fundamental theory and empirical evidence that enables the designer to assess component geometry and fluid conditions rapidly. Some of these techniques will be discussed in more depth in chapters two and three, however, the following sections highlight the key fundamental formulae which define the 1-dimensional flow in the centrifugal compressor.
For clarity, each component in the compressor has been assigned a location or station number, given in table 1-1, which will be references throughout this research.

<table>
<thead>
<tr>
<th>Station number</th>
<th>0</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>6</th>
<th>7</th>
</tr>
</thead>
<tbody>
<tr>
<td>Component</td>
<td>Inlet casing</td>
<td>Inducer</td>
<td>Impeller exit</td>
<td>Diffuser inlet</td>
<td>Diffuser exit</td>
<td>Volute throat</td>
<td>Volute exit</td>
</tr>
</tbody>
</table>

Table 1-1

1.4.1 Inlet Casing

The inlet casing, station {0} is usually a converging nozzle cast as part of the volute casing and is generally made long enough to assume axial delivery and uniform velocity to the impeller. As the inlet casing is stationary, no energy transfer takes place between the casing and the air and the stagnation enthalpy remains constant. Losses incurred in the inlet casing are generally neglected in automotive turbocharger compressors since they are usually due to up-stream components such as air filters, also as casing designs vary, a theoretical prediction of the loss is not always possible.

1.4.2 Impeller

The most critical and geometrically complex component of the compressor is the impeller. It is the only rotating component and as such, transfers energy to the air resulting in an increase in pressure, velocity and enthalpy. Figure 1-8 shows a typical centrifugal compressor impeller, made up of a series of main vanes and shorter ‘splitter’ vanes.
The front section of the blades is known as the inducer \([1]\) and figure 1-9 shows a view looking down onto this section. The inducer is generally curved so that the air is directed as smoothly as possible into the impeller to avoid flow separation from the impeller blades.

The magnitude and direction of the air flow in the impeller is described by the use of velocity triangles. The air enters the compressor axially with an absolute velocity \(V_1\) shown in figure 1-10\(^{[18,19]}\). The rotational speed of the impeller is defined as \(U_1\) and the resultant vector between \(U_1\) and \(V_1\) is the velocity of the air relative to the rotating vane known as the relative velocity, \(W_1\).

The air angle, which the relative velocity makes with the axial direction, is \(\alpha_1\) and \(\beta_1\) is the vane angle to the axial direction. The vanes are usually pre-set for a particular air angle and the right match between \(\alpha_1\) and \(\beta_1\) will only be achieved when the mass flow rate is correct for a particular rotor speed. Typically \(\alpha_1\) and \(\beta_1\) are not the same and the difference between them is defined as incidence, \(i\) and is written:

\[
i = \alpha_1 - \beta_1\]

eqn. (1-4)
A key geometric feature of the impeller is the inlet annulus or cross-sectional area, $A_1$, (the narrowest point between the main blades often referred to as the inducer throat) which can be found from the relationship:

$$A_1 = \frac{\pi}{4}(D_{it}^2 - D_{hl}^2) \quad \text{eqn. (1-5)}$$

Where $D_{it}$ and $D_{hl}$ are the inducer tip and hub diameters respectively, shown in figure 1-5. The hub diameter is determined by several considerations, such as shaft size, bearing diameters, manufacturing techniques and the number of blades selected, and is often the initial starting point for a impeller wheel design. Careful consideration must also be given to the tip diameter where the inlet relative Mach number, $M_{tw1}$ is at its highest. As the tip diameter increases the blade speed, $U_1$ increases. With increasing tip diameter the fluid inlet velocity $W_1$ decreases to a minimum and then begins to increase again, producing the parabolic relationship shown in figure 1-11. Thus, for a given mass flow and rotational speed the inducer tip diameter is often chosen to give minimum relative Mach number $M_{tw1}$. By substituting equation impeller tip diameter (1-5) into the continuity equation, the relationship between geometry and fluid flow can be shown.

$$m = \rho_1 A_1 V_1 = \frac{\pi}{4}(D_{it}^2 - D_{hl}^2) \rho_1 V_1 \quad \text{eqn. (1-6)}$$

The ratio between the inducer tip diameter, and the impeller outer diameter, $D_{it}/D_2$ can be used as a guide by the designer to determine both the optimum impeller shape and peak performance and is often referred to as the 'Squareness' of the impeller. Optimum values of squareness are approximately 0.65-0.7 for peak efficiencies of moderate pressure ratio compressors (2.5:1) and approximately 0.6-0.75 for high-pressure ratio operation (3:1 & 4:1)[10].

![Figure 1-11 varying relative Mach number with impeller tip diameter](image-url)
1.4.2.1 **Backsweep & the Exit Velocity Triangle**

The earliest impellers were designed with radial vanes at the outer diameter\[5,19\], however, the increasing demand for higher boost pressures, wider mass flow ranges and higher levels of efficiency, along with improvements in materials, brought about the development of the backswept vane as shown in figure 1-8. The backsweep angle is ‘the angle of the vane to the radial direction of the impeller outer diameter’, \( \beta_2 \) (figure 1-9) usually ranging from 20° to 50° and opposite to the direction of rotation. Increasing the backsweep pushes the surge point of the characteristic to the left (lowering the mass flow rate), so widening the operating range\[18\] of the compressor. In order to determine the required impeller dimensions and rotational speeds it is typical to carry out calculations for a range of backsweep angles to determine the optimum. There are also an optimum number of vanes for any given geometry and this number depends on the chosen value of backsweep angle\[19\].

The exit velocity triangles are shown in figure 1-12. The impeller tip rotational speed \( U_2 \) and the relative velocity, \( W_2 \) ideally follows the direction of the blade and the absolute velocity \( V_2 \) completes the triangle. The exit gas angle, \( \alpha_3 \) is the angle between the absolute velocity and the radial direction. Two further components of the absolute velocity can also be drawn; \( V_{w2} \) and \( V_{r2} \) being the tangential and radial components of the absolute velocity respectively\[18\]. The exit gas angle, \( \alpha_3 \) can then be written as a ratio of the two velocity components:

\[
\tan \alpha_3 = \frac{V_{w2}}{V_{r2}} \tag{1-7}
\]

This ratio is very important as it is associated with the selection of the diffuser. It has been found from empirical data\[3\] that values of velocity ratio between 2 and 3 are suitable for vaneless diffusers and values between 3 and 4 favour the vaned diffuser. It is also accepted
that an increase in the backswep angle is beneficial to the diffuser as it reduces the whirl velocity $V_{w2}$, thus reducing the kinetic energy $(V^2_{w2}/2)$ at the inlet to the diffuser, so improving its performance by reducing associated losses.

### 1.4.2.2 Impeller Efficiency

The impeller efficiency can be defined in a similar way to the overall compressor efficiency and assuming $C_p = \text{constant}$, can be written:

$$
\eta_{\text{imp}} = \frac{h_{2s} - h_{01}}{h_2 - h_{01}} = \frac{T_{2s} - T_{01}}{T_2 - T_{01}}
$$

**eqn. (1-8)**

### 1.4.2.3 Impeller Work

The accurate prediction of impeller work is fundamental not only to the impeller design but to the overall performance of the compressor itself. The value of work can be derived from the Euler Equation which relates the specific work done to the change in tangential momentum in the form:

$$
\Delta h = U_2 V_{w2} - U_1 V_{w1}
$$

**eqn. (1-9)**

When the whirl velocity at the impeller inlet, $V_{w1} = 0$, the equation is reduced to:

$$
\Delta h = U_2 V_{w2}
$$

**eqn. (1-10)**

For the ideal radial vaned impeller the work remains constant as the flow reduces because $V_{w2}$ is always constant, shown in figure 1-13. But, for the backswep vane there is an increase in work as the flow decreases. This shows that the work absorbing capacity of the backswep impeller is much lower than that of a radial vaned impeller, however, the negative sloping characteristic gives an increase in stability across the range of mass flows making the backswep vane more desirable.
1.4.2.4 Slip

Due to the inertia of the air between the vanes, the air becomes reluctant to move around with the impeller and does not rotate at exactly the vane speed as would be expected. $W_2$ lags behind its expected position (directly in-line with the vane direction) as shown in figure 1-14. The distance between the expected and actual value of $W_2$, in the tangential direction, is called the 'Slip Velocity', $W_{61}$.

A considerable number of studies have been carried out into this phenomena and several empirical models have been presented which have long been established as the most appropriate...
methods for impeller work prediction. The earliest of these was an equation derived by Stanitz for radially-vaned impellers\textsuperscript{[5]} to describe a relation between $V_{w2}$, $U_2$ and the number of vanes, $n$ and is referred to as the 'Slip Factor', $\sigma$.

\[
\sigma = \frac{V_{w2}}{U_2} = 1 - \frac{0.63\pi}{n} \equiv \frac{(n-2)}{n} \tag{eqn. (1-11)}
\]

Wiesner\textsuperscript{[20]} reviewed various methods of estimating slip factor at design point and concluded that the equation determined by Busemann\textsuperscript{[21]}, which showed that for a given backsweep angle the impeller work increases with increasing blade number, figure 1-15, provided the most suitable slip factor prediction.

\[
\sigma = 1 - \left( \frac{\sqrt{\cos \beta_2}}{n^{0.7}} \right) \frac{V_{r2} \tan \beta_2}{U_2} \tag{eqn. (1-12)}
\]

Wislicenus\textsuperscript{[22]} verified this equation for a range of blade angles and numbers but introduced a limiting value of impeller radius ratios ($r_2/r_1$)\textsuperscript{[23]}, beyond which, the slip factor required a correction factor, $c_{\text{limit}}$. Dean\textsuperscript{[24]} showed that slip factor is not constant for a given impeller, but rises as mass flow function reduces and approaches a value of 1 as the flow angle approaches that of the blade angle.

Subsequently, Whitfield\textsuperscript{[25]} developed a slip factor prediction for off-design flow rates, based on the jet-wake model of Dean\textsuperscript{[24]} and slip factor theory presented by Wiesner\textsuperscript{[20]}. He assumed that the majority of impellers operate with flow separation and that equation (1-11) could be applied to this type of flow. Whitfield’s results are presented in figure 1-16, showing the slip factor to be parabolic in nature, with the minimum occurring at a point where incidence becomes negative.
Combining the work equation (1-10) and equation (1-11) a relationship between slip factor and enthalpy can be defined and is generally used by today’s designers to determine impeller work and is often referred to as the work coefficient:

\[ \Delta h = \sigma U_2^2 \quad \text{or} \quad \sigma = \frac{\Delta h}{U_2^2} \quad \text{eqn. (1-13)} \]

To reduce slip, splitter vanes, as shown in figure 1-8, are generally used to provide the largest available cross sectional area at the inducer for maximum flow, whilst providing a large number of vanes at exit.
1.4.2.5 Impeller Loss

Another commonly used approach to obtaining a 1-D solution for the flow inside the compressor is to determine the energy losses. By their nature, these losses are not constant values but vary from one compressor to another, depending on the application. From experimental and aerodynamic analysis the impeller losses have been classified into five categories\cite{26}:

- Friction losses: Disc and surface
- Diffusion and Blade loading losses due to boundary layer growth
- Incidence losses due to off-design performance
- Shock losses due to high Mach number
- Re-circulation losses

It is extremely difficult to determine all of the individual losses experimentally and in many cases especially in the initial design stage, it is easier to combine them into a single loss. Denton\cite{27} reviewed some of the categories, which make up impeller loss and suggested using an ‘entropy loss coefficient’ definition which could be used for rotating blade rows and is defined as:

\[
\zeta_s = \frac{T_2 \Delta s}{h_{o2} - h_{o1}} = \frac{T_2 \left[ C_p \ln \left( \frac{T_{o2}}{T_{o1}} \right) - R \ln \left( \frac{P_{o2}}{P_{o1}} \right) \right]}{h_{o2} - h_{o1}} \quad \text{eqn. (1-14)}
\]

Denton pointed out that at low speeds other definitions of loss coefficient approach the same value, but at higher Mach numbers, above 0.3, the difference between the different techniques is increasingly noticeable. Since automotive centrifugal compressor impellers can reach Mach numbers above 0.3, it is appropriate to use equation (1-14) to determine the impeller loss coefficient which will enable impellers of different designs to be compared.
1.4.3 Diffuser

There are two main types of diffuser used in turbochargers; the vaned and vaneless diffusers\cite{5,17}. Despite the simplicity of the vaned and vaneless diffuser the ability to predict the performance characteristic accurately is somewhat difficult due to the 3-D flow profiles and a tendency towards flow separation, due to viscous drag. If the flow is reduced sufficiently, the flow begins to deteriorate, causing flow separation. It is essential then, that the optimum diffuser design be used in the compressor so that the best overall performance characteristic can be established.

The vaned diffuser is usually used in turbochargers that require high-pressure ratios and efficiencies with lower flow ranges and are more commonly found on larger compressors. The overall rate of diffusion in the vaned diffuser is higher than that of the vaneless diffuser for the same radius ratio. The tangential component of velocity is reduced more quickly reducing the surface friction and producing a high peak efficiency, which can be several percentage points above the vaneless diffuser efficiency over the same flow range however, the efficient operating range is much reduced. The reduced tangential velocity results in a shorter flow path and a more compact diffuser. This gives the vaned diffuser a greater advantage where more compact compressor designs are required.

![Figure 1-17](image-url)  
**Figure 1-17** meridional and cross-sectional view of the vaneless diffuser showing the main geometric features and the direction of the flow path
Typically, the vaneless diffuser is used in automotive turbocharger compressors due to its broad operating range and low manufacturing cost. In its simplest form, the vaneless diffuser consists of two parallel walls giving increasing cross-sectional area with radius shown in figure 1-17. The vaneless diffuser generally has a constant width, $b_2$ that is determined by the impeller tip width. Usually there is a small section directly after the impeller that tapers in to meet the diffuser walls. This is the narrowest point in the diffuser ($D_3$) and is known as the diffuser pinch diameter, $D_3$. The flow area across this section is known as the diffuser throat, $A_3$ and is the controlling section of the passage, influencing the compressor characteristic in a similar way to the inducer throat area. The ratio of diffuser outer diameter to impeller tip diameter, $D_4/D_2$ is used as a non-dimensional criterion of the vaneless diffuser for determining efficiency, and is usually a value below 2.

1.4.3.1 Vector Triangle

Assuming that the friction between the air and diffuser walls can be neglected, then by the requirement of constant angular momentum at any radius, ‘a’ in the vaneless diffuser, the tangential component of the absolute velocity, $V_{wa}$ will be reduced. The radial component of the absolute velocity, $V_{ra}$ will also be reduced as the cross-sectional area increases, shown in figure 1-18. It can be shown that with constant width and constant angular momentum and assuming a constant density, $\rho_a = \rho_2$, the path traced out by the air is a logarithmic spiral and the angle at any radius in the vaneless diffuser can be written:

$$\tan \alpha = \frac{V_{w2}}{V_{r2}}$$

eqn. (1-15)

![Figure 1-18 velocity triangles of vaneless diffuser at different points in the flow path](image)
1.4.3.2 Diffuser Efficiency

As with the impeller, the vaneless diffuser efficiency can be defined as the ‘ratio of isentropic enthalpy rise to the actual enthalpy rise between the diffuser inlet and exit’. Typically the conditions at diffuser exit are of most interest and the equation can be written:

$$\eta_{\text{diff}} = \frac{h_{4s2} - h_2}{h_4 - h_2} = \frac{T_{4s2} - T_2}{T_4 - T_2}$$  \hspace{1cm} \text{eqn. (1-16)}

1.4.3.3 Diffuser Losses

There are two frequently used methods of determining diffuser loss between different designs; the ‘stagnation pressure loss’ or ‘static pressure rise’ coefficients\[28\]. The diffuser pressure rise characteristic is defined as a non-dimensional ratio of ‘static pressure rise between the inlet and outlet of the passage to the inlet dynamic head’, more commonly called the pressure recovery coefficient, $C_p$:

$$C_p = \frac{(P_4 - P_3)}{(P_{03} - P_3)}$$  \hspace{1cm} \text{eqn. (1-17)}

The value of $C_p$ does not take into account the ideal pressure recovery coefficient $C_{pri}$ which relates the pressure recovery coefficient and diffuser area ratio. $C_{pri}$ can be defined:

$$C_{pri} = 1 - (1/AR^2) = 1 - (1/DR^2)$$  \hspace{1cm} \text{eqn. (1-18)}

The stagnation pressure loss coefficient is defined as the ratio of ‘the loss in stagnation pressure across the diffuser divided by the inlet dynamic head’ and an be written:

$$\zeta = \frac{(P_{03} - P_4)}{(P_{03} - P_3)}$$  \hspace{1cm} \text{eqn. (1-19)}

From $p = P/RT$ and (1-18), it can be shown that relationship between $C_{pri}$ and $\zeta$ can be written:

$$C_p = \left[1 - \left(\frac{D_4}{D_3}\right)^2\right]\left[1 - \frac{P_{03} - P_4}{P_{03} - P_3}\right] = (1 - \zeta)C_{pri}$$  \hspace{1cm} \text{eqn. (1-20)}
Rogers\cite{28} produced a consistently accurate correlation between pressure recovery and inlet gas angle for varying width to diameter ratios (WDR) of a specific compressor. He presented an equation for the stagnation pressure loss coefficient expressed in terms of a skin friction factor $C_f$ based on a series of pressure recovery plots.

$$\zeta = \left( C_f \frac{D_4}{b_3} \right) \left( \frac{b_3}{D_4} \right) \left( 1 - \frac{D_3}{D_4} \right)$$

eqn. (1-21)

Generally, skin friction values of between 0.005 and 0.01 are used. Early loss modelling assumed a single value of friction factor, however, this value does vary with compressor size, surface roughness, and Reynolds number.

### 1.4.4 Volute

The volute or compressor housing is typically made in cast aluminium; both gravity die and sand casting techniques being used in its manufacture. Highly accurate profile machining is required to fit the impeller to the internal geometry of the casing as accurate installation is important to achieve performance consistency.

In the turbocharger centrifugal compressor with a vaneless diffuser, the volute plays a significant role in determining the overall performance characteristic, yet a relatively small amount of research has taken place in volute design compared to impellers and diffusers\cite{29}. Many of the volute designs have remained unchanged in decades and since there is only a small gain in pressure ratio in this component, the volute is generally treated as a simple means of directing the air flow and not as an additional component to improve performance.

Figure 1-19 shows two typical volute symmetries with 1-19(a) being the most commonly used in turbocharger compressors. The volute shown in figure 1-20 is essentially a pipe of varying cross-sectional area rolled around the diffuser\cite{15}. It can be divided into two sections; the first section is the scroll,
which collects the air as it exits the diffuser shown in figures 1-20 and 1-21. The second section is the exit diffuser shown in figure 1-20, where a small amount of diffusion takes place, although a significantly smaller amount than in the diffuser, before the air leaves the compressor. It is expected that even at maximum efficiency the volute will lose 10% of the energy increase produced by the impeller\(^{16}\).

The tongue (shown in figure 1-20) is the narrowest point in the volute and is a point at which flow instabilities can occur resulting in flow separation. Figure 1-22 shows how variations in the exit angle from the diffuser can cause flow separation if the flow angle does not match the volute spiral shape. It is therefore, important to ensure that the design of the volute closely matches the angle of the air exiting the diffuser. In practice, a small gap is left between the tongue and the diffuser exit to enable the airflow to equalise. The size of this gap determines the maximum flow through the compressor, particularly at higher pressure ratios and higher rotational speeds.
If friction in the scroll is neglected, then the volute can be designed so that the static pressure is uniform throughout the circumference and the angular momentum continues to remain constant from the diffuser exit.

One approach to designing a volute is to use the following geometric features:

- The cross-sectional area \( A \), of the volute scroll.
- The mean radius \( \bar{r} \), to the centre of the cross-section.
- The azimuth angle \( \phi \), around the volute scroll.

Knowing the exit velocity from the diffuser and \( \bar{r} \), and calculating \( \dot{m} \) from the volumetric flow rate \( \dot{V} \), the area ratio, \( \frac{A}{\bar{r}} \) can be found from which the area at any cross-section \( n \) can be determined, giving:

\[
\frac{\dot{V} \phi}{2\pi k} = \frac{A_n}{\bar{r}} = \frac{A_n}{r_n}
\]

The frictional losses associated with the volute are quite small compared to the diffuser and impeller and generally, the losses are neglected. The primary form of loss in the volute is due to turbulent mixing as the flow from the diffuser enters the spiral. Loss due to friction is also considered and modified forms of the simple pipe loss equation have been produced to take account of this, however, in large volutes this is usually neglected.
1.5 Conclusion

The operation and configuration of a typical automotive turbocharger compressor has been presented. It has been shown that one-dimensional flow assumptions have formed the backbone of modern-day turbocharger compressor analysis for many decades and their usefulness in determining the performance of a compressor is in no doubt.

The benefits of combining these well tested methods into a concise performance prediction tool has long been the goal of many turbomachinery experts, and whilst some models do exist, and will be looked at in detail in the ensuing chapters, the majority have been developed for particular functions. It is believed that by application of the simple assumptions presented in this chapter combined with the wealth of theoretical and experimental knowledge accumulated over several decades, along with modern computational analysis, a significant improvement to existing 1-D prediction techniques can be made.

The following chapter will place this research work into context with an examination of some of the many myriad works relating to 1-D modelling techniques. It will shows how the assumptions presented in this chapter have been applied, with attention focused on the diffusion system components; the vaneless diffuser and volute casing. From this a case is presented to support the research work along with the key points for considerations during its implementation. Finally, an outline of the chapter content is given as a guide to the research process.
2 State of Art Literature Review

Synopsis
A review of state-of-art literature is presented, focusing on the performance, design, fluid flow and geometry of the vaneless diffuser and volute casing. Also included in this chapter are an examination of the effects of boundary layer growth, surge and choke on the performance of centrifugal compressors. This chapter will show how the theory, reviewed in chapter one, has been applied in previous research and how this work will support the hypothesis of the research project. The objective of this research are presented at the end of this chapter along with supporting justifications and an overview of subsequent chapters.
2.1 Introduction

Much of the research carried out over the last 30 years has been focused on the gas turbine market, mainly for power generation and aerospace applications. Increasingly, many turbomachinery manufacturers are turning to academic institutions to assist in the design and development of these highly complex and expensive components and companies such as Rolls Royce have formed several partnerships with academia to investigate a range of subjects including turbine cascade aerodynamics, fatigue and blade icing. Likewise, Solar Turbines and QinetiQ have both been working alongside university research facilities, developing gas turbines as alternative power sources as it is foreseen that the market for robust, cost-effective gas turbine power plants will be greatly in demand over the next 20-30 years.

This increase in research has not only been confined to the gas turbine industry. Whilst the turbocharger has remained largely unchanged in it basic shape over the last half century, manufacturers have been seeking ways to improve the performance and efficiency of the turbocharger and its components. Considerable and steady progress has been made industry-wide advancements such as improved sealing techniques and reduction in component vibration have led to significantly improved in performance. According to the report on the turbocharger global market in 2004, 62% of all new passenger cars made for the European market were fitted with turbochargers. Even with the tightening of engine emission regulations, turbocharger manufacturers see no future let-up in demand for their products.

Developments in manufacturing methods, materials and design techniques have pushed the turbocharger efficiency from approximately 40% up to the mid to high 60% region with pressure ratios increasing by approximately the same percentage. Figure 2-1 shows the average decade-on-decade increase in the pressure ratio’s of ABB turbochargers over the last 40 years, clearly illustrating how demand for power and improvements in technology have effected the turbocharger market.
However, today’s manufacturers are not solely concerned with the improvement of boost pressures and engine performance. Issues of energy conservation, the environmental impact of the engine and the introduction of ‘Miller Timing’ to engine operation are playing a significant role in the design and development of new turbocharger technologies. Whilst Amos\textsuperscript{[36]} makes it clear that it is difficult to predict where the next stage of turbocharger development will come from, indications from industry suggest that assisted turbocharging, new centre housing and bearing systems and enhanced compressor performance, including combined axial and radial compressor stages, are all areas requiring considerable investment.

A large amount of study has been directed towards the turbocharger turbine over the years, mainly due to the high thermal loading and extreme stresses it is required to handle. During the 1990’s several cutting-edge breakthroughs in technology were made\textsuperscript{[36]} in turbine design, including variable geometry (VG) guide vanes\textsuperscript{[39]}, and electronic control of the waste-gating system, enabling improved turbine operating range and controllability. Turbocharger manufacturers have also taken advantage of the additional expertise available at academic institutions and several universities have been carrying out investigations into radial and mixed-flow turbine rotor design\textsuperscript{[40,41]}, performance improvements\textsuperscript{[42]} and developing an understanding of flow instabilities\textsuperscript{[43]}.  

Figure 2-1  Statistical mean compressor pressure ratio of ABB turbochargers delivered for two- and four-stroke engines over the last 45 years. [ABB website]
Initially research into compressor performance, specifically those used on medium-speed engines, studied the components in isolation, with a substantial proportion of the work looking at the impeller and to a lesser extent the vaned diffuser. With a desire to make the centrifugal compressor more compact whilst maintaining - and even increasing - the amount of compression, automotive turbocharger manufacturers have concentrated considerable effort into improving impeller performance by developing the compressor as a complete system\[12,13,26,44\]. The reason for this are primarily to do with manufacturers moving from designing compressors for a particular application to a system of turbocharger ‘families’, where a single impeller is used with a series of different sized diffusers/volute casings. In addition, the need for higher boost pressures has led designers to seek out new materials for the impeller, which could cope with the increasingly high stresses. The use of titanium alloy\[2,45\] for example has led to the development of new machining and casting techniques and consequently improvements in high volume/low cost production.

Flow conditions and geometry have, in chapter 1, been shown to be inherently linked and considerable effort has been made over the last 30 years to investigate and develop techniques which enable the geometry of the compressor to be matched appropriately to the air flow\[46,47\], thanks in part to the development of new computational analyses and the improvements in machining techniques\[2,48\], the former being discussed in more detail in chapter three. A greater understanding of the instabilities in the compressor stage has led to improvements in the way surge can be predicted, resulting in modern compressors having much wider flow ranges than their predecessors.

However, a relatively modest amount of research has been carried out on the vaneless diffuser and volute casing in the last 30 years, in comparison to that of the impeller\[49,50,51,52\]. Despite the relatively simple geometric shapes of the two components, the complex numerical analysis required to analyse the flow has, until the advent of CFD, been extremely difficult to carry out. Coupled with the assumption that the impeller was the ‘performance critical’ component and the diffusion system, whilst vital, was of secondary importance, has resulted in relatively little change in the component’s design.

However, in recent years, with engines becoming smaller whilst power demands increase\[43,53\], designers have been forced to turn their attention back to the diffusion system
components in an attempted to improve performance whilst reducing their overall size. As stated earlier, the use of a range of casing sizes for a single compressor has allowed the designer more scope, not only in dealing with size and weight issues but enabling him to ‘tune’ the compressor’s performance by selecting the most appropriate components for a particular application. This increased flexibility, especially at the initial design stage, also has its disadvantages. The additional time required to optimise the compressor design; to find the overall efficiency required using the most appropriate components whilst ensuring the device fits the client's requirements, can be a laborious and iterative process. Again, the application of three-dimensional fluid and stress analysis software has greatly assisted this process but the designer is still required to make a series of adjustments in order to select the most appropriate components.

The problem then is two-fold, firstly further examination of the diffusion system components are needed to provide a greater understanding of their behaviour and secondly development of rapid analysis tools are required to provide the designer with a quick and simple method of matching components, especially at the initial design stage, using well-established and available relationships between geometry and flow conditions like those presented in chapter one.

In order to understand the existing processes applied to modern turbocharger design, and in particular those applicable to the diffusion system, it is necessary to examine the historically significant developments which have contributed to today’s state-of-art. Whilst a full literature survey of this vast field is beyond the scope of this research, this chapter examines some of the notable developments in both diffuser and volute research and puts into context the nature of this research project.

It will demonstrate that considerable study of the two compressor components has been carried out and much of this work can be used to validate and support the experimental and numerical analyses of the present research project. But it will also show that few comprehensive techniques exist which encompass the compressor as a complete unit and are generic enough to be applied to any centrifugal compressor design.
2.2 Diffusers

2.2.1 Channel Diffusers

Much of what is known about fluid flow in centrifugal compressor diffusers originated from the study of flow in straight pipes and channel diffusers similar to those shown in figure 2-2, particularly straight-walled, figure 2-2(d), and conical diffusers figure 2-2(a) & (b), the former having undergone detailed examination over many years, looking at changes in flow profile, surface roughness and efficiency\(^{[16]}\) with several key works highlighting the relationships between geometry and performance\(^{[54-57]}\).

Since channel diffuser correlations have been used to good effect on vaned diffuser models, by creating an 'equivalent' geometry similar to that of the channel shown in figure 2-3, it is postulated that the same techniques could be applied to vaneless diffusers.

\(^{1}\) Additional information can be found in appendix I page I-1 on this subject
Kline et al\textsuperscript{[55]} presented one of the earliest studies on straight-walled diffusers. They showed that there were four main geometrical parameters that could be used to describe the geometry of the straight-walled diffuser, those being:

- Length to width ratio $L/W_1$ (LWR)
- Aspect ratio $b/W_1$ (AS)
- Area ratio $A_2/A_1$ (AR)
- Divergence angle ($\theta$)

They also proposed four important optimisation points for good design of a straight-walled diffuser:

- Minimise losses
- Maximise pressure recovery ($C_{pr}$) for a given AR (regardless of the length)
- Optimise $C_{pr}$ for a given length
- Optimise $C_{pr}$ for any geometry given specific inlet conditions

Using the relationship between geometric parameters and performance, Kline et al\textsuperscript{[55]} formulated a set of ‘optimum pressure recovery coefficient’ criteria for each of the geometric parameters which were also corroborated by other authors\textsuperscript{[56,57,58]}.\textsuperscript{[1-1]}

- For a given AR, as long as the design is below the line of appreciable stall, any length can be used without affecting the $C_{pr}$.\textsuperscript{[55,56,58]}
- For a given LWR, the optimum $C_{pr}$ can be found as $\theta$ increases just above the line of appreciable stall.\textsuperscript{[55,56,57]}
- For any geometry the value of $C_{pr}$ increases uniformly as LWR increases and $\theta$ decreases.\textsuperscript{[55]}
- For varying LWR, the maximum efficiency occurred when $6^\circ \geq 2\theta \leq 7^\circ$.\textsuperscript{[55,56,57]}
- Maximum $C_{pr}$ occurs at high AR’s for long diffusers with low $\theta$ and depends on the boundary layer thickness and Reynolds number.\textsuperscript{[55,56,58]}
- $C_{pr}$ is changed if there are flow obstructions in the flow up-stream of the diffuser throat and the magnitude of $C_{pr}$ depends on the type of obstruction and its location.\textsuperscript{[57]}
A graphical example of their findings is shown in figure 2-4 with actual and ideal pressure recovery $C_{pr}$ and $C_{pri}$, effectiveness $\eta_p = C_{pr}/C_{pri}$, and head loss due to dissipation $H_L$ against varying $\theta_0$ for a fixed LWR.

Figure 2-4 shows that as $\theta_0$ increases $H_L$ rises and $C_{pr}$ also rises to a peak where $\eta_p$ is at a peak. The relationships between loss, recovery and efficiency in the vaneless diffuser have been established in chapter one and it is clear that these type of concise design rules are equally as applicable to this research.

Reneau et al.\cite{59} and [I-i] presented similar findings in the form of contour maps (figure 2-5) of $C_{pr}$ for varying AR, LWR and $\theta_0$ with fixed values of boundary layer thickness, $2\delta^*/W_1$ (represented as a fraction of diffuser inlet area blocked by boundary layer) these maps are referred to in many channel diffuser studies and are still referred to in diffuser analysis today. No reference has been found as to their usefulness in assessing compressor vaneless diffusers, but it is envisaged that equivalent geometric data could be validated using these maps.
The most notable work on channel diffusers was carried out by Runstadler et al\cite{runstadler} who combined the work mentioned above\cite{halleen, reneau, renau} with work by Halleen et al\cite{halleen} to produce a detailed study of flow, Mach number and pressure recovery coefficient. Their investigations looked at the changes in $C_{pr}$ over a range of high inlet Mach Numbers. From their experiments they produced a series of plots of $C_{pr}$ and blockage against Mach number, for fixed aspect ratio $AS$, as a function of the geometric features, 20, LWR and AR shown in figure 2-6. They verified Reneau’s results and demonstrated that throat blockage factor was the most critical parameter in correlating diffuser performance, showing that increases in blockage factor reduced $C_{pr}$.

Aspect ratio was also important in establishing the pressure recovery coefficient of the diffuser. They showed that at low $AS$ ($< 1$), $C_{pr}$ increased rapidly due to the fact that at zero aspect ratio $C_{pr}$ was zero and that maximum $C_{pr}$ occurred below an $AS$ of 1. Since blockage factor plays an equally important role in the performance of the vaneless diffuser\cite{halleen, vil} it would be appropriate to compare any vaneless diffuser data with these available data.
2.2.2 Curved and Vaned Diffusers

The fundamental design requirement of the centrifugal compressor diffuser is that the diffusion takes place in the shortest possible length. It is for this reason that curved passages, which decelerate and turn the fluid at the same time, were originally selected for the vaned diffuser.

An illustration of early curved diffuser design as shown in figure 2-7 [62]. These curved diffusers utilise the smallest space whilst still accommodating a long diffuser channel [63]. The experimental methodology used was similar to that of the straight-walled diffuser, however, due to the curvature of the passage, the effects of pressure gradients across the channel become more significant. Fox et al. [62] investigated the effect of turning angle $\beta$ (the angle through which the fluid is turned from the inlet measured at the diffuser centre line) on flow regime by varying the angle from $0^\circ$
≤β≤ 90° by 10° intervals. The results showed many similarities between the curved and straight diffusers with two main points being:

- For diffusers with a curved arc centre line and linear area distribution the line of appreciable stall was unaffected by turning through 30° or less.
- As the radius of curvature of the inner wall decreased the flow along the inner wall tends to remain laminar.

Since the flow in the vaneless diffuser follows a log spiral path these data from curved channel diffusers more realistically describes the flow and would be invaluable in validating the vaneless diffuser flow conditions.

The correlations found between geometry and performance were applied by Clements & Artt \cite{64,65} to two studies on turbocharger compressor vaned diffusers of a straight-wedge design. The first of these looked at how the vaned diffuser geometry influenced the efficiency and flow range of the compressor and the second looked at the influence of LWR on the compressor efficiency. This work was compared to the straight-walled diffuser data and the authors found that:

- \(C_p\) increased with increasing channel divergence up to a maximum of 12°, above which little more was gained in recovery. This differed slightly from the straight-walled diffuser divergence angle found by Reid\cite{56} & Reneau\cite{59} of 7° but was consistent.
- Channels of identical AR produce the same \(C_p\) irrespective of sidewall divergence angle, which was identical to straight-wall diffusers.
- At surge, as divergence angle increased the blockage decreased. This was related to the pressure gradient, in that as the boundary layer thickened there was a greater tendency to separate if the pressure gradient was low.
- Pressure gradient was increased as \(20\) increased so reducing the blockage.

Existing vaned diffuser analysis of geometry and performance, whilst not directly comparable, will go some way to validating the data from vaneless diffuser.
2.2.3 Summary
The work on channel diffusers has been widely used in performance prediction of vaned diffuser mainly due to the similar geometric features of the two types of diffuser. Clearly the relationship between straight-walled and curved diffuser geometry and performance has been thoroughly investigated and considerable amounts of empirical data are available. It is speculated that these data would provide a suitable datum from which to validate experimental data from vaneless diffusers and is discussed later in this thesis.

2.2.4 Vaneless Diffusers – flow distortion and component interaction
A limited amount of research has been generated over the last 40 years with regard to radial vaneless diffusers. Periodically focus has retuned to the component as advancements in technology have been made; where traversing stagnation pressure probes and laser-doppler anemometry have enabled more precise measurements of the flow to be made. Much of the work has studied the steady and unsteady flow phenomenon\cite{66,67} generated at the diffuser inlet by the rotation of the impeller and the majority of this work has involved extensive experimental analysis.

![Figure 2-8 position of jet/wake flow at impeller exit](image)

Selecting the optimum impeller and diffuser to give the best overall performance can be as difficult as turbocharger/engine matching. The close proximity of the impeller and diffuser requires particular attention, as the highly distorted flow from the impeller severely influences the pressure recovery of the diffuser. Several notable works have been produced on the subject\cite{24,66-70} and in one influential work, Dean et al\cite{24} investigated unsteady and asymmetric flow patterns. To describe the non-uniformity of the flow, the authors broke the impeller exit flow into two distinct regions, ‘Jet’ & ‘Wake’, shown in figure 2-8.
The authors concluded that the flow distortions created by the impeller could have a significant effect on the performance of the vaneless diffuser and the size of that effect is dependant on:

- Impeller exit angle
- Diffuser wall friction
- Work and flow coefficients (although this was not investigated)
- The level of flow distortion
- Diffuser geometry

Also

- The mixing shear stresses between the jet and wake were less important than the wall shear stresses in governing the flow behaviour and wall shear stress data for flat plates correlated well with the vaneless diffuser.
- Stagnation pressure loss coefficient for the diffuser was lower for a distorted than for an undistorted flow.

Krain\textsuperscript{[67]} and Fisher et al\textsuperscript{[68]} both carried out analyses on the flow distortion, but on vaned diffusers. Whilst the flow in the vaned diffuser is different from that of the vaneless diffuser the flow distortion require consideration as it is clear from Dean\textsuperscript{[24]} that the distortions are carried some way into the diffuser from the impeller. Krain's analysis of a radial impeller with two different vaned diffusers found that both of the diffusers had similar flow profiles within the discharge region of the impeller. Using Laser velocimeter method Krain recorded the magnitude and direction of the flow between the impeller exit and diffuser inlet.

Dean\textsuperscript{[24]} proposed that distortion in the flow from the impeller in the radial direction should be mixed out relatively quickly, however, flow distortions in the meridional direction took somewhat longer to mix out. Krain showed that the flow pattern from hub to shroud was slightly twisted and that the flow angle variation at the diffuser throat compared to that at the diffuser leading edge varied up to 13°. The results for the vaned diffuser showed there was no mixing-out of the flow right up to the diffuser throat shown in figure 2-9. This it
was believed, was due to the highly distorted flow from the impeller and the angle of the diffuser vanes and is supported by Elder et al.\cite{71} who suggested that diffusers with larger vane passages are less likely to become blocked at high flow rates.

Figures 2-9 and 2-10 show how flow enters the diffuser as the impeller blades pass by. The inclusion of a vaneless space before the vaned diffuser acts to reduce the diffuser inlet Mach number and reduce the turbulence of the flow. The disadvantage of such a vaneless space increases the overall size of the compressor and encourages boundary layer growth in the diffuser.

Since the geometry of the diffuser effects the flow behaviour most of the studies carried out have looked at differing designs of diffuser to assess the most successful shapes. Jansen's\cite{66} work developed from the Dean et al.\cite{24} analysis, but looked at the boundary layer growth and its effect on pressure loss coefficient and efficiency along the walls of an isolated vaneless diffuser. He categorised the flow into two regimes symmetric and un-symmetric and demonstrated that whilst the theoretical and experimental results showed good correlation the experiment was only valid for low speed flows where an axi-symmetric inlet flow was obtained. He highlighted some of the factors affecting pressure loss in a vaneless diffuser as:

- Unsteady and unsymmetrical flow at the diffuser inlet and into the diffuser were caused by compressor blade wakes resulting in skewed flow profiles.
- Boundary layer growth and flow separation were caused by wall shear stresses.

Most of this work has required considerable amounts of experimental analysis, as emulating the flow purely numerically or by artificial means is extremely difficult. One prime example of how emulation does not work is shown by Jansen\cite{66} who used a rotating screen to induce swirl. It was found that at higher speeds the only way to create a sufficiently swirling flow would be by using an impeller.
Ludtke[72] investigated the changes in flow, surge, choke and efficiency over a range of Mach numbers (0.94 to 1.07), on a four-stage compressor. Focusing his attention to the first stage the work looked at the behaviour of the vaneless diffuser relative to the first stage impeller. A selection of vaneless diffuser designs shown in figure 2-11 were tested.

The author asked two pertinent questions of the investigation:

- 'Could the surge limit be improved without changing the impeller?'
- 'Could the performance curve be shifted to smaller volume flows without changing the impeller?'

By maintaining the same impeller throughout the experimental process, it was possible to determine the effect the diffuser had on the stage characteristic. This would seem a sensible approach and one that would be applicable to this research, since improving the impeller characteristic is not the main focus for this project.

The diffusers investigated included two parallel-walled types, one with a reduced area compared to the other, two tapered-wall types, one with a highly tapered passage and the other with a constant meridional area from inlet to outlet. It was shown that it was possible to move the overall surge point by varying the diffuser shapes, but the different designs had little effect on the overall performance characteristic of the stage at either full or part load. This the author suggested was due to the relatively small flow range of the unshrouded impeller compared to a backswept impeller at Mach numbers around 1.
Yingkang et al\cite{731} also investigated the aerodynamic characteristics of a series of vaneless diffusers with diverging and converging passage designs ranging from $-3^\circ$ to $8.2^\circ$ respectively. In conjunction with these diffusers, five different impeller rotors of varying blade number, inducer and backsweep angle were also used. The impeller geometries are given in table 2-1:

<table>
<thead>
<tr>
<th>Impeller</th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>E</th>
</tr>
</thead>
<tbody>
<tr>
<td>No of blades</td>
<td>24</td>
<td>12</td>
<td>12</td>
<td>12</td>
<td>12</td>
</tr>
<tr>
<td>$\beta_1$ deg (measured from the tangential)</td>
<td>55</td>
<td>55</td>
<td>55</td>
<td>55</td>
<td>30</td>
</tr>
<tr>
<td>$\beta_2$ deg (measured from the tangential)</td>
<td>30</td>
<td>30</td>
<td>60</td>
<td>90</td>
<td>60</td>
</tr>
</tbody>
</table>

Table 2-1

They produced a correlation for pressure recovery $C_{pr}$, and stagnation pressure loss coefficient $\Omega$, against diffuser inlet flow angle $\alpha_3$ for the parallel walled diffuser, shown in figure 2-12. The two figures indicate similar levels of loss and recovery for the parallel-walled diffuser with each impeller, impeller rotor ‘A’ giving the best performance.

![Figure 2-12 variation of $C_{pr}$ and Loss coefficient with diffuser flow angle \cite{731}](image)

Figure 2-13 also shows the effect of diffuser geometry on pressure recovery coefficient (rotor B for diffusers 1-4 and rotor A for diffuser 5). This shows that the highest peak recovery was produced by the $5.5^\circ$ convergent diffuser. They concluded that the diffuser performance was not very sensitive to changes in impeller blade number and backsweep angle as shown in figure 2-12.
They also confirmed that a small amount of sidewall convergence, approximately $5^\circ$, resulted in a negatively sloping pressure recovery curve, which improved the performance, stability and surge margin of the vaneless diffuser shown in figure. This fully supported the findings of Ludtke\cite{72} whose sidewall convergence also produced some improvement to $C_p$. Typically, automotive vaneless diffusers are of a parallel-walled design and as has been shown here, converging and diverging passages provide little in the way of improvement to performance.

These works highlight the difficulty met by the designer in determining the flow conditions inside the vaneless diffuser. Extensive and laborious experimentation is required for what could be considered as ‘limited results’. It is believed that because of this, the vaneless diffuser has been somewhat neglected over the last 20 years and little has been done to improve its design or understand further its behaviour. However, technological advancement has encouraged some further analysis to be carried out over the last decade.

Aungier\cite{74} applied a computer based mathematical model of the diffuser using mean-line performance models to assess the design. Comparisons with experimental data did produce good results and he was able to conclude that the optimised design of the diffuser yielded an exit flow angle of $30^\circ$ to $35^\circ$ however the need for experimental validation was still required.
More recently Japikse et al's work\cite{75,76} has added considerable weight to the need for increased analysis of the vaneless diffuser. They point out that little has been achieved in diffuser development since the jet/wake model of Dean et al, and indeed the limited relevant research presented here supports that case. They suggest that whilst 'single zone' or mean-line flow assumptions do not truly predict the flow conditions in the vaneless diffuser (which has been the mainstay of the turbomachinery industry for over 50 years) the results are accommodating enough to enable good correlation between experimental and predicted performance. They conclude that the way forward in improving the prediction of performance in a vaneless diffuser would be to return the to the jet/wake model of Dean from which they present a new diffuser design based on their findings.

2.2.5 Conclusion

Whilst some notable work has been produced on the vaneless diffuser, particularly on flow instability, limited evidence can be found relating geometry to performance certainly compared to that available for channel and vaned diffusers. It is anticipated that an analysis of the vaneless diffuser geometry would be required to determine some form of equivalent geometry, which could then be validated by channel or vaned diffuser data. It is clear that impeller/diffuser interaction would have to be addressed particularly in experimental testing, where the position of instrumentation is of critical importance. It is believed that whilst Japikse argues a good case for the application of a 'two zone' models, the need to improve existing mean-line techniques using modern methods needs to be addressed.
2.3 Voluttes

The problems associated with volute design stem from a lack of understanding of the differences between the conditions that give uniform flow and best efficiency, particularly for compressors with backswept impellers [77]. It is therefore necessary to examine the volute geometry and look at some earlier techniques used in determining the volute performance.

Brown et al [30] investigated the design of a family of diffusing volutes for use with a mixed-flow impeller and vaneless diffuser. From available data, they showed that there was a relationship between the diffuser diameter and compressor performance, which indicated that:

- Diffuser efficiency decreased with a reduction in diameter.
- Diffuser efficiency was proportional to the losses at the volute inlet.

Thus, by increasing the diffusing ability of the volute, a smaller up-stream diffuser could be used and the total losses could be reduced. The design criteria assumed a uniform pressure around the scroll inlet. This determined the position of the volute cross-sections for pre-assigned areas, by considering the radial variation in density and friction. The width of the scroll was also assumed to be a function of the radial distance only and not the azimuth angle. Each design had a different divergence angle (between the scroll walls) shown in figure 2-14 and several had been manufactured using different processes to alter the surface roughness.
The authors provided a mathematical analysis of the radial variations in density and used this to find the location of the cross-sectional positions in the scroll. The following procedure was applied:

1. Choose a cross-sectional shape for the scroll.
2. Find the width and radius ratio of chosen scroll.
3. Calculate the density ratio from the diffuser exit conditions.
4. Determine the location of the scroll sections.
5. Determine the friction coefficient and azimuth angle for each cross-section.

It was found that there was a small amount of variation in the performance between the scrolls tested, with the peak efficiencies varying slightly for each scroll.

Early experiments suggested that the flow pattern in the volute changed as the mass flow rate changed\(^{[19]}\) shown in figure 2-15 and that at the design point in a symmetrical volute, a double vortex was formed. It was suggested that an asymmetrical volute shown in figure 2-14 might produce a higher efficiency than a symmetrical one figure 2-15, due to the lack of these vortices. Brown et al\(^{[20]}\) however, found no significant difference between the symmetric and asymmetric scrolls. It was suggested that the small differences in efficiency were probably due to the miss-matching of the diffuser or impeller to the volute. When the volute scroll was replaced with a diffusing scroll, an increase in efficiency of 8 to 12 points was noted and the operating range was considerably extended.

![Figure 2-15](image_url) variation in flow pattern with change in mass flow rate [30]
The design of the volute relies heavily on the uniformity of the flow exiting the diffuser especially in the circumferential direction. Most of the losses are associated with the volute's inability to utilise the radial kinetic energy and this can lower its efficiency by 2 to 5%. Because this energy cannot be diverted into the circumferential direction, it is dissipated. An explanation for this can be demonstrated if one looks at points on a constant speed line shown in figure 2-16. Point one is close to the uniform pressure line denoting that the design point has been achieved. Point two is close to the surge line, here the radial velocity is less than that at the design point and therefore the scroll is too large, the flow will decelerate and the pressure will rise in the spiral. Point three has a much higher radial velocity and therefore the flow is too great and the scroll too small thus accelerating the flow. This means that for a given rotational speed there is only one mass flow rate that will give uniform static pressure but this may not be the peak efficiency. Cumpsty[77] suggested that if the design constraint of uniformity was removed then a decrease in the circumferential pressure would allow the flow to be directed in the tangential direction and the energy of the radial component could be utilised.
Stiefel[78] tested a compressor with the same impeller and vaneless diffuser but with two different volutes. The one producing the left-hand map in figure 2-17, being 30% larger than the right-hand map. This produced extremely different performance characteristic curves. He noted that at higher pressure ratios (above 3) the smaller compressor performed better. Since the impeller and diffuser were the same in both cases, this would suggest an improved volute performance i.e. a smaller volute than that designed for uniform pressure is generally more efficient. Increasing the size resulted in an increase in static pressure around the spiral causing flow re-circulation and huge energy losses.

Stiefel[78] noted that on the performance curve the line of peak efficiency did not always correspond to the line of uniform pressure, this lead to a compressor with a very low efficiency even at it’s given pressure ratio. He concluded that in order to gain the highest efficiency from the compressor the volute cross-section should be between 10 and 15% less than that which gives uniform pressure around the circumference. A volute of this kind would give a drop in static pressure but there would be no significant change in stagnation pressure. Stiefel’s analysis according to Cumpsty gave a much distorted view of the optimum volute and generally resulted in a choice of volute unsuited to its application. This poor performance at the design point has according to Cumpsty, been attributed to viscous
effects in the volute. However, the low velocities compared to those in the upstream section of the compressor are negligible and do not justify such considerations.

Sideris et al.\(^{[79]}\) investigated volute off-design operation and performance and measured the non-uniformity of the flow and pressure distortion and its effects on the up-stream side of the volute. Their theoretical model was based on the rotating stall model of Frigne et al.\(^{[80]}\) but was adapted to calculate off-design conditions.

The authors showed that at high mass flow rates, the volute could not handle the fluid and the flow was re-accelerated towards the exit. This resulted in a decrease in pressure between the tongue (at 90°) and the exit (at 360°) figure 2-18. At low mass flow rates the volute was too large for the flow and acted as a diffuser, increasing the static pressure between the tongue and the exit. The pressure variation around diffuser inlet and exit and theoretical prediction are shown in figure 2-18. It is also noted that there are differences in pressure between the diffuser hub and shroud but these are attributed to the non-uniformity of flow from the impeller exit.

Iso-pressure contour plots of the diffuser show that the volute exerts a great deal of influence on the up-stream components of the compressor. At choke, (maximum flow in figure 2-19) the iso-pressure contours are extremely distorted throughout
the diffuser due to the large circumferential pressure gradients. At surge, (minimum flow in figure 2-19) the pressure lines are less distorted but there is still noticeable distortion at the volute tongue. At the medium flow, the iso-pressure lines are almost concentric circles indicating a uniform flow field within the diffuser. The authors suggested that improvements to the theoretical model could be made with the addition of a 3-D boundary layer model. It is clear that when considering the flow uniformity of the volute it is of paramount importance to select the most suitable geometry.

![Figure 2-19](image.png)

**Figure 2-19** pressure contours showing flow distortion at the volute tongue at surge, choke and mid-flow [79]

### 2.3.1 Volute Interaction

As with the impeller/diffuser interaction, the volute performance is affected by the performance of the impeller and diffuser to which it is applied. Whitfield et al. [81] constructed two alternative volutes for comparison to a standard design; the first one transferred the diffusion process to a conical diffuser downstream of the volute and allowed no diffusion in the volute scroll (P1). The second one used the whole volute as a diffusing system (P2). Both diffusing scrolls had long tongues, designed to inhibit flow re-circulation. In contrast, the standard volute had a cut back tongue, which allowed greater flow re-circulation.

Whitfield found that the conical diffusing volute produced a small reduction in stage efficiency and pressure ratio and a reduction in operating range, but maintained the position of the surge line. It was suggested that the reduction in pressure ratio was because of the increase in losses caused by friction due to the high exit velocities with the reduced diffuser diameter ratio. The second volute produced a similar performance map to the standard
volute but produced a shift in the surge line to higher mass flow rates. Again, the operating range and efficiency were reduced.

For both diffusing volutes, there was a significant drop in static pressure from tongue to exit at choke, this is reflected in the variation in static pressure at three locations in the diffuser, shown in figure 2-20 for volute P1. The variation in static pressure at surge for three diffuser radius ratios is also shown in figure 2-20 for volute P1 and shows a significant rise in static pressure from tongue to exit. This plot clearly shows how the diffuser can be affected by the volute. Both these alternative designs produced comparable results to the standard volute, even with a reduction in the vaneless diffuser radius ratio although no significant improvement to performance was recorded.

2.3.2 The Work Of Braembussche
Van Den Braembussche has been prolific in his research into volute performance and has produced a considerable amount of work\cite{82-85} on the subject, which deserve particular mention. This work has focused on developing a 1-D prediction model for the volute based on the 3-D swirling flow. This has been achieved by simplifying both the volute shape and the flow to produce an extremely accurate technique.
Van Den Braembussche et al\textsuperscript{[82]} developed a 1-D flow model for off-design conditions to enable the accurate prediction of the circumferential pressure distribution and the estimation of blockage within the volute scroll and the experimental rig is shown in figure 2-21. Four tappings, shown in figure 2-22, equally positioned around the circumference at five cross-sections, recorded the static and stagnation pressure and velocity using a traversing probe. The results showed that the variation in stagnation pressure between each cross-section was mainly due to friction and diffusion and the stagnation pressure variation across the cross-section was due to swirling flow in the volute. The boundary layer observed near the walls of the volute were thin and this was attributed to the air entering the volute wrapping itself around the air already in the volute and also the curvature of the volute itself. The conclusions drawn from these experiments were:

- There was little variation in the swirl velocity at varying cross-section at different mass flow rates.
- There was a large variation in through-flow velocity at varying cross-sections at different mass flow rates.
- Flow and pressures measured near the walls are not representative of the average values. The below average velocity at the volute centre at low mass flow changes to a higher than average velocity at high mass flow which has the same effect as a positive or negative blockage value.

The theoretical flow calculations:

- Explain how the stagnation pressure loss influences the through-flow velocity and leads to blockage at the volute centre.
- Enable the stagnation pressure loss to be determined as a function of flow parameters and friction coefficient.

Van Den Braembussche\textsuperscript{[82]} investigated the flow in actual volutes with elliptical cross-sections to improve the previous theoretical model. The investigation measured the
stagnation and static pressure, tangential and radial velocity and flow in a compressor test rig. Isobaric plots of pressure, figure 2-23(b) and directional plots of velocity, figure 2-23(a) supported the findings of previous work and improved the theoretical model by improving the loss prediction.

(a) (b)

Figure 2-23 (a) swirl velocity vectors at varying volute cross-sections at high mass flow rate
(b) static pressure contours at varying volute cross-sections at high mass flow rate

From these result the authors validated the model based on Weber et al[86] to predict the losses due to the swirling flow. The model contained four loss mechanisms:

- Meridional dump loss – which assumed that swirl was dissipated in the volute.
- Tangential flow loss – corresponding to the loss of angular momentum.
- Friction loss – assumed to be that of a pipe friction flow.
- Exit cone losses – assumed to be due to sudden expansion at the exit.

The authors conclude that:

- Distribution of the swirl velocity was due either to internal friction or to radial velocity distribution at the volute inlet.
- The static pressure distribution depended on centrifugal forces due to the swirl in the cross-section.
- The core flow velocity resulted from the stagnation and static pressure distribution over the cross-section and can be very large at the centre.
Van Den Braembussche’s more recent work presents an improved 1-D prediction technique which takes into account the swirling velocity across the volute as well as the total pressure loss. This model will be assessed later in chapter six and is presented in detail in appendix I, section I-iii. These significant studies have been supported by the increasing use of CFD where the complex flows can be modelled with a greater degree of accuracy.

2.3.3 Conclusions
Whilst the development of the volute scroll has been limited over the last 30 years, particularly relating to volutes used in automotive application, it has been shown that a significant number of studies have been carried out to investigate the volute’s effect on upstream components and the swirling flow within it. As with the diffuser, the majority of works have involved large amounts of experimental analysis combined with numerical techniques and whilst Van Den Braembussche’s work has provided detailed data, little of this seems to have been applied directly to improving the way flow conditions in the volute are predicted. It is therefore considered that some of the works presented here could be used to support the development of a 1-D prediction technique to aid in the selection of suitable volute geometries at the initial design stage.

2.4 Flow Limitations
As described in section 1.2.2, several phenomena occur in the compressor as a result of particular flow conditions and these phenomena limit the performance of the compressor. This section highlights these flow limiting conditions and presents some of the research which has been carried out into these phenomena.

2.4.1 Surge
Many explanations of the surge process and contributing factors to its occurrence have been derived, mainly in bladed components, yet a definitive surge prediction is unavailable. Although surge conditions would vary in the vaneless diffuser, several of these points are equally as valid. Rogers considered surge to be caused by a combination of factors involving unstable characteristics and stall on the bladed components. He proposed that changes in geometry such as reduction in tip width and vaneless space widths would reduce
the possibility of surge but would not remove it. Elder et al\cite{71} examined the effects of surge in compressors with vaned diffusers and suggested several factors that affect surge for example:

a) **Impeller backswing and inlet swirl** - Both these parameters have the effect of changing the gradient of the work characteristic resulting in a lower flow at which the impeller stalls.

b) **Impeller and diffuser vanes and flow in the vaneless space** - Generally, it is the case that for a given impeller at high rotational speeds, surge occurs at a higher mass flow with a vaned diffuser than with a vaneless diffuser. At low speeds the vaned and vaneless diffusers tend to surge at similar mass flow rates, suggesting that inducer stall is more predominant. The ratio of impeller vanes to diffuser vanes also affects the flow range. This is related to the size of the wakes leaving the impeller to the width of the diffuser passage. The diffuser inlet area is the most susceptible to surge and flow separation in the vaneless space can result in stall in the vaned diffuser. The authors recommend that further study of $C_p$ at surge would be of great value to the design stage of a compressor.

c) **Surface roughness**\cite{48,91,92} - can affect the operating range and efficiency of the compressor by; improving the width of the efficiency curve but lowering the peak value, permitting some flow re-circulation resulting in work increasing more rapidly towards surge causing the pressure rise characteristic to peak at a lower mass flow rate and by improving the outlet flow conditions allowing downstream components to operate more efficiently. The surface roughness is dependent on the method of manufacture and will vary even between identical units. It is clear then that whilst surface roughness is an important factor in predicting the volute loss the major factor is still flow separation and turbulence.

### 2.4.2 Stall

Stall commonly appears when there is a high positive or negative incidence\cite{93,94}. Kline\cite{93} demonstrated that boundary layer theory alone is not enough to fully describe stall and that geometry must be of equal importance. He concluded that the following points must be considered in the design of a straight-walled diffuser:
- Stall should not be considered as a phenomena that occurs in a particular flow condition, but as a spectrum which develops from minor to severe for varying flow conditions.
- Stall in straight-walled diffusers with high LWR is distinct from stall on aerofoils as the stall characteristics are greatly influenced by the passage geometry.
- In order to predict the onset and behaviour of stall a complete knowledge of the inlet conditions is required.

Rotating stall is a cyclical fluid motion passing from one impeller blade passage to the next. Jansen\textsuperscript{95} suggested that rotating stall could occur in a vaneless diffuser and would precede surge at low flow rates when the fluid flow angle exceeded a critical value, causing pulsations in the flow. The author concluded that the flow oscillations were due to the radial component of velocity being directed inwards (towards the impeller). One way to increase the radial component is to move the diffuser walls closer together, but this increases the chance of boundary layer blockage. Studies of the boundary layer showed that the location of the separation depends on Mach and Reynolds number, flow angle and passage width.

Frigne et al\textsuperscript{80,85} analysed the rotating stall/boundary layer interaction and its effect on flow distortion in the vaneless diffuser. They hypothesised that due to adverse pressure gradients, boundary layer separation was likely to occur when the diffuser inlet flow angle reached a critical value. Experimental analyses of the rotating stall in different types of impeller and diffuser were carried out\textsuperscript{85}, measuring both pressure and velocity fluctuation. Various configurations were tested including the use of IGV’s to alter the flow profile. They found that stall could be divided into three categories which could be classified as:

- Diffuser rotating stall due to the interaction of the core flow and the boundary layer.
- Impeller stall due to impeller/diffuser interaction.
- Rotating stall due to flow separation in the impeller.
2.4.3 Choke

Determining the choke flow in a compressor is much easier than predicting surge. Lown et al.\textsuperscript{[96]} produced one of the earliest 1-D correlation methods for the prediction of choke flow in impellers using a 1-D expression for the choke flow based on upstream stagnation conditions. The authors state two possible requirements for the existence of choke flow:

- The physical area somewhere in the blade passage becomes a minimum or the increasing boundary layer forms an artificial throat.
- The stagnation pressure in the passage is such that critical conditions result in a minimum throat area.

The author suggested that the correct 1-D expression for the choke flow would also consider boundary layer growth presented in the form $A_{\text{effective}} = A_{\text{geometric}}(1-\delta)$ and density which would be modified by the losses to give $\rho_{\text{effective}} = k\rho_{\text{geometric}}$. These two parameters could be combine into one factor called the contraction ratio, $C_r = (1-\delta)k$. Four impellers of varying design were tested using this technique and a correlation of contraction ratio against incidence angle was presented. The contraction ratio varied with the incidence angle, with the least amount of contraction occurring at zero incidence.

Seidel et al.\textsuperscript{[94]} put forward a novel 1-D loss model for determining the aerodynamic parameters for choke conditions inside a vaned diffuser. The diffuser passage was divided up and the conservation equations were applied. By combining the mass and energy equations and using the equations of enthalpy and entropy, H-S diagrams were plotted for a range of inlet Mach numbers called a FANNO curve. The conservation equations of mass and momentum were combined in a similar way and produced an H-S diagram called a RAYLEIGH curve. Overlaying these two sets of curves produced a unique set of intersection points.
at various Mach numbers shown in figure 2-24 where the inlet flow conditions at choke could be determined. Whilst this method provided an almost infinite number of solutions for choke flow it is a more cumbersome method than the Lown et al[96] technique. However, both techniques are equally valid in their determination of choke.

2.4.4 Conclusions
The selection of works here represents only a handful of studies on flow limitations in centrifugal compressors. Certainly the prediction of surge has become the 'holy grail' of turbomachinery experts over the decades and it is clear from the work presented in this section that the effects of surge, choke and boundary layer growth are of extreme importance when determining the operating range and performance of a compressor. Many techniques have been developed over the years, typically based on empirical evidence and the majority using 1-D flow assumptions since the complexities of phenomena like stall and surge are still relatively difficult to understand.

2.5 Summary
This chapter has presented a selection of works relevant to this research topic. Whilst numerous research has been carried out into the performance and geometry of the centrifugal compressor only a small percentage has focused on those compressors with vaneless diffusers. Likewise the volute has undergone little in the way of development over the years, certainly as regards the automotive market, even though several authors have presented detailed models on the flow conditions within volute scroll.

This project will therefore bring together some of the work presented in these studies along with the mean-line flow assumptions presented in chapter one, to develop a 1-dimensional method of performance prediction for the automotive turbocharger compressor which will include not only an impeller model but will enable the user to optimise the design of the vaneless diffuser and volute casing both as individual components. The following section outlines how the project will be approached and what the hoped for outcomes will be.
2.6 Objectives and Justification for the Research

At the initial design of a centrifugal compressor, where performance is significantly affected by the geometry of the components - especially in terms of stability, range and losses - it is invaluable to have an accurate prediction of the compressor's behaviour. The prime objective of this research therefore is to,

Develop a 1-D prediction technique to enable compressor characteristics to be produced using the minimum of geometric data.

This objective can be achieved by:-

1. Understanding the behaviour of the flow inside a typical turbocharger compressor.

2. Identifying key geometric features which relate to performance.

3. Identifying areas of needed improvement and, using/adapting existing 1-D techniques, develop new prediction methods.

4. Extracting experimental data from typical compressors to formulate new correlations and simple models.

5. Validating the prediction technique against existing techniques.

Consideration has been given to the objective and the points stated above, and the following justifications have been made to support their validity:

It has been shown in chapter two that techniques for modelling specific forms of centrifugal compressor are well developed and have been used successfully on many occasions. Past academic and industrial research has also produced a significant amount of data for predicting compressor component behaviour. However, it has been shown that the vaneless diffuser and volute casing have undergone little change in design in recent years and have remained similar in appearance to earlier configurations. Limited amounts of performance prediction data exist for these components, certainly compared to the impeller, and in order to improve 1-D prediction techniques in general this issue needs to be addressed.
It is therefore, justified that an investigation into the relationship between the geometric features of the compressor components and performance is carried out. Appropriate combinations of data from past research will be examined further to identify these relationships.

Two major questions, need to be asked in terms of the feasibility of this research:

- Can a suitable method be produced to predict the performance of a turbocharger centrifugal compressor?
- If a positive answer can be given to the above question then how does this new method vary from previous methods, and does it improve on other techniques?

1-D prediction methods are inherently limited in describing the real flow behaviour in a compressor. Their main attribute however is that only limited amounts of geometric data are required to describe the compressor, thus providing the designer with a reasonable initial ‘guess’ at the overall performance. Methods which have been developed however, are generally for specific applications produced ‘in house’ by turbomachinery companies and cannot easily be applied to other forms of centrifugal compressor.

It is necessary therefore to develop a method of performance prediction which not only ensures that the range and matching of the components is satisfactory but which can be applied to any turbocharger compressor design. It is intended to utilise existing techniques in this process and identify those with appropriate features that can be adapted to suit this research application. It is believed that a PC-based technique is the most appropriate solution to producing an improved prediction technique and any models developed will be written in this medium.

It has been shown in chapter two that 1-D models of this type require considerable amounts of empirical data to enable accurate predictions to be produced. However, little experimental data usually exists at the initial design stage and a ‘catch 22’ situation then develops. In order to improve a 1-D prediction technique experimental data is vital.
It is essential that an experimental programme be undertaken, the aim of which will be to collect data from the components within the compressor. The experimental data will be used to determine the 1-D flow conditions in each of the three main components. It is intended to compare the fluid properties determined experimentally with those calculated by the 1-D theoretical assumptions and develop new models to describe the component performance. The purpose of these comparisons is to ensure that an optimum form of geometry is chosen which will improve existing designs of centrifugal compressor and benefit the development of new designs.

Although 1-D methods can provide an initial description of the performance, it will be necessary to validate whether the chosen geometric features best describe the nature of the compressor, or whether the model requires modification. Techniques such as 3-D Computational Fluid Dynamics (CFD) can be used for such validation. They are capable of modelling the complex 3-D flows which turbomachinery researchers have hitherto been unable to solve in a conventional numerical form. There are some difficulties presented in using this method namely that of adequately modelling the geometry and the complexity of the flow. Even so, the technique can provide a valuable insight into fluid behaviour and provide validation of the experiential results.

The 3-D analysis will be compared to the 1-D results to assess the accuracy of the 1-D model. Available published data, particularly for diffusing passages will be used to validate the CFD technique.
2.7 Structure of the Thesis

This thesis presents the work which has been carried out to support the research objective.

- Chapter three presents some of the significant numerical techniques in this field and introduces the specific 1-D prediction technique which forms the basis of this research work.

- Chapter four discusses some of the flaws in the 1-D technique. It is demonstrated that with limited geometry and overall characteristics a simple model for the prediction of ‘diffusion system’ performance can be produced, based on an ‘inverse’ assumption. Improvements are also made to some of the existing impeller models and significant improvements to the overall prediction are produced.

- Chapter five introduces the experimental programme and identifies various techniques which are applicable to this work. It discusses the compressors tested, the appropriate instrumentation techniques applied and data acquisition methodology. An overview of the limitations of the experimental procedure is presented along with a quantitative analysis of the data.

- Chapter six examines the overall characteristics from the compressors tested and looks at the pressure variations and flow conditions associated with the compressors. It discusses the occurrence of swirl phenomena downstream of the compressor, acknowledged but hitherto unrecorded and compares the predicted characteristic with the experimental data. Validation of the experimental data is also presented against a CFD model of one of the compressors.

- Chapter seven presents the component correlations developed from the experimental data and further improvements to the models for impeller surge and work are presented.

- Chapter eight presents existing techniques which can be applied to the prediction of performance in the diffuser and volute. The limitations of each techniques are presented. It is shown that the experimental data can be used to create separate
models for the diffuser and volute and that new empirical models, based on stagnation pressure loss and static pressure recovery equations, can be created.

- The correlations developed in chapter seven and eight are assessed in chapter nine. These models, written into the 1-D prediction technique, are validated for accuracy and the results presented.

It will be shown that the objective of developing a 1-D performance prediction technique to predict the performance of automotive centrifugal compressors has been achieved and that 1-D assumptions can be applied with a good degree of accuracy. It will also be shown that with the application of empirically-based models predictions can be produced for the diffuser and volute components that improve on existing prediction techniques.
3. Numerical Methods & The Performance Prediction Technique - CAPRICE

Synopsis

The accumulated knowledge and experience reviewed in chapters one and two has led to the development of many design and performance prediction techniques. These systems consist of a series of numerical models which enable the designer to select the most appropriate design for a given flow range. This chapter is divided into two sections; the first section looks in detail at existing numerical prediction techniques & discusses their key features. The second section outlines the 1-D performance prediction technique CAPRICE which will be used in this research project, looking at its operation, the data it produces, its limitations and the requirements necessary to improve it.
3.1 Introduction

The fundamental purpose behind assessing a compressor's performance is to develop a compressor that is both stable and operates efficiently within a chosen environment. It has been shown in chapters one and two that there are a wealth of experimental and theoretical techniques which have been used to good effect over the decades. The purpose of this research is to apply some of these established methods to develop an existing 1-D prediction technique, so improving its ability to predict compressor performance.

In this chapter an overview of several prominent 1-D prediction models is presented. This will put into context the direction this research is to take and what areas should be given attention as the during the work. Following this, a 1-D prediction technique, CAPRICE will be introduced. This is a PC-based prediction program which produces both numerical and graphical outputs of an overall compressor map based on component geometries and rotational speeds. This program will form the basis of this research work and will be used throughout the remainder of this thesis. The origins of CAPRICE's development are presented, as well as the processes by which it determines compressor performance. The limitations to the code and the areas requiring improvement are also outlined and conclusions drawn as to how this will be achieved.

3.2 Numerical Techniques

The majority of works presented in chapter two all have one major feature in common, they have all applied some form of numerical analysis to the problems associated with their research. Many of these methods however, tend to be developed and tailored to accommodate particular experimental tests and as such are only applicable to a specific task. Nevertheless, all these techniques generally follow the same methodology of gathering experimental data and comparing it with a numerical model to determine the model's predictive capabilities.

Modern numerical analyses generally take one of two forms; 1-D or mean-line modelling or 3-D invicid or boundary layer analysis using CFD. With the advent of tools such as CFD, 1-D prediction techniques could be considered outmoded, yet it is clear from the myriad
works that 1-D models play an important part in the initial assessment of turbomachinery problems and indeed consultancy companies such as PCA, Concepts NREC and Meerex are all presently developing and using their own versions of 1-D performance prediction software.

The following section looks at several notable numerical techniques used in performance prediction, focusing predominantly on 1-D analyses. A brief overview of CFD is however provided as it will be used later in this work as a verification tool only.

### 3.2.1 1-D Techniques

Herbert\textsuperscript{[11]} presented an analytical prediction method based on 1-D assumptions and typical compressor features, which could be applied to most compressors. This general method provided the designer with a means of comparing different compressors without specifying any complex blade geometry. He stated however, that the amount of available data with which to base an empirical prediction technique is scarce, particularly data on vaned and vaneless diffusers. This is undoubtedly the biggest obstacle to improving existing prediction techniques. Herbert maintained from the outset that:

\begin{quote}
'It (1-D technique) cannot be expected that any treatment (analysis) ... will achieve more than limited success'.
\end{quote}

This is a widely accepted view, but such 1-D techniques are still used and provide important data at the beginning of the design process. This can be seen in Herbert's results of predicted work, efficiency and
performance characteristic shown in figure 3-1 which correlated well with the experimental data. One important detail arising from this analysis was the unsatisfactory surge prediction. Analysis of the experimental data suggested that pressure recovery, \( C_{pr} \) in the vaneless diffuser was the most significant parameter, which tended to give a constant \( C_{pr} \) value at surge regardless of speed for a given compressor. He concluded that:

'With present knowledge, the onset of surge cannot reliably be included in this performance prediction method'

Clearly though, surge cannot be ignored and must at least be addressed in a prediction techniques to provide appropriate boundaries to the characteristic.

Although Hande's\(^{[44]}\) work looked specifically at compressors in the hydrocarbon industry, it is typical of most 1-D techniques. He developed an off-design prediction technique, which predicted the operating range for any given inlet condition. The flow model shown in figure 3-2, was established using conservation equations from which the velocity triangles and performance were determined. For off-design analysis, the efficiency was obtained as a function of incidence which was determined by the change in inlet flow coefficient, \( V_{in}/U \). Hande suggested that a suitable value of efficiency should be approximately 0.5 to 0.85 for a hydrocarbon compressor, which was wide enough to cover all eventualities. The determination of surge was assumed to be at the point when either the inducer or diffuser was stalled. Inducer stall was determined when incidence reached a critical value of around \(+9^\circ\) and diffuser stall was found when the inlet angle reached a maximum value of \(72^\circ\). Choke was defined as the point at which the impeller reached a minimum efficiency. For a particular set of hydrocarbon compressors the results of
experimental and numerical results compare well. Figure 3-3 shows an example of the discharge pressure against volumetric flow for one of the compressors. Around surge and choke, the numerical method was less accurate, suffering the same problem as Herbert's model. However, no modifications to the technique were suggested by Hande.

![Figure 3-3 Compressor performance characteristic](image)

Whitfield's extensive contribution to 1-D modelling has shown how simple analysis can be applied with great success[99-101]. He showed that by application of generalised gas-dynamic analysis – which could be applied to any form of turbomachine - a model can be produced to assess the performance of the impeller at inlet and exit in order to minimise the velocities in terms of absolute and relative Mach numbers. This he shows, leads to minimum overall losses in the compressor. He points out that whilst this particular technique: 'cannot give the correct answer' it can, with the appropriate application of empirical evidence and experience produce a satisfactory result. This 'incorrect' answer is clearly seen in Whitfield et al's work on turbocharger performance prediction[102] where the predicted compressor efficiencies were significantly lower than the experimental. Whilst they expended considerable effort in developing a detailed impeller model, it is believed that the limited diffusion system modelling resulted in this discrepancy in the overall characteristic of the compressor. Whitfield's later work although on turbine volutes[103] showed that with detailed experimental analysis and accurate correlation of the data and satisfactory prediction of flow inside a volute can be made.
3.2.2 Computer-Based Techniques

PC-based 1-D techniques have been used for some considerable time\textsuperscript{[104,105]}. They have one main advantage over traditional numerical approaches in that they reduce design time and allow repetitive iterations to be carried out rapidly, leaving the designer with greater flexibility to change the geometric features and optimise the compressor design.

One such technique presented by Campos et al \textsuperscript{[104]} enabled the compressor characteristic map to be plotted using a simple FORTRAN code which determined the operating points of the compressor at the intersections of efficiency and speed; a process normally involving numerous repetitive calculations, solved rapidly by using the processing power of the computer.

Al-Zubaidi\textsuperscript{[106-108]} demonstrated the use of a more complex FORTRAN-based program, which automated the aero- and thermodynamic prediction process to find the primary geometry and operating characteristics for impellers. The author divided the procedure up into four stages:

1. A design process for choosing the pressure ratio and mass flow rate.
2. Impeller blade geometry selection.
3. Analysis of the flow in the impeller.
4. Performance prediction to verify the design requirements.

He assumed zero pre-swirl at the impeller inlet and used conservation equations and 'assumed' values of blockage, slip factor and efficiency to determine the flow properties. The flow diagram in figure 3-4 shows the operation of the FORTRAN program. The method was relatively complex, using a series of iterative loops to determine the mass flow rate and relative velocity across a series of blade-to-blade and hub-to-shroud streamlines, from which an acceptable aerodynamic performance was achieved. Finally, these data were combined to form a quasi 3-D picture of the flow between the blades of the impeller. The resulting technique enabled the author to adjust the flow conditions quickly in order to produce the optimum impeller geometry.
3.2.3 3-D techniques - Computational Fluid Dynamics

Many of the more complex 3-D analysis dates back to the advent of the computer and its ability to process vast and complex amounts of data. Some of these techniques focused solely on the impeller due to its complex blades shapes and flow paths\textsuperscript{109,110}. Only in the last decade has the computing power to handle the complete compressor stage been available, allowing visualisation of the flow in the components.

3-D Computational Fluid Dynamics (CFD) is the analysis of systems involving fluid flow, heat transfer and other phenomena using computer simulation\textsuperscript{111}. It enables the fluid flow inside complex geometries such as casings and blade passages to be described in terms of fluid properties across small elements of fluid\textsuperscript{111}. The reason behind it's flexibility is due...
to the governing equations, Euler and Navier-Stokes (NS)\[^{111}\], being of relatively the same form for all turbomachinery, with the addition of equations for special cases.

As with all techniques such as this, CFD has its limitations. The main reason why CFD has lagged behind other conventional methods such as 1-D analysis is due to the complexity of the equation solvers, which are required to drive the software. Lakshminarayana\[^{112}\] stressed the need for improvements and validation to present-day codes and the development of codes to better handle tip clearance flow and cavitation. He also suggested improvements to codes to achieve faster convergence and the development of artificial intelligence to completely automate the solver procedure. Tsuei et al\[^{113}\] also suggests that refining the approach used by designers when applying CFD to a fluids problem would also help to improve its usability, for example the over-refining of grids leading to excessive numbers of nodes does not necessarily result in more accurate modeling.

The complexity of CFD code has limited software designers to purely laminar or turbulent flows, the modelling of transient behaviour remains a significant problem at this time. In the past, CFD has generally only been used as a verification tool by mass-averaging the flow data, as it will be in this research. However, many large companies are now beginning to use CFD as an integral part of research, to improve design and reduce production time.

The accuracy in predicting complex 3-D flows has led CFD to become an accepted method of flow prediction, yet it is clear that at present 3-D analysis is not ready to totally replace the simple 1-D numerical techniques. Until that time, the need for refinements in both modelling methods is still of great importance.

3.2.4 Conclusion

The numerical techniques presented here are only a tiny proportion of those that have been produced over the last 40 years however, the methods highlighted in section 3.2 are typical of the techniques used in compressor prediction models and, with the exception of Herbert\[^{111}\], tend to be specifically created for particular applications or compressor types.
The basic premise of the techniques however are essentially the same and it is this approach to predicting the performance that will be used in this research.

The weakest areas for many of these methods tends to be the lack of detailed modelling of the diffusion system and accurate surge prediction as highlighted in chapter two and this needs to be addressed. Many of these techniques, as has been shown, use considerable amounts of experimental data to develop the models and this cannot be avoided in such methods. However, large amounts of experimental data are not generally available at the initial design stage and this can make predicting the performance of a new compressor very difficult. It is proposed therefore that the relationship between experimental data and geometry be investigated to determine commonalities so that predictions can be made with the least amount of knowledge required.

To facilitate such analysis, an existing computer-based prediction technique will be used. The following section introduces this program, highlighting its similarity to many of the techniques in section 3.2. The 1-D flow assumptions and numerical process are presented along with an overview of the inherent limitations of the prediction technique which are shown in the form of compressor characteristic maps.
3.3 Introduction To The Integrated Design System

In recent years computer-based 1-D ‘Integrated design systems’, which incorporate both
turbine and compressor performance prediction have become more prolific\textsuperscript{[8,11]}, and a
typical integrated system is shown in figure 3-5\textsuperscript{[8]}.

![Diagram of integrated design systems](image)

These techniques have been developed to their present level through validation with experimental data, culminating in a highly effective analysis tools. The compressor design and prediction system shown in figure 3-5 comprises of:

- Preliminary Design
- Off-Design Performance Prediction
- Impeller 3-D Geometry Modelling
- Impeller Aerodynamics
- Other Aerodynamic Components
- Mechanical Analysis
- Impeller Manufacturing Data

The preliminary design\textsuperscript{[5]} program is based upon 1-D flow assumptions to achieve the basic geometry of the compressor and in the integrated system shown in figure 3-5 this is carried out by OONA. OONA uses operating point data, (mass flow rate, efficiency, rotational speed) to determine the appropriate 1-D geometry for the compressor (impeller outer diameter, hub & shroud diameters, vane angles etc.). The off-design prediction program
called CAPRICE uses the geometric data created by OONA to predict the off-design performance and operating limits in terms of surge and choke. The CAPRICE also calculates the overall efficiencies, pressure ratios and mass flow rates for selected rotational speeds. This program will form the basis from which an improved prediction technique will be developed and presented in this thesis. The following sections introduce the background to the CAPRICE program and the modelling procedure which was originally created with a vaned diffuser.

3.3.1 Background To The 1-D Off-Design Prediction Program - CAPRICE

The CAPRICE program was originally written for a compressor using a vaned diffuser stage. Because the vaned diffuser controls the flow in that type of compressor, a detailed volute model in the program was not required, as the volute range is much larger than the range achieved with a vaned diffuser. Later, the vaneless diffuser version of CAPRICE was developed by Swain[9]. It is typically used for automotive turbocharger compressors where the capability of the volute to diffuse between the tongue and the outlet flange is more important, especially in the limited envelope of the compressor map, typical of these type of compressors. This program has worked considerably well for many years, with the limited geometric features available at the initial design stage and includes parameters describing the diameter and width of the diffuser as well as the volute exit conditions.

CAPRICE was originally based on work presented by Rogers[90,115], whose technique at the time provided the designer with one of the most accurate 1-D models. It enabled the optimum selection of components, without having to compromise on performance. Rogers stated that at the initial design stage it was desirable to have a good ‘estimate’ of the compressor characteristic. The techniques at that time used a method of scaling the performance of existing geometrically similar compressors to achieve a characteristic. However, this method can be criticised as physical changes in impeller blade thickness, casing surface roughness and blade clearance do not size accordingly.

Rogers addressed the relationship between geometry and performance using widely accepted linear cascade theory. He presented a method of estimating the compressor characteristic given the geometry and peak efficiencies of the impeller and diffuser. He carried out tests on 27 geometrically different compressors with radially-vaned impellers
looking at the principles of operation and the non-dimensional similarities. He stated that the impeller characteristics was primarily governed by the inducer blade angle and throat geometry and concluded that:

- Peak impeller efficiency varied slightly with inducer blade angle and impeller diameter ratio.
- Inducer blade angles should not exceed 60°.

At inducer choke the efficiency will be low, away from choke the efficiency will rise to a peak value and then begin to fall because of inducer stalling losses. In a similar manner, the diffuser throat geometry and the calculated choking flow can be used to find the diffuser performance characteristic. Determining the incidence at stall in the diffuser is, as stated by Rogers, very difficult as the flow is extremely distorted after leaving the impeller, and the 1-D value is unlikely to represent the actual value, but will provide a reasonable assessment.

### 3.3.1.1 Centrifugal Compressor Losses

Rogers calculated the losses in the centrifugal compressor impeller and diffuser, based on blade cascade data from axial compressor studies. This was considered a reasonable assumption since the impeller and diffuser were both bladed components and considerable data was available on axial machines. The approach for axial compressors is traditionally based upon mean-line methods, mainly because the hub-to-tip ratio is usually large, but in contrast in a centrifugal compressor the blade length particularly at inlet is very large. It can therefore be seen that the RMS section better represents the rotor section rather than the mean diameter.

These cascade data were presented by Stratford et al.[116] using the relationships between loss and deflection against incidence as in

![Figure 3-6: Deflection/incidence correlation from Howell cascade theory (18)]
figure 3-6, developed by Howell and summarised by Horlock\textsuperscript{117}, and Rogers showed that compressor data could be correlated in a similar way. For any cascade, the best operating point is a compromise between the largest deflection and the minimum loss. The losses are as a result of changes in the incidence angle as the mass flow alters away from the design point. Rogers stated that loss as a function of incidence was difficult to determine in radial impellers and it was better to consider choke flow as a more stable parameter. Instead of a 'loss/incidence' relationship, (similar to that proposed by Hande) he proposed a 'loss/proportion of choke flow' relationship. By determining the impeller and diffuser flow ranges and efficiencies Rogers was able to match the characteristics and determine where stall would occur in the inducer and diffuser.

In accordance with Roger's model, CAPRICE assumes there are two main sets of losses in the compressor; impeller loss associated with the inducer throat and diffuser loss associated with the diffuser throat. Improvements to Roger's loss model were made by Swain\textsuperscript{9} to account for the increase in loss due to surface friction. This was achieved using an expression in terms of changes in relative velocity in the impeller passage in the form of an efficiency decrement $= 6 \times 10^{-7} W_1^2$.

3.3.1.2 Surge

The Rogers technique was adapted by Swain\textsuperscript{9} for use on turbocharger compressors with backswept impellers and vaned diffusers. This involved incorporating Wiesner's slip factor equation (1-12) to determine the work and velocity triangles. The surge correlation produced by Rogers, using data from radial impellers, was found to produce a poor surge prediction for the backswept impellers. Swain therefore chose an empirical equation which was a ratio of surge to choke capacity as a function of Mach number $M_{\infty}$, backsweep angle $\beta_2$ and diffuser vane number $N_D$ to calculate flow range.

$$\frac{\text{Surge capacity}}{\text{choke capacity}} = K_1 \cdot K_2$$

Where

$$K_1 = -0.333M_{\infty}^2 + 1.233M_{\infty} - 0.04$$

$$K_2 = (0.11\beta_2^2 - 0.14\beta_2 + 1) \ast (0.0007N_D + 0.975)$$
3.3.1.3 Blockage

In order to simplify his model, Rogers assumed a constant value of blockage factor. He surmised that at surge the boundary layer will be large, but at choke the blockage would be much smaller and as the flow through the throat would be almost orthogonal, the fluid properties and hence the choke flow can be determined. Swain’s model did not use a constant blockage, instead he used a varying blockage factor based on the Stratford et al\textsuperscript{[116]} correlation of boundary layer thickness using Mach number of the form:

\[ B = 0.046(1+0.8M^2)^{0.44}X\text{Re}^{-0.2} \tag{3-1} \]

Where

\[ X = (M/(1+(M^2/5)))^4 - 1 \int ((M/(1+(M^2/5)))^4) \, dx \]

3.3.1.4 Impeller Model

The inducer model, is shown in figure 3-7, where the proportion of loss is equivalent to the proportion of flow to choke flow and the efficiency ratio is \( \eta_{\text{imp}}/\eta_{\text{imp, peak}} \).

![Figure 3-7: Efficiency correlation curves](image)

The position of peak efficiency is defined by the empirical equation:

\[ \frac{m_{\text{peak}}}{m_{\max}} = \frac{U_2}{\sqrt{T_1}} + 5.64 \tag{3-2} \]

\[ \frac{m_{\text{peak}}}{m_{\max}} = \frac{U_2}{36.98} \]
This expression is modified at higher speeds to ensure that the efficiency ratio does not exceed 1. The efficiency ratio either side of the point at which peak flow rate occurs is found from the following empirical expressions:

Above peak efficiency flow rate.

\[
\eta_{\text{imp}} = 1 - \frac{0.4 \left( \frac{m^*}{m_{\text{max}}} - \frac{m_{\text{peak}}^*}{m_{\text{max}}} \right) \left( \frac{6.08 \frac{m_{\text{peak}}^*}{m_{\text{max}}} - 1}{\frac{m_{\text{peak}}^*}{m_{\text{max}}}} \right)}{\eta_{\text{imp}-\text{peak}}^{\text{eqn. (3-3)}}}
\]

Below peak efficiency flow rate.

\[
\eta_{\text{imp}} = 1 - \frac{\left( \frac{m_{\text{peak}}^*}{m_{\text{max}}} - \frac{m^*}{m_{\text{max}}} \right)^2}{5} = \frac{\eta_{\text{imp}-\text{peak}}^{\text{eqn. (3-4)}}}{\eta_{\text{imp}-\text{peak}}^{\text{eqn. (3-4)}}}
\]

### 3.3.1.5 Diffusion System Model

The diffusion system model\[9,90\] was derived from performance characteristics of a centrifugal compressor fitted with a vaned diffuser and the characteristic is shown in figure 3-8. The maximum losses that occur when the flow is fully choked i.e. at some negative angle of incidence, are where the efficiency will be at a minimum. At some point close to zero incidence the losses will be at their lowest and the efficiency will be a maximum.

![Diffusion system efficiency ratio](image)

**Figure 3-8** Diffusion system efficiency ratio [9]
3.3.1.6 Early CAPRICE Predictions

Initial tests using this modified model on a 6.5:1 pressure ratio compressor produced favourable results when predicting both efficiency and performance compared to experimental data, as shown in figure 3-9. However, some differences are clear, particularly the efficiency prediction and the range of mass flows in the characteristic map as well as the very straight surge line.

![Graph showing predicted efficiency and performance characteristic](image)

Figure 3-9  predicted efficiency and performance characteristic [9]

3.3.2 CAPRICE’s Iteration Procedure

This section describes the algebraic operation of the CAPRICE program, the data it requires and the calculations it makes to determine the overall compressor map over a range of prescribed speeds. A flow chart is presented, showing the iteration process and the limitations of the program are discussed.

Figure 3-10 shows the input data screen produced by the program from a data file created by the user. Data input includes geometric features such as inducer diameters, backsweep...
angle, and number of impeller vanes, throat areas and A/R ratio as well as properties such as inlet temperature, pressure and peak efficiencies. The compressor fluid properties are calculated at pre-determined stations; from impeller inlet \{1\} through the impeller exit \{2\} to the diffuser throat \{3\} and finally to the volute exit \{7\}. A series of mass flow rates and fluid properties, between the surge and choke points, at each speed are calculated, and are stored as a data array.

![CAPRICE geometric data input (screen shot)](image)

The program firstly determines the choke mass flow associated with the inducer which is calculated from the equation:

\[
m_c = \frac{(q_{ac}A_{TI}P_{01})}{\sqrt{T_{r01}}}
\]  

eqn. (3-5)
Where $q_{ac}$ is the critical non-dimensional flow parameter derived by Dixon$^{[118]}$.

$$
q_{ac} = \frac{\sqrt{\gamma}}{\sqrt{R}} \left( \frac{\gamma + 1}{\gamma} \right)^{\frac{\gamma - 1}{2}} \left( 1 + \left[ \frac{(\gamma - 1)}{2} \right] \right)^{2} 
$$

$T_{r01}$ is the relative stagnation temperature and $P_{r01}$ is the relative stagnation pressure at inducer RMS section, based on the conservation of rhothalpy with a constant blade speed between (1) and the impeller throat:

$$
T_{r01} = T_{i} + \frac{W_{RMS}^{2}}{2C_{p}} \quad \text{eqn. (3-7)}
$$

$$
P_{r01} = P_{i} \left( \frac{T_{r01}}{T_{i}} \right)^{\frac{\gamma - 1}{\gamma}} \quad \text{eqn. (3-8)}
$$

From this, an initial set of flow conditions is determined at the choke mass flow rate. A starting value of mass flow rate, $\dot{m}$ is determined as a proportion of the choke flow $\dot{m}_{c}$.

An initial guess of the peak overall efficiency $\eta_{t-p}$ peak is also made, which is defined as a fixed ratio of the peak impeller efficiency $\eta_{imp}$ peak and diffusion system efficiency, $\eta_{d}$ peak similar to that presented by Rogers$^{[115]}$. One advantage of the CAPRICE program is that it enables the user to select the values of peak diffuser and impeller efficiency, which adjusts the overall efficiency to suit. Giving the user the opportunity to select the peak efficiencies could be criticised, as prior knowledge of the component efficiencies would be required. However, the designer usually has a good understanding of the expected performance before hand. Once suitable peak efficiencies have been selected to ensure the level of performance is approximately correct these values are maintained constant and the process of matching the components can take place.
The diffusion system efficiency is defined as a ratio of diffuser efficiency \( \eta_d \) to peak diffuser efficiency \( \eta_{d \text{ peak}} \), (the peak efficiency being selected by the user as discussed above) and is found from an empirical equation:

\[
\frac{\eta_d}{\eta_{d \text{ peak}}} = -0.4924\alpha_2^2 + 0.9626\alpha_2 + 0.531
\]

eqn. (3-9)

This was based on previous correlations from a centrifugal compressor fitted with a vaneless diffuser[28]. The peak diffuser efficiency \( \eta_{d \text{ peak}} \) is a constant value and represents the peak efficiency of a state-of-art diffuser, which will be used throughout this work.

CAPRICE calculates the overall efficiency using equation (1-3), where \( T_{02} = T_{07} \) and:

\[
R_{ct4} = \frac{P_{07}}{P_{01}}
\]

eqn. (3-10)

\( P_{07} \) is found from:

\[
P_{07} = P_2 (\eta_7 + 1)^{\frac{r}{r-1}}
\]

eqn. (3-11)

Where \( \eta_7 \) is an overall performance parameter:

\[
\eta_7 = \eta_d \left( \frac{P_2}{P_{02}} \right)^{\frac{r}{r-1}} - 1
\]

eqn. (3-12)

And

\[
P_2 = P_{01} \left[ \eta_{imp} \frac{T_2 - T_{01}}{(T_{01} + 1)} \right]^{\frac{r}{r-1}}
\]

eqn. (3-13)

The impeller efficiency, \( \eta_{imp} \) is determined as a function of the peak impeller efficiency, \( \eta_{imp \text{ peak}} \) figure 3-7 using equation (3-2) to (3-4).
The CAPRICE program then determines a surge mass flow value using two iterative routines which are written as functions of both the inducer incidence angle and the impeller exit flow angle:

\[
m_{n+1}^* = \frac{m_i^*}{i_s^*} \quad \text{eqn. (3-14)}
\]

\[
m_{n+1}^* = \frac{m_{\alpha_2^*}}{\alpha_{2s}^*} \quad \text{eqn. (3-15)}
\]

The mass flow \(m_*,\) is adjusted until the inducer RMS incidence, \(i^*\) is equal to the inducer RMS incidence at surge, \(i_s^*\), using equation (3-14). The RMS incidence at surge can be found from:

\[
i_s^* = i_{\text{stall}} - \text{(user selected adjustment angle)}\]

Where the ‘user selected adjustment angle’ is an arbitrary value used only to get a better agreement with the experimental data. The stall incidence \(i_{\text{stall}}\) is determined from the nominal incidence, \(i^*\) and the nominal deflection, \(\varepsilon^*\) defined by Howell\[^{116}\] where \(i_{\text{stall}} = 0.52\varepsilon^* + i^*\). The nominal incidence, \(i^*\) is found from nominal gas angle at inducer outlet, \(\alpha_2^*\) (found from a modified version of Carter’s Rule using typical values) and the blade angle to the axial at the RMS section of the blade, \(\beta_{1\text{rms}}\), where \(i^* = \varepsilon^* - \beta_{1\text{rms}} + \alpha_2^*\).

The surge value of \(\alpha_{2s}\) is initially determined as a function of the vaneless diffuser width \(b_3\) and volute A/R ratio from:

\[
\alpha_{2s} = \tan^{-1}(2\pi b_3/(A/R)) + \text{(user selected adjustment angle)}\]

Again, the ‘user selected adjustment angle’ is an arbitrary value. Impeller exit mass flow \(m_{\alpha_{2s}}^*\) is adjusted until \(\alpha_2\) equals \(\alpha_{2s}\) using equation (3-15). The greater of \(m_{is}^*\) and \(m_{\alpha_{2s}}^*\) is used as the surge mass flow. Having calculated surge and choke conditions the program then determines a series of intermediate points to produce a performance curve. The flowchart in figure 3-11 shows how the program carries out this iterative process.
Start

Read geometric data

Calculate inducer and throat parameters

Read in speeds

Calculate initial gas properties

Calculate initial surge value of alpha2

Calculate \( m_s \) and stall incidence \( i_s \)

Calculate inducer choke flow

Calculate initial value of \( m \) as \( f(m_e) \)

Calculate inducer entry conditions

Adjust \( m \) as defined

Calculate inducer losses

Calculate vaneless diffuser conditions

Calculate impeller exit conditions

Compare \( \alpha_2 \) to \( \alpha_{2s} \)

Adjust \( m \) as defined

Compare \( m_i \) & \( m_c \) select the larger of the two values

Calculate range of flows between choke & surge

Compressor calculations:
Performance & flow characteristics

Compare \( m \) to \( m_c \)

\( m > m_c \) -> output data

\( m < m_c \)

Figure 3-11   CAPRICE flow chart of operation
The data produced by CAPRICE enables a compressor characteristic map to be plotted and an example is shown in figure 3-12. This is not a typical characteristic but has been modified to enable the data to be easily compared. The map is plotted with pressure ratio, \( R_{ct-t} \) against mass flow function, \( \frac{m \sqrt{T_{01}}}{P_{01}} \left( \frac{1}{10^{-3}} \right) \) and the speed lines are presented as a percentage of the highest speed.

The contour lines of efficiency are an efficiency ratio \( \frac{\eta_{t-t}}{\eta_{t-t, peak}} \). \( \eta_{t-t, peak} \) is a constant value and represents the peak efficiency for a state-of-art centrifugal compressor, which will be used throughout this work.

![Figure 3-12 CAPRICE predicted performance characteristic](image)

The overall performance data is also stored in a data array which includes pressures, temperatures and velocities, efficiencies and geometric data at each mass flow rate and speed.
3.3.3 Limitations of the CAPRICE Prediction Program

There are two fundamental problems with the CAPRICE program. Firstly, in order to produce a suitable characteristic map it is necessary to manipulate the values of inducer RMS incidence at surge $i_s$ and the impeller exit gas angle at surge $\alpha_{2s}$ directly in the code. Figure 3-13 shows two characteristics both predicted using the same compressor geometry. The blue speed lines were produced with an $i_s$ adjustment of $+2^\circ$ and an $\alpha_{2s}$ adjustment of $+2^\circ$ and the green speed lines were produced with an $i_s$ adjustment of $+5.3^\circ$ and an $\alpha_{2s}$ adjustment of $+10^\circ$, resulting in the map moving from right to left. From an analysis perspective this is useful as it enables the affect of $i_s$ and $\alpha_{2s}$ to be investigated, however, it requires the user to have a detailed knowledge of the code. Clearly, this would not be acceptable for commercial use; as firstly, the user would not have this knowledge and secondly, commercially viable software is required to operate with the minimum of intervention and code manipulation would render it useless.

Undoubtedly these issues needed to be addressed and an analysis of the impeller characteristic would be necessary to determine the effect of inducer incidence and exit gas angle on performance. It was also speculated that improvements to the surge model could...
be made which would remove the need for manual manipulation of the code during a prediction.

Secondly and more crucially, a problem lay in CAPRICE's inability to determine the individual performance of the vaneless diffuser and volute casing. As described in section 3.2.1 the diffuser and volute are combined into a single 'diffusion system' model which calculates the fluid properties between station \( \{3\} \) the diffuser throat and station \( \{7\} \) the volute exit. It was evident that some development of the CAPRICE program would be necessary to enable the component characteristics to be determined and suitable models describing the components behaviour produced. This would clearly require both experimental and theoretical data to produce the necessary correlations.

### 3.4 Conclusion

It has been shown that a suitable 1-D performance prediction technique exists but that it has several areas which require significant development. These improvements would not only enable the program to predict the overall compressor map more accurately but would provide the user with more accurate predictions of the diffuser and volute operating conditions.

To achieve this, a method of determining the diffusion system characteristics from the overall characteristic needed to be found. The following chapter focuses on this hypothesis and it will be shown that using the overall characteristic data from four existing compressors, along with their geometry, and by assuming the existing impeller characteristic is accurate, the diffusion system characteristic can be determined. It will also be shown that by analysing these data simple modifications to the CAPRICE program can be achieved.
4. CAPRICE Prediction & an Initial Examination of the Diffusion System

Synopsis

Data extracted from four turbocharger compressors, along with corresponding geometry has been used to determine the accuracy of the CAPRICE program outlined in chapter three. The program has been modified to enable the diffusion system characteristic to be extracted from overall compressor data and the existing impeller models within the program. It is shown that the existing technique does not adequately model the diffusion system. Correlations of the data have been produced and it will be shown that significant improvement to the surge and work models were made.
4.1 Introduction

Chapter three introduced the 1-D performance prediction technique - CAPRICE, which will be used throughout the remainder of this research. It has been shown that whilst the prediction technique produces adequate overall performance maps, several improvements to the existing code were required. With this in mind, a process of assessing the CAPRICE code needed to be devised.

Since CAPRICE produces an overall characteristic map based on inputted geometry it was hypothesised that if that process could be reversed i.e. the overall characteristics were inputted along with some or all of the compressor geometry, then data corresponding to the component characteristics could be extracted. This would allow the accuracy of the these characteristics to be assessed and improvements made to the models. However, to achieve this experimental data was required.

Garrett Engine Boosting Systems (GEBS) have an extensive selection of compressor variants however, due to the complexity and cost of instrumentation, detailed experimental data for each component is not generally available and compressor performance is usually only presented in terms of overall compressor maps. It was believed however that this, and some key geometric features of the compressor would be all that was required to prove the hypothesis.

This chapter is divided into two main sections, the first looks at the approach used to extract geometric data from available technical drawings and the overall characteristics maps of four existing compressors. The second section presents the approach used and the assumptions made to ‘inverse’ the CAPRICE program to accept these overall data. As a demonstration of its abilities and to emphasise the differences between the experimental data and theoretical prediction the CAPRICE predictions are presented against the four compressor maps provided by GEBS.

This chapter will show that:-

- The accuracy of the models in CAPRICE can be assessed by applying an ‘Inverse’ approach to the code, which allows experimental data to be ‘used’ instead of predictions ‘generated’ by the program.
The existing CAPRICE program poorly predicts the overall maps and diffusion system characteristic and supports the need for further improvement.

The outcome of the inverse approach enabled the existing impeller models to be improved and new formulae for the prediction of impeller work and impeller surge have been produced.

A modified CAPRICE program has been developed as a 'first stage' improvement and graphical comparisons are presented of overall performance and efficiency as well as impeller surge and work predictions.

4.2 The CAPRICE Prediction and Experimental Data

To produce a prediction of these existing compressors, CAPRICE required two items: a set of component geometric data, and a set of experimental data. It was therefore necessary to find a means of extracting useful data from the overall compressor maps to enable a comparison of the models used in CAPRICE.

The section is divided into two parts; the first looks at the compressor maps and the experimental data extracted from them. The second part uses programs from the ‘integrated system’ presented in section 3.3 to extract the four compressor’s geometric data which will be used in the CAPRICE program.

4.2.1 Experimental Data Extraction

GEBS provided data from a particular range of compressor builds, all with the same impeller tip diameter. This range of compressors was chosen as they were known to have sufficient available experimental and geometric data and have been reliably used for many years. A single impeller type was used in an attempt to isolate the performance of the diffusion system from the impeller. Four compressors were selected with varying impeller trims according to the availability of technical drawings and compressor maps.

The experimental data takes the form of compressor characteristic maps with 49 points on each map; 7 data points on 7 speed curves. The compressor maps are shown in figures 4-1.
to 4-4 with pressure ratio against mass flow function, \( \frac{m \sqrt{T_{01}}}{P_{01} / 1e^{-3}} \) and an overall efficiency

\[
\text{ratio} = \frac{\eta_{I-I}}{\eta_{I-I \text{ peak}}} \]

where \( \eta_{I-I \text{ peak}} \) represents the peak efficiency for a state-of-art centrifugal compressor. Three sets of data (mass flow, pressure ratio and overall efficiency) are extracted from these maps at each point for each rotational speed and stored as an array for use in the CAPRICE program.

![Compressor 1 experimental characteristic map](image)
Figure 4-2  compressor 2 experimental characteristic map

Figure 4-3  compressor 3 experimental characteristic map
The main characteristic of these maps is the prominent 'kink' in the surge line, notably between the second and third speed lines which is commonly accepted as the point at which surge in the impeller moves into the diffuser. The change in shape of the speed lines between low and high speed, from a broad and flat range to highly curved and narrow range is also a common feature. This is due to the behaviour of the components and the limitations associated with choke flow. The intention of the research is to match the prediction made by CAPRICE as closely as possible to these common features.
4.2.2 Geometric Data Extraction

The original design data for the four compressors was unavailable so it was necessary to extract the geometric data from technical drawings and blade definition data supplied by GEBS. Table 4-1 shows the non-dimensionalised geometry for the four compressors and the following section describes how these values were extracted.

<table>
<thead>
<tr>
<th>variable\comp</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\frac{D_t}{D_2}$ (squareness)</td>
<td>0.775</td>
<td>0.755</td>
<td>0.755</td>
<td>0.735</td>
</tr>
<tr>
<td>$\frac{D_{h1}}{D_2}$</td>
<td>0.234</td>
<td>0.234</td>
<td>0.234</td>
<td>0.234</td>
</tr>
<tr>
<td>$\beta_{rms}$ (deg)</td>
<td>57.113</td>
<td>56.379</td>
<td>56.379</td>
<td>55.622</td>
</tr>
<tr>
<td>$\beta_2$ (deg)</td>
<td>25.82</td>
<td>25.82</td>
<td>25.82</td>
<td>25.82</td>
</tr>
<tr>
<td>$\frac{b_4}{b_2}$</td>
<td>0.679</td>
<td>0.71</td>
<td>0.642</td>
<td>0.643</td>
</tr>
<tr>
<td>$\frac{b_2}{D_2}$</td>
<td>0.0912</td>
<td>0.0793</td>
<td>0.0793</td>
<td>0.0749</td>
</tr>
<tr>
<td>$\frac{D_3}{D_2}$</td>
<td>1.196</td>
<td>1.180</td>
<td>1.180</td>
<td>1.392</td>
</tr>
<tr>
<td>$\frac{D_4}{D_2}$</td>
<td>1.695</td>
<td>1.695</td>
<td>1.695</td>
<td>1.695</td>
</tr>
<tr>
<td>volute exit length/$A_6$</td>
<td>0.0363</td>
<td>0.0408</td>
<td>0.0219</td>
<td>0.0515</td>
</tr>
<tr>
<td>splitter vanes</td>
<td>6</td>
<td>6</td>
<td>6</td>
<td>6</td>
</tr>
<tr>
<td>main vanes</td>
<td>12</td>
<td>12</td>
<td>12</td>
<td>12</td>
</tr>
<tr>
<td>$A/R$ (mm)</td>
<td>17.78</td>
<td>17.78</td>
<td>15.24</td>
<td>12.7</td>
</tr>
<tr>
<td>$\frac{A_2}{A_1}$</td>
<td>0.632</td>
<td>0.588</td>
<td>0.588</td>
<td>0.592</td>
</tr>
<tr>
<td>$\frac{A_3}{A_2}$</td>
<td>0.764</td>
<td>0.837</td>
<td>0.758</td>
<td>0.895</td>
</tr>
<tr>
<td>$\frac{A_4}{A_2}$</td>
<td>1.0827</td>
<td>1.202</td>
<td>1.088</td>
<td>1.09</td>
</tr>
<tr>
<td>$\frac{A_6}{A_4}$</td>
<td>0.880</td>
<td>0.938</td>
<td>0.850</td>
<td>0.750</td>
</tr>
<tr>
<td>$\frac{A_7}{A_6}$</td>
<td>1.322</td>
<td>1.285</td>
<td>1.213</td>
<td>1.454</td>
</tr>
<tr>
<td>speed %</td>
<td>38</td>
<td>37</td>
<td>37</td>
<td>37</td>
</tr>
<tr>
<td>62</td>
<td>59</td>
<td>59</td>
<td>59</td>
<td></td>
</tr>
<tr>
<td>73</td>
<td>70</td>
<td>70</td>
<td>70</td>
<td></td>
</tr>
<tr>
<td>85</td>
<td>82</td>
<td>82</td>
<td>82</td>
<td></td>
</tr>
<tr>
<td>92</td>
<td>89</td>
<td>89</td>
<td>89</td>
<td></td>
</tr>
<tr>
<td>100</td>
<td>96</td>
<td>96</td>
<td>96</td>
<td></td>
</tr>
<tr>
<td>100</td>
<td>100</td>
<td>100</td>
<td>100</td>
<td></td>
</tr>
</tbody>
</table>

Table 4-1: Compressor Geometric Data

The geometric features extracted from the drawings and blade files were:

- Blade profile
- Throat area
- Rake and wrap angles
- Blade thickness
- Backswept angle
- Inducer inlet angle
- Blade RMS diameter
4.2.2.1 Blade Profile

In the integrated system described in section 3.3 and figure 3-5, VANESSA is the ‘3-D parametric vane definition’ program, which creates the impeller vane shape. EDITH is a program based on VANESSA and was specifically developed for GEBS by PCA. EDITH extracts data from a blade definition file (EX134) which describes the vane in a set manner using wrap angles, axial and radial distances and vane thickness’ (in the form of spheres) at set points. These data are processed using the analytical surface definition routines of VANESSA to produce a 3-D surface definition of the entire vane form. The blade inlet and outlet angles, throat widths, impeller tip widths and meridional profiles can then be found. A meridional projection of the impeller blade produced by EDITH is shown in figure 4-5.

Figure 4-5  meridional blade view created by VANESSA using EX134 blade data file

The grid superimposed on the profile consists of quasi-streamlines, which run equidistant to the blade hub and shroud profiles and sets of lines normal to the streamlines called quasi-orthonals. The data files for the chosen impellers were used to extract the throat areas and backsweep angles for each of the different trims. This was achieved by superimposing the shroud profile of the impeller vane on to the blade meridional projection. Shroud profiles come in the form of a technical drawing with associated data points along the profile shown in figure 4-6.
The EX134 data file only contains discrete points along the blade profile. It was therefore necessary to determine the tip width by interpolating from the closest discrete data points in the EX134 file.

4.2.2.2 Throat Area

Defining the blade shape enables the throat area between two consecutive blades to be calculated. The throat area can be found if the radial distance to the intersection of the shroud and blade profile is known. The meridional projection of the impeller blade, figure 4-7 shows the streamlines and the position of the throat in the form of a blue line. The axial distance to the intersection between the throat line and impeller shroud profile was determined from figure 4-7 using a series of axial and radial co-ordinate data (not shown in figure 4-6).
Cumulative throat cross-sectional openings, corresponding to the positions of intersection between six streamlines and the throat line, were extracted from the EX134 file. These values were plotted against radius shown in figure 4-8. A best-fit using a least squares method was found and the corresponding equation determined, from which the throat area at any location along the throat line (shown in blue in figure 4-7) could be determined.

\[ y = 1.2514x^2 + 18.402x - 347.56 \]

\[ R^2 = 0.9995 \]
4.2.2.3 Rake and Wrap Angles

The rake angle $\beta_{r2}$ (the angle the blade makes to the hub) can be calculated using the wrap angle $\beta_{w2}$ (the angles which describe the blade surface from some reference point) across the blade width at the tip, figures 4-9 and 4-10(a) and (b).

![Diagram of rake and wrap angles](www.meerex.com)

**Figure 4-9** visual representation of rake angle found from wrap angle (Meerex)

![Graphs showing axial and meridional positions of wrap angle](www.meerex.com)

**Figure 4-10** axial and meridional positions of wrap angle along the blades surface as shown in figure 4-9
4.2.2.4 Blade Thickness

The blade thickness at the impeller tip was calculated from the normal thickness plot taken from EDITH, figure 4-11. Knowing the blade dimensions at the tip it was possible to calculate the actual thickness of the blade.

![Figure 4-11](image-url) Normal thickness of the blade tip as determined from the back surface of the blade using EX134 data

4.2.2.5 Backsweep Angle

The 'geometric' backsweep angle, $\beta_{g2}$ of the impeller changes from hub to shroud at the impeller tip, largely because of the rake angle, $\beta_{r2}$. The combination of geometric and rake angles is called the 'effective' backsweep $\beta_{e2}$ and is required by CAPRICE. GEBS use the following equation based on past experience, which is commonly applied to their impellers, using $\beta_{r2} = 45^\circ$:

$$\beta_{e2} = \beta_{g2} + \beta_{r2} / 3$$  \hspace{1cm} \text{eqn. (4-1)}

For consistency this same equation was applied here.
4.2.2.6 Inducer Inlet Angle

The location of the inducer blade angle at hub, shroud and an intermediate streamline were found from figure 4-12 and are plotted as blade angle against radius in figure 4-13 from which the blade angle at any radius could be determined using the equation presented in figure 4-13.

![Figure 4-12](image)

Figure 4-12 meridional variation of blade angle along the surface of the blade
4.2.2.7 RMS Diameter

The RMS diameter, $D_{IRMS}$ was determined from the equation:

$$D_{IRMS} = \sqrt{\frac{D_{h1}^2 + D_{i1}^2}{2}}$$

eqn. (4-2)

where $D_{i1}$ and $D_{h1}$ were found from technical drawings. Since the relative velocity $W_1$, relative Mach number $M_1$ and airflow angle $\alpha_1$ vary along the inducer from hub to tip, it is the root mean squared (RMS) inlet diameter, $D_{IRMS}$, which is usually considered as this divides the inlet area into two equal areas. From figure 4-13, the RMS blade angle could be determined.

4.2.3 Summary

The data extracted from the compressor maps and from the technical drawings and ‘blade definition’ file will be used in CAPRICE to predict the overall characteristics of the four compressors. The following section presents the results of these predictions and shows that the prediction is very poor.
4.3 Predicted and Experimental Compressor Maps

The extracted geometric data were used in the CAPRICE prediction program and with the inducer incidence and impeller exit angles at surge equations set to +2° respectively, a prediction was produced for each compressor. These were overlaid onto the experimental maps are shown in figures 4-14 to 4-17. Broadly, the forms of the predicted and experimental characteristic maps show reasonable agreement. However, it can be seen that there are some major differences between the speed curves in terms of pressure ratio, mass flow and efficiency, notably the predicted speed curves are much flatter and the mass flow rate at surge are considerably higher than the actual values. This is believed to be as a result of:

- The user manipulated inducer incidence and impeller exit angles at surge being inaccurate.
- The mass flow range (between surge and choke) is not wide enough.
- The exit pressure is being under predicted, resulting in the flatter speed curves.

Figure 4-14 compressor 1: predicted and experimental performance characteristics
Figure 4-15  compressor 2: predicted and experimental performance characteristics

Figure 4-16  compressor 3: predicted and experimental performance characteristics
4.3.1 Predicted and Experimental Overall Efficiency Ratio

To further support these arguments the predicted overall efficiencies were determined using equations (3-10) to (3-13) and were compared against the overall efficiencies extracted from the experimental data, as described in section 3.3.2. The efficiencies were referenced to a state-of-art peak efficiency (as described in section 3.2.2) to produce an overall efficiency ratio, $(\eta_{tf}/\eta_{t_peak})$, which is presented in figure 4-18, for compressor 4 only.

As can be seen there is little agreement between the predicted and experimental efficiency ratios (and was the same for compressors 1 to 3), with the predicted $\eta_{t_f}/\eta_{t_peak}$ showing very flat efficiency curves, particularly at the higher speeds with peaks around 0.9. The experimental $\eta_{t_f}/\eta_{t_peak}$ drops quite rapidly at higher speeds and the experimental peak efficiencies are noticeably higher than the prediction. Since the overall efficiency is a function of the diffusion system and impeller efficiencies, an analysis of these efficiencies was necessary to determine their influence on the overall efficiency.
4.3.2 Summary

It has been shown in figures 4-14 to 4-18 that the predicted overall compressor maps and the overall efficiency ratio are poorly predicted by CAPRICE. Several parameters have been identified which require further investigation. These being:

- Surge
- Flow range
- Overall efficiency
- Diffusion system and impeller efficiency

By examining the behaviour of these parameters new models can be created which would go some way to improving the CAPRICE program. In order to achieve this, a method of extracting appropriate data had to be devised.

Using the geometric and overall data extracted from the four compressors it will be shown that it is possible to ‘inverse’ the CAPRICE program to output diffusion system data. The following section shows how this was achieved and what assumptions had to be made.
4.4 Examining the Diffusion System using an Inverse Approach

It was postulated that an 'inverse' approach to the data could be applied by using the extracted overall characteristic and the existing impeller model, which had been well tested over many years and was assumed to provide an accurate impeller characteristic. From this the diffusion system performance could be determined and compared with the CAPRICE prediction. The assumption focused on the CAPRICE program being modified to allow the experimental data to be 'read in' instead of allowing the program to calculate the conditions itself. It could be argued that using the overall data to determine the diffusion system performance is not realistic, since the interaction between the impeller and diffusion system have been shown to affect the overall performance. However, if it is assumed that the diffusion system has no influence on the behaviour of the impeller then it could be hypothesised that the diffusion system characteristic could be extracted from the overall and impeller characteristics.

A method was devised to extract the diffuser system characteristic from the experimental data and the following section shows how this was achieved. An opportunity to verify the accuracy of the model is also presented. An assessment of the impeller model was carried out and several improvements to the impeller work and surge predictions were made.

4.4.1 The Inverse CAPRICE Program

The data input subroutine of the program was modified to accept the experimental data array discussed in section 4.2.1 The blade speed calculation was modified to use the rotational speed values from the experimental data but the initial calculations of mass flow at choke and surge were unaltered from the original program. The experimental mass flow was then compared to the calculated choke mass flow to determine whether the experimental value was within the range predicted by CAPRICE for the given rotational speed. If the experimental mass flow was less than the calculated choke value then the experimental mass flow was accepted and used to determine the compressor characteristics using the original subroutines. If the experimental mass flow was greater than the calculated choke flow then the program moved on to the next experimental speed line. The flow chart in figure 4-19 shows the structure of the inverse program and how it was linked to the existing program.
The overall pressure ratios, extracted from the experimental data of the four compressors, were used to calculate the diffusion system efficiency which was defined as the 'total loss between the impeller outlet and volute outlet' using the following iterative scheme.

1. An initial guess was made of the diffusion system efficiency \( \eta_d \), based on impeller exit angle \( \alpha_2 \) using the equation (3-9).
2. The predicted overall pressure ratio $R_{c\,\text{pred}}$ was calculated, using the diffusion system efficiency.

3. The predicted overall pressure ratio was compared with the experimental overall pressure ratio, $R_{c\,\text{exp}}$.

4. The diffusion system efficiency, $\eta_d$, was adjusted by the error between the predicted and experimental pressure ratio. For simplicity the following adjustment routine adopted was:

$$\eta_d = \eta_d \left[ \frac{R_{c\,\text{exp}}}{R_{c\,\text{pred}}} \right]^3 \quad \text{eqn. (4-3)}$$

The use of a third power produced a more rapid divergence to the final solution.

5. Step 2 was repeated until the difference between the predicted and experimental pressure ratios were less than $1 \times 10^{-5}$.

The diffusion system pressure recovery $C_{pr}$ was also calculated between the impeller exit {2} and the volute exit {7} using the equation.

$$C_{pr} = \left( \frac{P_7 - P_2}{\frac{1}{2} \rho_2 V_2^2} \right) \quad \text{eqn. (4-4)}$$

### 4.4.2 Diffusion System Efficiency Ratio

The CAPRICE-predicted diffusion system efficiency ratio, $\eta_d / \eta_{d\,\text{peak}}$ and the diffusion system efficiency ratio, $\eta_d / \eta_{d\,\text{peak}}$ for compressor 4 are shown in figure 4-20. The peak diffuser efficiency $\eta_{d\,\text{peak}}$ is a constant value and represents the peak efficiency of a state-of-art diffuser, which will be used throughout this work.

As can be seen there is a significant difference between the two sets of data; the predicted curves being appreciably flatter, while the extracted diffusion curves show a considerable change in efficiency over each of the mass flow ranges, particularly at high speeds. This difference is due to CAPRICE’s limited diffusion system model and particularly its lack of a realistic volute model, which at the higher mass flow rates has a significant influence on the
flow. From this comparison, it is apparent that the existing diffusion system efficiency ratio model equation (3-9) does not adequately predict the diffusion system characteristic, especially at high speeds.

As the diffusion system is composed of two components, namely the vaneless diffuser and volute casing, it is impossible at this stage to ascertain which of these components dominates the characteristic until the two components can be separated. It can be postulated however that the angle of the volute tongue will impose a significant effect on the characteristic. This is because of the high mass flow trying to pass through the tongue area and the flow separation caused by the tongue itself. Both these factors will result in high diffusion system losses, hence the narrowing of the efficiency characteristic at high speeds. This will be discussed further in chapters five and six.

Using the experimental data available from the four compressors, it was surmised that an improvement to the existing diffusion system model could be made by correlating the efficiency ratio against impeller exit angle in the same way as equation (3-9). This relationship is shown in figure 4-21. The peak efficiency ratio for all four compressors
corresponds to an exit gas angle of approximately 64°. The existing CAPRICE diffuser system efficiency ratio, equation (3-9) is also shown (brown line).

An empirical equation for the diffusion system efficiency ratio was determined from a curve-fit of all the data in figure 4-21 (blue line) and is given:

$$\frac{\eta_d}{\eta_d_{\text{peak}}} = -0.0008191\alpha_2^2 + 0.1044463\alpha_2 - 2.4023869$$  \hspace{1cm} \text{eqn. (4-5)}$$

While there was considerable scatter of the experimental data at the lower impeller exit gas angles (below 65°, towards choke), there is a definite distribution to the data, which gives a much narrower curve-fit than the original empirical equation. Figure 4-22 shows an example of the difference between the high and low speed efficiencies more clearly. At low speeds, (37% of maximum speed) the diffusion system efficiency ratio is quite broad, operating over a range of gas angles between 50°-80°. At much higher speeds, (100% of maximum speed) the diffusion system efficiency ratio becomes extremely narrow, only operating between gas angles of 64°-72°. At the surge end (68°-71°) the two speed lines closely agree, however at the choke end (50°-65°) there is an extremely large separation between the speed-lines, which is also validated by figure 3-8.
In spite of this, it is believed that the above correlation provides a reliable first step to determining an improved diffusion system efficiency ratio particularly at surge, and this model will be tested in a modified CAPRICE program in section 4.7.
4.5 Improving the Impeller Model

The accuracy of the impeller model needed to be assessed to support the ‘inverse approach’ shown in section 4.4. A set of correlations were made to validate this assumption and are presented here.

4.5.1 Impeller Entropy Loss Coefficient

The experimentally-predicted impeller entropy loss coefficient $\zeta_s$ was determined using equation (1-14) and is presented in figure 4-23, against inducer incidence. It can be seen that there is a significant rise in $\zeta_s$ at incidence angles below $+3^\circ$, as flow moves towards choke. At low mass flow rates (positive incidence), the loss coefficient does not seem to be affected by changes in rotational speed and impeller work will therefore be high. Whereas at high mass flow rates (negative incidence) the rotational speed contributes to the loss and work will approach a minimum as predicted by Whitfield\textsuperscript{[25]}. Most of the data fell in the region above $+3^\circ$, where $\zeta_s$ was almost constant at 0.2. To have such a constant impeller loss was somewhat unrealistic and therefore improvements to the impeller model were justified to improve the prediction program. The accuracy of this relationship will be verified in chapter five using experimental data and equations by Denton\textsuperscript{[27]}.

![Figure 4-23: Experimental Impeller Loss Coefficient](image-url)
4.5.2 Impeller Work

The small variations between the theoretical and experimental characteristic maps figures 4-14 to 4-17 may represent differences in predicted and actual impeller work and therefore it is necessary to understand how work effects the overall characteristic of the compressor.

Firstly, consider the two hypothetical speed lines of the same value with their respective iso-efficiency lines shown in figure 4-24. Figure 4-24(a) shows that the work is approximately the same when $R_c$ and $\eta$ are the same, (ignoring any change attributable to the variation in mass flow). Figure 4-24(b) shows that the work will be different if $R_c$ is different but $\eta$ are approximately the same and vice-versa. As may be inferred from the above and from figures 4-14 to 4-17, determining this relationship visually is not at all straightforward and a better way of determining work have to be devised.

The incorrect selection of effective backsweep angle $\beta_{e2}$ may cause differences between the experimental and predicted work. Figure 4-25 shows the theoretical relationship between work and effective backsweep angle at a specified mass flow. Here, the experimental work point does not agree with the theoretical work line, which may be due to an incorrect effective backsweep value.

If the differences between the theoretical and experimental sets of data are consistent then an adjustment to the proportion of rake could be made so that the work difference was minimised. However, the difference in work may not be consistent, and this may indicate
that some adjustment to the slip relationship is needed. It must be remembered however, that even if the effective backsweep was varied to give the correct work, it does not ensure that $R_c$ and the iso-efficiency lines will match.

The following section looks at the impeller work of the four compressors. It will be shown that:

- The Wiesner equation, typically used to determine work, does not support the data found using the inverse program.
- A plot of impeller work with velocity ratio $V_{e2}/U_2$ yielded a correlation based on a power law form.
- A relationship between the power function and impeller backsweep angle was derived.
- An equation is presented that is shown to improve the prediction of impeller work for modern backswept impellers.

### 4.5.2.1 Work and Mass Flow Function Correlation

Based on those figures presented by Whitfield\[25\] in section 1.4.2.4, the impeller work was determined using the inverse program from the equation.

$$\Delta h = \frac{C_{pm} T_0 \left( \frac{\gamma - 1}{\gamma} \right) R_c^{\gamma/\gamma - 1}}{U_2^2 \eta_{etl}}$$

\[eqn. (4-6)\]

![Figure 4-25: Change in work with varying backsweep](image-url)
And was plotted against mass flow function, \( \frac{m \sqrt{T_{01}}}{P_{01} \cdot e^{-3}} \) in figure 4-26 for three speeds from compressor 1.

As a means of comparison, the impeller work was also determined from the Wiesner equation (1-12) using CAPRICE. It can be seen that the experimental data shows a definite parabolic shape to the experimental work data, in comparison to the linear increase of the prediction using Wiesner, which was also true for the other three compressors. This is supported by the findings of Dean\textsuperscript{[24]} and Whitfield\textsuperscript{[25]} in section 1.4.2.4, who showed that work increased in this form as mass flow function reduced. Since the differences between the predicted and experimental work were not consistent as discussed in section 4.6, a way of presenting the data was required that would produce a consistent correlation which could be applied to any compressor at any speed.

![Figure 4-26 predicted and experimental work for varying rotational speed](image)

**4.5.2.2 Work and Velocity Ratio Correlation**

Equation (1-12) shows work to be a function of both radial velocity and blade speed. If one considers for a particular compressor that the number of blades \( n \), and the backsweep angle \( \beta_2 \) are constant then work can be plotted against velocity ratio, \( V_{r2}/U_2 \), where the slope of the
line is the combination of \( n \) and \( \beta_2 \). As radial velocity can be determined for both sets of data, it was deemed appropriate to correlate work with \( V_{r2}/U_2 \). This could be criticised, as the determination of the velocity triangles depends on the correct determination of loss in the impeller. Since actual experimental data at the impeller exit was not available at this stage, the impeller loss coefficient could not be extracted using the data from the four compressors. However, this will be addressed in chapter six and is shown to be an acceptable assumption.

Figure 4-27 shows work against \( V_{r2}/U_2 \) for the four compressors found using CAPRICE and the inverse program. The data generated by the inverse program shows the impeller work to be slightly scattered which is a result of the difference between their blade trims. The data from the four compressors indicates that a ‘power’ curve-fit best describes the data, as shown in figure 4-27 (blue line), and this curve-fit produced an uncertainty of 0.78.

![Figure 4-27 experimental work correlation showing Wiesner prediction and experimental data curve fit](image)

The experimental and Wiesner-derived work (purple line) shows some similarity in the mid flow range of the data at around work values of 0.7–0.8. However, at the extremes of operation particularly at the low mass flows, below \( V_{r2}/U_2 \leq 0.25 \) and at high mass flow rates \( V_{r2}/U_2 \geq 0.4 \) the agreement is very poor. It is clear that the Wiesner work prediction
used in CAPRICE is unsuitable for these type of backswept impellers (the Wiesner model being originally for radial impellers).

The following section shows how these finding were used to determine an expression for the impeller work based on impeller backsweep angle and the velocity ratio in the form of a power equation.

### 4.5.2.3 Improved Work Correlation

Although this curve-fit shown in figure 4-27 reached a value of 1 before the velocity ratio reached zero, the curve-fit conforms moderately well with the experimental data. However, this single curve-fit did not describe all the data fully and therefore a curve-fit for each set of data was required. Curves of the form:

\[ y = cx^b \]  

were derived from a least-squares fit through the points, where \( c \) and \( b \) are constants.

It was found that the power values for the four compressors, \( b \) lay between -0.24 and -0.27, and multiplier values \( c \), lay between 0.54 and 0.56. Since the power constants \( b \), varied for each compressor, it was believed possible to determine a simple correlation reducing the number of variables, which would still describe the data accurately. It was found through a process of adjustment that the best form of the equation was that given in equation (4-8).

\[ y = 10x^b \]  

where only the variable \( b \) needed to be varied. This allowed the work curve to move vertically up or down by varying the power value \( b \) corresponding to the four compressor's data.

The Wiesner equation was used to determine a series of curves for a range of backsweep angles, \( \beta_2 \) (dashed lines) at 15°, 20° and 35° shown in figure 4-28. A range of power values between -0.2 and -0.4 were also used with equation (4-8) to find a match for the Wiesner curves. Figure 4-28 shows three power curves at \( b = -0.21 \), -0.26 and -0.4 (solid lines) which approximate to the mid sections of the Wiesner curves, previously discussed in
section 4.5.2.2. Whilst these simple curves do not pass through 1 as the experimental curve-fit equation (3-4-7) did, the curves conform sufficiently well to the experimental data to be accepted.

It was clear that the change in backsweep angle for the Wiesner equations corresponded with the change in the power values for the four compressors. It was believed that a correlation of these two parameters could be produced that would result in a relationship between work and backsweep angle similar to that found in the Wiesner equation.

A range of powers between -0.2 and -0.4 where plotted against the tangent of the blade angle (\(\tan \beta_2\)) over the range of angles between 15° and 35°. Figure 4-29 shows the resulting linear relationship between the two parameters. The corresponding equation, where the power variable is a function of \(\tan \beta_2\) is given as:

\[
b = -0.435(\tan \beta_2) - 0.097
\]
Figure 4-29  Backsweep angle and power function curve fits for varying angles

Incorporating this relationship into equation (4-8) produced an expression for determining the work for any given backsweep angle as a function of velocity ratio \( \frac{V_r}{U_2} \) referred to as the tan beta-power equation, and is given as:

\[
\frac{\Delta h}{U^2} = 10 \cdot \left( \frac{V_r}{U} \right)^{(-0.435 \tan \beta_i - 0.097)}
\]

eqn. (4-9)

Figure 4-28 shows the resulting ‘tan beta-power’ curves (hollow markers) at 15°, 20° and 30° using the (4-9). The curves correspond extremely well to the initial power curve using equation (4-8). This simple correlation provided the means to predict the impeller work with a single geometric feature, namely the impeller backsweep angle, with a good degree of accuracy and will be tested in the modified version of CAPRICE in section 4.7.

4.5.3 Impeller Surge

The prediction of surge in the centrifugal compressor, as discussed in section 2.4.1, is extremely difficult, particularly at the initial design stage. It had been shown that the incorrect selection of components, combined with an unstable pressure characteristic would invariably lead to the inception of surge. Due to the complexity of the flow under these
conditions, it is generally accepted that it is very difficult to accurately determine the fluid properties using any available 1-D prediction techniques. Realistically the designer can only use such data as a guide. However, a more accurate method of surge prediction was necessary to improve that already used in CAPRICE.

As is shown in section 3.3.2, CAPRICE calculates the surge in two ways, firstly by iterating the mass flow rate as a function of $i/i_0$, and then as a function of $\alpha_2/\alpha_{2s}$. Whichever of these iterations produces the bigger value of mass flow rate, that is the one used at surge. The problem with this is that the equations of incidence and exit angles at surge use purely arbitrary values and it was shown in figure 3-13 what effect this can have on the overall compressor map if a poor selection of ‘user adjusted angle’ is made. The designer would need to have a good understanding of typical numerical values for these parameters which would of course vary from compressor to compressor. Not only that but these values have to be input directly into the CAPRICE code which is not commercially acceptable.

With no existing precedent in the CAPRICE code as to what the values of incidence and exit angle should be it was concluded that an approach could be devised which would utilise the data from the four compressors, providing values of incidence and exit angle at surge. Since the air angles change at inlet and exit of the impeller with varying speed it was assumed that a correlation could be made against blade speed, $(U/\sqrt{T})$ for each of the parameters. Given that the impeller inlet and exit blade angles as well as the inducer and exit cross-sectional areas were similar for all four compressors it was hypothesised that the values of incidence and exit angle would be approximately the same. It was again assumed that the existing CAPRICE impeller model was accurate and, using the inverse program the inducer incidence and impeller exit angle at surge were extracted.

The following section looks at the data for each of the tested compressors and presents a correlation of the angles against blade speed. It will be shown that:

- The hypothesis of similarity of the angles between compressors was correct.
- The correlated data shows how stall moves through the impeller.
- The resulting correlations produced two new sets of equations for the prediction of surge.
4.5.3.1 **Inducer Incidence At Surge**

Figure 4-30 shows the variation of inducer incidence (at the root mean square section) at surge with blade speed $U_{1\text{rms}}/\sqrt{T_0}$ for the four compressors. All four compressors have similar surge incidence values between 10° and 27°, with a slight separation of the values at speeds below $U_{1\text{rms}}/\sqrt{T_0} < 12$. A possible explanation for this is inducer stall, which causes flow instabilities and can contribute to surge, but can equally be present in stable operating ranges.

![Figure 4-30 inducer incidence at surge correlation](image)

During compressor operation the stall phenomena can move from the impeller to the diffuser at higher speeds. This is supported by the kink in the overall compressor maps at surge and the findings of Frigne et al\textsuperscript{[80,85]}, and would account for the close match of incidence at high speeds. The experimental values of stall incidence $i_{\text{stall}}$ (derived in section 3.3.2) remained relatively constant across the speed range, with an average of 13° for the four compressors. These were slightly higher than the ‘user-selected’ stall incidence which was set at 10° for the initial CAPRICE prediction and would go some way to explaining the differences in the experimental and predicted maps.
Although Hande’s\textsuperscript{[51]} compressors where for a different application (discussed in section 3.2.1) and the value of stall incidence was slightly lower than those found here, they compared well and suggest that stall incidence does not vary by a large amount from compressor to compressor. This also suggests that the impellers were operating on the negative slope of the characteristic and was relatively stable.

Two straight-line fits of the data are shown in figure 4-30 (brown dashed line $6 < U_{1 \text{rms}}/\sqrt{T_{01}} \leq 10.4$ and blue dashed line $10.4 < U_{1 \text{rms}}/\sqrt{T_{01}} \leq 15$) which provided the simplest relationship between the blade speed and inducer incidence at surge. It could be argued that these curves do not accurately fit the data, especially in the mid-speed range where a step change occurs. But by splitting the data into two curves rather than using, for example a parabolic curve-fit, enables the curve shape to be better predicted and more importantly eliminates the need to manually manipulate the code. The inducer surge incidence equations are given as:

Below $U_{1 \text{rms}}/\sqrt{T_{01}} = 10.4$

$$i_s = -1.0185(U_{1 \text{rms}}/\sqrt{T_{01}})^2 + 31.434$$  \hspace{1cm} \text{eqn. (4-10)}

Above $U_{1 \text{rms}}/\sqrt{T_{01}} = 10.4$

$$i_s = -2.5506(U_{1 \text{rms}}/\sqrt{T_{01}})^2 + 47.366$$  \hspace{1cm} \text{eqn. (4-11)}

and the uncertainty in these best fit lines is given as $R^2 = 0.72$ and 0.9 respectively, providing a reasonable level of confidence in the equation’s ability to predict the inducer surge. These equations replace $i_s = i_{\text{stall}} - \text{(user adjusted angle)}$ in the CAPRICE program.
4.5.3.2 Impeller Exit Angle At Surge

Figure 4-31 shows the impeller exit flow angle at surge for the four compressors. All four compressors had consistently varying exit angles between 70° to 85°.

![Impeller exit angle at surge correlation](image)

The exit flow angle varies no more that 5° between the first three speeds ($U_2/\sqrt{T} = 9$ to 19), as compared with the 8° difference at the higher speeds ($U_2/\sqrt{T} = 20$ to 28). This suggests that at high speeds, stall occurs mainly in the inducer as suggested by figure 4-30. As the rotational speed increases the stall moves from the inducer to the impeller exit shown as the step-change in exit angle between speeds $U_2/\sqrt{T} = 16$ and 22. The final two speeds $U_2/\sqrt{T} = 26$ and 27, show a convergence of the four compressors data with an almost constant exit angle, suggesting the stall had moved from the impeller into the vaneless diffuser. These data are extremely useful for determining the speed at which impeller stall gives way to diffuser stall. The original equation for impeller exit angle at surge,

$$\alpha_{ls} = \tan^{-1}\left(\frac{2\pi b_2}{\text{AR}}\right) + \text{user adjusted angle}$$

was replaced with two straight-line fits of these data (brown dashed line $10 < U_2/\sqrt{T_0} \leq 18.7$ and blue dashed line $18.7 \leq U_2/\sqrt{T_0} \leq 27$),
and are defined as:

Below $U_2/\sqrt{T_0} = 18.7$

$$a_{2s} = -1.139 (U_2/\sqrt{T_0})^2 + 98.8005 \quad \text{eqn. (4-12)}$$

Above $U_2/\sqrt{T_0} = 18.7$

$$a_{2s} = -0.4552 (U_2/\sqrt{T_0})^2 + 85.948 \quad \text{eqn. (4-13)}$$

and the uncertainty in these best fit lines is given as $R^2 = 0.72$ and 0.86 respectively.

These two sets of equations have been added to the CAPRICE program to assess their accuracy and will be discussed in section 4.7.

### 4.5.4 Mach Number at Surge

Mach Numbers, which enable the fluid velocities from different compressors or components within the compressor to be compared, were derived from the overall data. A compressor is generally designed for a particular optimum operating point at each speed, and as such there is a corresponding Mach Number associated with that point. When the compressor operates above or below this point so the Mach number varies as the flow conditions change. By looking at the Mach numbers for the components it is possible to see how the compressors characteristic varies over its operating range.

The values of interest here are those associated with the surge point and inducer throat relative Mach number and diffuser pinch diameter absolute Mach number have been calculated from the equations:

$$M_{rel} = \frac{W_1}{\sqrt{\gamma R T_1}} \quad \text{and} \quad M_{abs} = \frac{V_3}{\sqrt{\gamma R T_3}} \quad \text{eqn. (4-14)}$$

These have been selected because they are two points in the compressor where cross-sectional area changes occur and at surge they are prone to flow separation. By determining the Mach number at these points gives some idea of where these components will surge.
The diffuser pinch Mach number is only an approximation at this stage since no actual data for the diffuser was available. However a reasonable estimate can be made based on the assumption that there is a 10\% difference in diameter between the impeller exit diameter and the vaneless diffuser pinch diameter based on established noise and vibration considerations. Thus the tangential velocity can be found from the impeller exit tangential velocity and the ratio of $D_3/D_2$. From this an estimate of the flow conditions can be made which include values for the absolute velocity $V_3$, from which the Mach number can be calculated.

Figure 4-32 shows the relationship of inducer throat relative Mach number and diffuser pinch diameter absolute Mach number against blade speed, along the surge line.

![Figure 4-32 Inducer and diffuser Mach number with varying speed](image)

In plotting the data in this way, an understanding can be gained as to when surge occurs in the impeller and diffuser and how it moves from one component to the other. At low speeds, diffuser Mach number is slightly higher than inducer Mach number and but a significant difference in Mach number exists at high speeds; the inducer Mach number reaching values of $M_{rel} \geq 1$. The occurrence of surge in the diffuser, at higher mass flows and speed, than those in the impeller was an effect identified by Elder et al\textsuperscript{[71]} (section 2.4.1)
and is validated here. It is also confirmed by the data presented in figure 4-31, which shows that beyond speeds of \( U_2/\sqrt{T} = 24 \) the stall moving from the impeller into the diffuser.

It can be seen that the two sets of data cross over between speeds \( U_2/\sqrt{T} = 13 \) to 17, showing that the movement of surge from the impeller into the diffuser is a gradual change across the mid-speeds of the compressor, hence the kink in the surge line on the compressor maps.

Below \( U_2/\sqrt{T} = 16 \), where the Mach numbers are low, indicates a region where increased separation and instability in the flow in both the diffuser and impeller will occur, with the inducer being the overall 'flow limiter' in this region. This would support the reason why the CAPRICE prediction is more accurate at the lower speeds since a surge model exists for this area whilst a diffuser surge prediction does not, resulting in the overly straight surge line on the compressor maps.

The curve-fit equations for the two sets of data are given as:

\[
\text{Inducer throat Mach N}^o, \quad M_{\text{int}} = 0.040939(U_2/\sqrt{T_{01}}) - 0.034204 \quad \text{eqn. (4-15)}
\]

\[
\text{Pinch diameter Mach N}^o, \quad M_{\text{abs}} = 0.020115(U_2/\sqrt{T_{01}}) + 0.267776 \quad \text{eqn. (4-16)}
\]

And will be used in later chapters to validate the assumptions presented here, particularly for the diffuser Mach number.
4.5.5 Summary

An assessment of the accuracy of the CAPRICE impeller model has been presented and several correlations have been produced using available overall experimental data and an ‘inverse’ approach derived from the CAPRICE code. Several key impeller characteristics have been examined, those being:

- Impeller Loss coefficient
- Impeller Work
- Inducer & Impeller exit angle at surge
- Inducer Mach number

It has been shown that whilst the original models produced reasonable predictions, significant improvements have been achieved from this new ‘inverse’ approach..

It has been shown that:

- A model of the impeller loss coefficient needs to be derived, which can only be achieved by using experimental data from an impeller.
- A relationship which improves the existing impeller work equation used in CAPRICE has been derived, based on the relationship of velocity ratio and impeller backsweep angle. It fully supports the findings of Dean\textsuperscript{24} and Whitfield\textsuperscript{25} and shows that Weisner’s prediction is not appropriate for modern-day backswept impellers.
- For the first time, a prediction of mass flow at surge based on inducer incidence and impeller exit angle has been created for CAPRICE that is fully free of user intervention.

These correlations have been included in a new version of CAPRICE and the resulting predictions are presented in the following section.
4.6 CAPRICE2 Predictions

The correlations discussed in sections 4.4 and 4.5 have been incorporated into the CAPRICE prediction program and this version was named CAPRICE2. Using only the geometric data from the four compressors, new performance predictions have been made. The following section looks at these new predictions and compares them to the experimental compressor maps. It is shown that with only modest adjustments to the code a significant improvement has been made to the performance prediction technique.

4.6.1 Overall Characteristics

Figures 4-33 to 4-36 show the actual and predicted characteristic maps for compressors 1 to 4. The predicted flow ranges at each speed stall remain flatter than the experimental speed lines, particularly above the 70% speed line. However, the predicted surge lines for the four compressors are much improved and compare well with the experimental surge lines for compressor 2 to 4 although differences exist between the predicted and experimental surge lines for compressor 1. Figures 4-34 and 4-36 show how the new surge prediction for the impeller matches the experimental data, up to the point where surge moves into the diffusion system. Additional modelling of surge in the diffuser and volute would improve the surge prediction at higher speeds and this will be investigated further in chapter six.
Figure 4-34  compressor 2: CAPRICE2 prediction and experimental performance characteristic

Figure 4-35  compressor 3: CAPRICE2 prediction and experimental performance characteristic
4.6.2 Overall Efficiency Ratio

Figure 4-37 shows the predicted and experimental overall efficiency ratios ($\eta_t / \eta_{t,peak}$) for compressor 4 (The efficiencies are referenced to a state-of-art peak efficiency as described in section 3.2.2). There is still a noticeable difference between the experimental and predicted curves, particularly in the mid to high speed range where the predicted efficiency curves are very flat. The CAPRICE2 predicted efficiencies are on average 0.01 of a percentage point lower than the original CAPRICE predicted efficiencies. However, the shape of the predicted efficiency ratios at low speeds (37% - 70% max. speed) show a wider range of mass flow rates compared to figure 4-18.

The poor prediction of overall efficiency at high speeds was anticipated based on the findings presented in section 4.4.2, in which the diffusion system characteristic was shown to be poorly match at those speeds. Since the overall efficiency in CAPRICE and CAPRICE2 were calculated as a function of the diffusion system efficiency, equations (3-10) to (3-13), the slightly lower overall efficiency ratio values shown in figure 4-37 can be associated with the new diffusion system characteristic equation (4-5).
4.6.3 Diffusion System Efficiency Ratio

The predicted and experimental diffusion system efficiency ratios $\eta_d/\eta_{d\text{ peak}}$ are shown in figure 4-38. The peak diffuser efficiency $\eta_{d\text{ peak}}$ is a constant value and represents the peak efficiency of a state-of-art diffuser, as discussed in section 4.4.2. There is a good match between the two sets of data at the lowest speed and this is due to the match between the curve-fit, (equation (4-5)) in figure 4-21 and the extracted data from the four compressors. However, the remaining predictions are still extremely poor, particularly at the highest speeds. The flattening off of the highest speed curve is caused by the large difference between the curve-fit correlation and the experimental data, shown in figure 4-21. As discussed in section 4.6.2., the shape of the predicted overall efficiency ratio curves is directly related to the diffusion system efficiency ratio and this is most noticeable at the highest speeds.
There is also a significant difference between the values of peak efficiency between the two sets of data. The predicted data again shows an almost constant peak efficiency ratio point of just above 0.9 compared to the experimental, who’s peak varies between low and high speed. This again is related to figure 4-21, where it can be seen that the experimental peak efficiency ratios vary by a similar number of percentage point from the curve-fit. This also explains why the previous CAPRICE diffusion system efficiency prediction was a better match in the mid-range but poor at low and high speeds. The CAPRICE2 diffusion system efficiency however, has improved the prediction at the surge end of the curves but good prediction at the choke end is still lacking.

This new prediction and the findings presented in section 4.4.2 both support the need for improvements to the diffusion system model. In order to produce a suitable correlation that would encompass the variation in the efficiency curves, it was postulated that the ‘shape’ of the curves at each speed line would need to be predicted. This is investigated further in chapter six.
4.6.4 Impeller Work

The CAPRICE2 work prediction (red points) and experimentally-determined work for compressor 4 from figure 4-27 (blue points) is shown in figure 4-39. The prediction, using equation (4-9), is a little lower than the experimentally-determined work at the two extremes due to the simplified power equation, but in general, the shape of the curve is a good match to the experimental work. The error band between the experimental and predicted data was a maximum of +/- 0.15 but only one point lay at this extreme, the majority were in a range +/-0.05. This was deemed to be an acceptable error and the new work prediction was incorporated into the impeller model.

![Figure 4-39: CAPRICE 2 work prediction showing experimental data scatter](image)

Page 135
4.6.5 Impeller Surge

The impeller exit angle at surge is shown in figure 4-40 along with the experimentally-determined data from 4-30. The predicted impeller exit angle error band was +/-0.7°, with only two data points (at the highest speeds) being in error greater than +/-2°. The inducer surge prediction shown in figure 4-41 was in error by +/-0.6° again, the two highest speeds where outside the band, having a maximum error of +/-2°. Both these correlations have improved the prediction at surge, however it would be necessary to test this correlation against impellers of a different design to determine if it could be applied to any type of backswept impeller.

Figure 4-40 Compressor 4: Impeller exit angle at surge CAPRICE 2 prediction showing experimental data points
4.6.6 Conclusion

The main objective of this chapter was to assess the accuracy of the performance prediction program CAPRICE and to determine the diffusion system performance using available experimental data from four compressors. This was achieved by extracting the diffusion system characteristic from the overall and impeller characteristics by means of an ‘inverse’ technique. The predicted diffusion system efficiency ratio compared well with the experimental at surge and mid-flow, however at high speeds and high mass flow rates the prediction was poor and this was due to the shape of the diffusion system curves at high speed. It is suggested that a method of predicting the shape of the efficiency at each speed would improve the prediction.

In order to assess the accuracy of the impeller model an analysis of the impeller work was produced. The work was determined from the overall experimental data and was shown to correlate with a power equation which was substantially different at the extremes of flow, (particularly low mass flow rates) from the generally used Wiesner correlation. The newly
determined work correlation was used in the prediction technique to improve the impeller model.

The inducer incidence and impeller exit angle at surge were determined from the experimental data in an attempt to improve the surge prediction of the CAPRICE program. The correlations produced, ensured that no manual input of the inducer and impeller exit angles at surge were required. Two pairs of straight-line curve-fits of the correlated data were used to predict the location of surge and produced a significant improvement to the existing prediction. The resulting surge prediction at the low speed end of the compressor maps shows excellent correlation with the experimental data. It is clear from these maps that further improved to the surge prediction could be made if the diffuser and volute flow at surge could also be determined.
4.7 Scope of the Initial Research

The prime objective of this research is to develop an existing 1-D prediction technique to enable compressor characteristics to be produced from the minimum of geometric data.

Three key stages have been achieved at this point:

1. Identification of the research project requirement.
2. Examination of the state of art within the field.
3. Initial development of an existing technique.

1. In the first instant, the following information was presented:

- An introduction to the relevance of 1-D performance prediction at the initial design stage which included an introduction to the principles of turbocharging and, in particular the operation and construction of the automotive turbocharger compressor and its constituent components.

- A review of fluid mechanic and thermodynamic theory required to understand the compressor performance characteristics and the relationship between compressor performance and geometry in terms of a 1-D numerical approach.

It was shown that:

- One-dimensional flow assumptions form the back-bone of modern-day turbocharger compressor analysis and are fundamental in determining the performance of a compressor.

2. An examination of existing work in this field:

- Demonstrated that considerable study of the compressor and its components has been carried out and much of this work can be used to validate and support the experimental and numerical analyses of this present research project.

- Examined some of the notable developments in both diffuser and volute research and put into context by focusing on the performance, design, fluid flow and geometry of the vaneless diffuser and volute casing.
• Showed that the theory has been applied and how this work will support the hypothesis of the research project.

• Showed that few comprehensive techniques exist which encompass the compressor as a complete unit and are generic enough to be applied to any centrifugal compressor design.

It was shown that:

• Even with substantial amounts of research in this field only a small percentage has focused on those compressors with vaneless diffusers.

• The volute has undergone little in the way of development over the years.

• The application of simple 1-D assumptions, combined with the wealth of theoretical and experimental knowledge can yield extremely accurate 1-D predictions for centrifugal compressor performance.

3. From this investigation:

• Available numerical performance prediction techniques where identified, showing their common features to be a series of empirical numerical models which enable the designer to select the most appropriate design for a given flow range.

• A 1-D performance prediction technique named CAPRICE was introduced, focusing on its operation, the data it produces and its limitations.

It was shown that:

• The numerical techniques presented tend to be specifically created for particular applications or compressor types.

• The basic premise of the techniques are essentially the same.

• The weakest areas for many of these methods tends to be the lack of detailed modelling of the diffusion system, shown to be the case in CAPRICE.

• Accurate surge prediction is difficult to attain, particularly so in the CAPRICE technique and needed to be addressed.
4.8 Further Development of the Research

It was identified that further improvements to the 1-D performance prediction technique were required and that:

- This would enable a more accurate characteristic to be predicted.
- It was necessary to isolate some of the relationships between geometry and performance from which improvements to the models could be made.
- The CAPRICE program could be compared against existing compressor maps to show the differences between experimental data and theoretical prediction.
- The accuracy of the models in CAPRICE could be assessed by application of an ‘Inverse’ approach.

It has been shown that:

- The existing CAPRICE program poorly predicts the overall maps, diffusion system characteristic and supports the need for further improvement.
- The results of the inverse program enabled the existing impeller model particularly the impeller work and impeller surge prediction to be improved.
- With some simple modifications the CAPRICE program could be vastly improved.

Based on these initial findings it was clear that:

1. Experimental data was required to enable models of the diffusion system to be created.

2. New models were required to improve the exiting CAPRICE program and enable more accurate compressor predictions to be made.

In order to achieve these two steps it was necessary to:

1. Devise an experimental test program from which suitable data could be collected.

   The aim of the test will be to collect interstage data from the components within the compressor. This will require extensive instrumentation of a compressor unit.
2. Develop a technique to extract and analyse the data.

This will be necessary to enable the data to be reduced and converted into a useable form, most likely in the form of a FORTRAN program similar to CAPRICE.

3. Determine the accuracy of the data.

An assessment of the data's accuracy will be needed to ensure that further analysis is correct.

4. Establish a series of suitable correlations using the experimental data.

The objective will be to identify common relationships between the data and show that these relationships can be used to produce 1-D models to describe the fluid properties and flow conditions.

5. Verify the data and correlations against recognized techniques and results.

It is intended to compare both the experimental data and the results of the selected correlations against well-established techniques and results in order to support the findings of this work.

6. Create a series of 1-D models using the correlated data for use in CAPRICE.

By application of the techniques and numerical assumptions already outlined in this work, a set of models will be created using FORTRAN code that will be applied to the CAPRICE program and used to predict the compressor performance.

7. Compare the prediction to the experimental data.

To establish the capability of the new models to predict performance a comparison of the prediction and the experimental data will be made.

The following chapters discuss each of these points in detail; the approaches used and assumptions made, as well as the findings, presented both numerically and graphically. It will be shown that significant improvement to the CAPRICE program has been achieved.
5. Experimental Test Programme

Synopsis

This chapter discusses the experimental programme undertaken to support the development of the 1-D prediction technique. Previous experimental work is presented, looking at the changes in approach over time. Ways in which these methods can be applied to the present research is also presented, from which an experimental approach has been developed. Also presented is the process by which the experimental test rig was developed and how the compressor components were instrumented. The operation and data acquisition methods are presented as well as an overview of the limitations of the test method. A quantitative analysis shows that the data for the two compressors is slightly scattered, but well within tolerable limits.
5.1. Introduction

It is common, particularly at the initial design stage of a new compressor, to have little or no experimental data available to support a designer's theoretical or numerical assumptions. Generally the designer is reliant on past experiences and existing correlations, all-be-it from different sized compressors, to produce a prediction for a new design.

As has been shown in earlier chapters, there is a two-fold reason for wanting to extract experimental data from a compressor and its components. Firstly, using the performance prediction technique available, in conjunction with these data, the accuracy of the numerical models can be determined. Secondly, the data extracted from the components – particularly the diffuser and volute - can be used to create new or improve existing models within the prediction program. It is therefore of great importance that experimental data be made available for this particular research work so that validation and improvement can be made to CAPRICE.

The purpose of the proposed experimental programme is therefore to collect data from a compressor which will be used to calculate the physical properties of the flow, such as velocities, densities, pressures and temperatures. A rig, consisting of a turbocharger compressor and a measurable air flow is required, and stagnation and static pressures and temperatures are the main sets of data needed to enable the flow conditions to be calculated. It is speculated that some existing experimental methods could be adapted to suit the requirements of the is work.

This chapter discusses the experimental work undertaken to support the development of the 1-D prediction technique. Firstly, previous experimental work is discussed, identifying different approaches to extracting data from centrifugal compressors, as well as the measurements commonly taken and some of the pit-falls associated with experimental work. Secondly, the scope of the work is outlined, including an overview of the available test rig and details of the compressors to be tested.

The instrumentation procedures are presented, explaining why they were chosen and how they were implemented. A detailed description of the measurement techniques is also given, discussing the equipment selected and the method by which the data was recorded.
Finally, the experimental test procedure is outlined along with a description of the limitations of the programme and an error analysis of the experimental data.

5.2. Experimental Techniques

Almost all of the previous work reviewed has included some form of experimental analysis whether it be diffuser performance or impeller blade passage design. These experimental studies have both supported and developed the common assumptions and understanding of fluid flow in the compressor and it is believed similar experimentation would go some way to improving the modifications already made to CAPRICE. The purpose of this section is to provide a general overview of some of the methods employed in experimental testing and to outline their advantages and disadvantages and possible use in the proposed experimental programme.

Early experimental analysis was of an invasive nature and many investigators, constructed large test rigs to study fluid flow in channel diffusers. Early methods took the form of dye streams or wool tufts, which were introduced into the fluid path to measure velocity and flow. Moore et al. and later Fox et al. used the dye method in their investigation into flow regimes. Waitman et al. used wool tufts to tabulate the type of flow according to the movement and orientation of the tuft. These methods gave an accurate indication of direction but no indication of the magnitude of the flow.

Digital methods of analysis and data extraction using hot wire probes to measure velocity were used by Fisher et al. These types of probes measure fluid speed from the convection of heat from an electrically heated wire. The probes were connected via a data acquisition system to a computer that logged very small changes in velocity. Different tapping locations in the volute casing were used in predetermined axial planes, which provided a complete 2-D map of the velocity across the passage. The hot wire probe has to be calibrated for magnitude and direction but an inherent problem is that of drift caused by changes in density in the fluid being measured. To minimise this, regular checks are required which can cause delays in the testing program. In some cases, an additional pressure measuring device is required.
Ayder et al\cite{83} study of flow in the volute used three-hole pneumatic probes to measure the diffuser outlet flow. 3-D flows and velocity distributions were constructed with the use of five-hole probes at seven cross-sections. Radial traverses were made across each section with one axial traverse along the centre line. Disk, cylinder or wedge static tube accuracy is dependent on the position of the sensing hole with respect to the nose of the tube and the tube itself. The nose tends to cause acceleration effects, which lower the pressure at the hole, and the tube tends to cause stagnation effects that act to raise the pressure at the hole. In a correctly designed tube however, these effects should be cancelled out at the hole. The stagnation pressure tube in its simplest form is a tube placed into the flow so that the sensing hole faces into the direction of the flow. The stagnation tube enables the ‘angle of attack’ of the flow to be determined by rotation of the tube in the flow.

One of the advantages of probe traverses across a pipe or in this case a volute, is that a 3-D picture can be constructed from the data. In a small automotive compressor, such as those used in this research project, the use of traverse probes is not appropriate. Mainly because the diameter of the passages could not accommodate a traverse probe without severely affecting the flow.

Whitfield et al\cite{12,13} used static pressure tapings positioned around the volute casing and at radial locations in the vaneless diffuser. These tapings were connected to a pressure transducer and a computer-controlled pressure scanner. The mass flow rate was controlled by a computer controlled throttle valve for accurate measurement. Using the measured static pressures, temperature and velocities determined from the mass flow rate and assuming zero swirl, the stagnation conditions could be determined.

Generally, wall tapings are assumed to be infinitely small, square-edged holes orthogonal to the flow. However, it is difficult to machine such holes (particularly in the context of

---

![Figure 5-1](image)

A typical experimental determination of hole size effect for 1.5<1/d<6
this work as automotive turbochargers are small and complex in shape) and to keep them free of small particles of dirt. Such small holes are also slow to respond to pressure changes. It is usual for a compromise to be made between finite hole size and static pressure error. Empirical data is available to determine the effects of hole-size on static pressure measurement and are shown in figures 5-1 and 5-2\cite{119}. The main advantage of static pressure wall tappings is that they are quick to implement and do not interfere with the fluid flow. Due to their small size, many can be positioned in a relatively small area.

![Figure 5-2 static pressure wall tapping](image)

The advent of laser anemometry vastly improved the measurement techniques in turbomachinery and many variations have been utilised. Two such methods include the 'Laser-Doppler' and 'Two-spot time of flight' systems. Laser-Doppler methods have been used on stationery components such as the diffuser and volute, whilst measurements of the rotating components have used the two-spot system. Although the instrumentation has to be placed through the compressor casing the laser does not interfere with the fluid flow. Their exceptional signal-to-noise ratios enables detailed readings to be taken of flow as close as 1mm from a surface.

Krain\cite{67} measured flow inside the impeller and diffuser using a two-spot laser. Flow vectors were measured at rotational speeds over 100,000rpm and produced 3-D graphs of velocity and pressure distribution. This method also allowed the flow field in adjacent blade channels to be measured thus permitting detailed flow visualisation in splitter vanes. The use of such methods is however, very expensive and time consuming and is more useful as a method of validating 3-D flow prediction programs.
5.2.1. Experimental Approach

Several considerations needed to be addressed before an experimental program could be carried out. A simple and repeatable method of testing was required and this eliminated dye tracer and hot-wire methods such as those implemented by Moore et al[58] and Fisher et al[68] since the equipment used was somewhat unreliable. It was necessary to consider approaches taking into consideration the physical size of the compressors to be tested; automotive compressor being relatively small. This ruled out pressure probe and LDA techniques similar to those used by Ayder et al[83] and Krain[67] which would not only be difficult to implement, particularly in the case of the laser technique, where small ‘windows’ would have to be cut into the casing, whilst the pressure probe method (which sits directly in the flow) would affect the air flow within the components. Whitfield et al's[12,13] approach seemed one which most suited to these type of compressors and best covered the issues of size, simplicity and repeatability.

Also of significant importance where the type of measurements required and the values to be calculated using those data. For a series of typical 1-D calculations, values such as static and stagnation pressures and temperatures, velocities and densities are fundamental, yet not all these values can be measured easily. Of those mentioned, static pressure and temperature are the most easily measurable since they require very little instrumentation. Many of the works reviewed in this research have used these type of measurements; Whitfield et al’s work again being a particularly good example. A series of static pressure tappings was seen as the most appropriate method to record data from the compressor and its components whilst additional measurements such as stagnation and static temperatures would have to be measured outside the compressor, preferably at inlet and exit. The following section discusses the additional issues of compressor type and test facilities made available to this research project, which ultimately defined the experimental approach applied.
5.3. Experimental Test Facility

All experimentation was carried out in a purpose-built test facility at Garrett Engine Boosting Systems (GEBS), which is shown in figure 5-3. The facility is designed to test complete turbochargers under steady state conditions over a range of speeds and mass flow rates. The rig records static and stagnation pressures and temperatures at the inlet and exit, rotational speed and mass flow rate data of both the compressor and turbine and was exactly what was required for this research. These data are typically reduced and used to plot maps of overall performance and efficiency.

5.3.1. Main Rig

Figures 5-4 and 5-5 show the schematic layouts of the test rig. A gas combustor powers the turbocharger turbine and the turbine vents to atmosphere via a large delivery ducting, which is not connected to the turbine. The compressor is driven by the turbine and draws air in via ducting from atmosphere. Air regulation is achieved via a throttle valve on the compressor discharge side and the mass flow is determined using a BS1042 orifice plate located in the inlet pipe.
The inlet ducting is sufficiently long to ensure an axial flow into the compressor and to enable mass flow measurements to be made. Inlet and delivery static pressure and temperature readings where taken at the locations shown in figure 5-4. On the inlet side, three platinum resistance thermometers, each in a 4mm sheath), are used to determine the inlet temperature. Each one is set at varying depths and an average temperature is calculated. The same types of thermometers are used on the delivery, each one in a 2.65mm sheath, again at varying depths. All the pressure measurements are static pressures. As may be seen in figure 5-3, the compressor delivery ducting is lagged between the compressor and the temperature instrumentation to reduce heat loss and the turbine inlet ducting is lagged to avoid radiated heat affecting the compressor delivery. The existing arrangement of instrumentation on the main rig suited the experimental requirements of this research as it provided all the necessary inlet and exit measurement apparatus.

![Figure 5-4 Turbocharger test rig inlet and exit locations](image-url)
5.3.2. Test Turbochargers

Several requirements of a turbocharger were highlighted as being necessary for a successful experimental programme to be implemented, these being:

- It must be typical automotive turbocharger of recent design.
- It must have a backshwept impeller and vaneless diffuser.
- It must be big enough to be instrumented.
- It must have some available experimental data, most importantly an existing overall compressor map.
- It must have available geometric data on all its components.

A turbocharger which fitted these criteria was the T04 compressor manufactured by GEBS. It had been in production for many years and was designed for commercial vehicle applications. A ‘backshwept’ and ‘vaneless’ design; both overall maps and geometric data being readily available. A second turbocharger of the GT25 design also fitted these criteria.
This was a slightly smaller turbocharger but of the same overall design as the T04 with a backswept impeller and vaneless diffuser and the particular model available was a new design for possible use in future passenger vehicle applications.

The GT25 Turbocharger was designated turbocharger 1 and the T04 turbocharger designated number 2. Both impellers had 12 vanes; 6 splitter and 6 main and the key geometric data for the two compressors is given in table 5-1.

<table>
<thead>
<tr>
<th>variable\comp</th>
<th>1</th>
<th>2</th>
</tr>
</thead>
<tbody>
<tr>
<td>$D_1/D_2$ (squareness)</td>
<td>0.707</td>
<td>0.755</td>
</tr>
<tr>
<td>$D_{m1}/D_2$</td>
<td>0.249</td>
<td>0.234</td>
</tr>
<tr>
<td>$\beta_{ms}$ (deg)</td>
<td>62.00</td>
<td>56.377</td>
</tr>
<tr>
<td>$\beta_2$ (deg)</td>
<td>35.00</td>
<td>28.82</td>
</tr>
<tr>
<td>$t/b_2$</td>
<td>0.145</td>
<td>0.215</td>
</tr>
<tr>
<td>$b_4/b_2$</td>
<td>0.850</td>
<td>0.643</td>
</tr>
<tr>
<td>$b_2/D_2$</td>
<td>0.0657</td>
<td>0.0509</td>
</tr>
<tr>
<td>$D_3/D_2$</td>
<td>1.075</td>
<td>1.180</td>
</tr>
<tr>
<td>$D_4/D_2$</td>
<td>1.767</td>
<td>1.695</td>
</tr>
<tr>
<td>Volute exit length/A6</td>
<td>0.0707</td>
<td>0.043</td>
</tr>
<tr>
<td>Splitter vanes</td>
<td>6 6</td>
<td></td>
</tr>
<tr>
<td>Main vanes</td>
<td>12 12</td>
<td></td>
</tr>
<tr>
<td>$A/R$ (mm)</td>
<td>15.25</td>
<td>15.24</td>
</tr>
<tr>
<td>$A_2/A_1$</td>
<td>0.599</td>
<td>0.615</td>
</tr>
<tr>
<td>$A_3/A_2$</td>
<td>0.914</td>
<td>0.758</td>
</tr>
<tr>
<td>$A_4/A_2$</td>
<td>1.502</td>
<td>1.089</td>
</tr>
<tr>
<td>$A_6/A_4$</td>
<td>0.837</td>
<td>0.850</td>
</tr>
<tr>
<td>$A_5/A_6$</td>
<td>1.345</td>
<td>1.213</td>
</tr>
<tr>
<td>Speed %</td>
<td>48 43</td>
<td></td>
</tr>
<tr>
<td>$74 81$</td>
<td>87 100</td>
<td></td>
</tr>
<tr>
<td>100</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 5-1 compressor geometric data
5.4. Compressor Interstage Rig

With the test facility and types of turbocharger fixed and the method of measurement chosen, the process of instrumenting the compressors needed to be addressed. The following section discusses how this was achieved and why these techniques were applied.

5.4.1. Station Selection

In order to position the turbochargers in the existing rig correctly and obtain the required data it was necessary to select suitable locations or 'stations' for the static pressure tappings that were common features of both turbocharger compressors. The station choice was based on several considerations:

- The size of the compressor.
- Access to the components and ease of assembly and disassembly.
- The number of stations required to record data.

Originally, only four stations were used in the CAPRICE prediction program; the impeller inlet and exit, diffuser exit and volute exit. However, in order to ensure that enough data was available to expand the prediction program, three extra locations were needed, these being at the vaneless diffuser pinch, the volute scroll and volute throat. It was decided that eight equally spaced tappings around the diffuser would provide the required amount of data. Likewise, eight tappings around the volute were considered to be adequate for the scroll. Table 5-2 shows the list of stations in the compressor where data needed to be recorded, along with their chosen designations.

<table>
<thead>
<tr>
<th></th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Impeller inlet</td>
<td>Impeller exit</td>
<td>Diffuser pinch</td>
<td>Diffuser exit</td>
<td>Volute scroll</td>
<td>Volute throat</td>
<td>Volute exit</td>
</tr>
</tbody>
</table>

Table 5-2
Figure 5-6 shows a cross-section of a typical compressor and the 'ideal' positions corresponding to the required stations.

On examination of the available compressors it was immediately clear that careful positioning of the tappings was needed to take account of each compressor's specific construction, particularly the strengthening ribs on the backplates. Figures 5-7 and 5-8 show the two compressors - both front and back - with the final choice of location for each set of tappings.
Figure 5-7  Turbocharger 1: static pressure tapping locations

Figure 5-8  Turbocharger 2: static pressure tapping locations
Tables 5-3 and 5-4 give the tapping positions, the numbers used in each location and the pressure tapping identification codes. With the locations identified the machining and fabrication of the tappings could be carried out.

Compressor 1:

<table>
<thead>
<tr>
<th>Position</th>
<th>Station</th>
<th>Number of tappings</th>
<th>Station Identification code</th>
<th>Tapping Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>1</td>
<td>1</td>
<td>CI</td>
<td>S1</td>
</tr>
<tr>
<td>Impeller Exit/Diffuser Pinch</td>
<td>2</td>
<td>5</td>
<td>PD</td>
<td>11,12,14,15,28</td>
</tr>
<tr>
<td>Impeller Profile</td>
<td>3</td>
<td>5</td>
<td>IP</td>
<td>16 – 19 &amp; 13</td>
</tr>
<tr>
<td>Diffuser Exit</td>
<td>4</td>
<td>8</td>
<td>DE</td>
<td>20 – 27</td>
</tr>
<tr>
<td>Diffuser Casing</td>
<td>5</td>
<td>8</td>
<td>VS</td>
<td>2 – 9</td>
</tr>
<tr>
<td>Volute Tongue</td>
<td>6</td>
<td>1</td>
<td>VT</td>
<td>10</td>
</tr>
<tr>
<td>Volute Exit</td>
<td>7</td>
<td>1</td>
<td>VE</td>
<td>1</td>
</tr>
</tbody>
</table>

Table 5-3

Compressor 2:

<table>
<thead>
<tr>
<th>Position Name</th>
<th>Station</th>
<th>Number of tappings</th>
<th>Station Identification code</th>
<th>Tapping Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Casing</td>
<td>1</td>
<td>2</td>
<td>CI</td>
<td>S1 &amp; S2</td>
</tr>
<tr>
<td>Impeller Exit</td>
<td>2</td>
<td>5</td>
<td>IE</td>
<td>11 – 15</td>
</tr>
<tr>
<td>Diffuser Pinch</td>
<td>3</td>
<td>5</td>
<td>PD</td>
<td>16 – 20</td>
</tr>
<tr>
<td>Diffuser Exit</td>
<td>4</td>
<td>8</td>
<td>DE</td>
<td>21 – 28</td>
</tr>
<tr>
<td>Volute Casing</td>
<td>5</td>
<td>8</td>
<td>VS</td>
<td>2 – 9</td>
</tr>
<tr>
<td>Volute Tongue</td>
<td>6</td>
<td>1</td>
<td>VT</td>
<td>TOG</td>
</tr>
<tr>
<td>Volute Exit</td>
<td>7</td>
<td>1</td>
<td>VE</td>
<td>1</td>
</tr>
</tbody>
</table>

Table 5-4

5.5. Fabrication & Assembly of the Pressure Tappings

This section looks at the construction and assembly of the pressure tappings and how associated problems during the fabrication process were overcome, paying particular attention to:

- The size of the holes with regard to flow interference.
- The proximity of the tappings to one another.
- The minimum amount of equipment required to record the data.
5.5.1. Location of the Pressure Tappings

The AGARD document\cite{120} AGARD-AR-245, suggests that the hole shown in figure 5-9 gives the best result for implementing static pressure tappings, particularly if the walls into which they are drilled are relatively thick. In the case of these turbochargers the wall thickness was sufficient to take this kind of tapping and all the tapping holes were machined in the same way.

Each hole was drilled through the casing with a 1mm diameter drill. The hole was then re-drilled to approximately half the depth of the original hole, to 1.5mm diameter. Hypodermic steel tubing (Gauge No 17, 1.47mm O.D.) was bent and cut to length before gluing into the hole with high temperature resistant adhesive as shown in figure 5-10.
5.5.1.1. Inlet Casing

The placement of the inlet casing tappings was critical; placement too close to the impeller would result in inaccurate readings because of boundary layer effects at the inducer, but too far up stream would place the tappings on the sloping surface of the casing. It was found that the ideal location was 25mm in from the front face of the inlet.

5.5.1.2. Impeller, Diffuser and Volute Tappings

The impeller tappings were placed circumferentially on the casing side of the compressor, four at 90° to each other and two at 45° as shown in figures 5-7 and 5-8. Two sets of diffuser tappings were installed as per figures 5-7 and 5-8. For compressor 2, on the diffuser pinch and exit, both sets were positioned through the backplate of the compressor. Due to the small size of compressor 1 the distance between the diffuser pinch radius and the impeller tip radius was found to be only 2.25mm. This made it impossible to locate both sets of tappings and the decision to include only one set of tappings had to be made. It was surmised that the diffuser pinch tappings could be used to determine the static pressure at the impeller tip based on a 'no loss' assumption between the two components. This method is discussed in greater detail in chapter 6.

The diffuser pinch tappings were installed in compressor 1 and the numerical assumptions made which justify this choice are described in detail in chapter 5 along with the calculations and an accuracy assessment of the approach. The volute tappings were straightforward to locate as the shape of the scroll made it easy to work on. Ten tappings were located evenly around the scroll between the tongue and the exit, all on the front face of the casing, as shown in figure 5-10.

5.5.2. Orientation & Access To The Components

The orientation of the turbocharger in the rig was dictated by the position of the oil supply for the central housing and the supply and delivery ducting. Before installing the pressure tapping tubing, the backplate was bolted up to the central housing and the volute casing delivery was angled to the required position of the ducting. Access to the impeller was of great importance both for installation and in case of damage and the decision to locate tappings as described in section 5.5.1 was proved to be the best approach. This allowed the
removal of the casing without the need to disconnect the hypodermic tubing from the bulkheads; the hypodermic tubing being flexible but strong enough to support the weight of the bulkhead when the bracket is unbolted from the backplate, shown in figure 5-11.

![Figure 5-11: Turbocharger 2 showing impeller and bulkhead bracket](image)

5.5.3. Bulkheads

The hypodermic tubing on both compressors was connected to two bulkheads supported on brackets. The bulkheads were positioned 80mm away from the compressor to avoid excessive heating and were constructed so that if necessary, access to the impeller could be gained by unbolting them. The brackets were also used to retain the backplate to the casing, replacing the original brackets. Figures 5-12 and 5-13 shown that there are several constructional differences between the two bulkheads and these are tabulated in table 5-5.

![Figure 5-12: Turbocharger 2 in test rig with tubing fitted](image)
The brackets were cut into shape to hold the backplate. The two brackets were positioned so that the hypodermic tubing from the volute side of the compressor were located on one bracket and the backplate hypodermic tubing were located on the other bracket. This allowed the volute casing to be removed from the backplate for access to the impeller. Holes were cut in the bracket to allow the hypodermic tubing to pass through. Slotted holes were cut in the brackets for the original bolts. This allowed the volute-side bracket to slide away from the backplate whilst remaining attached to the volute.

<table>
<thead>
<tr>
<th>Compressor 2</th>
<th>Compressor 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>Brackets: 80mmx10mm steel plate</td>
<td>Brackets: 80mmx10mm steel plate</td>
</tr>
<tr>
<td>Bulkheads: 100mmx100mmx22mm Aluminium bar</td>
<td>Bulkheads: 100mmx100mmx15mm steel bar</td>
</tr>
</tbody>
</table>

Holes in the bulkheads were drilled and threaded to take 1/8" BSP nuts, to half the thickness of the block.

3.3mm holes were drilled to half the thickness of the block and 30mm lengths of 3.3mm O.D. stainless steel tubing were glued in. The remaining thickness of the block was drilled with 1.5mm holes to take the hypodermic tubing. The tubing was glued in using a high temperature adhesive, ensuring that the glue did not block the hypodermic.

Table 5-5
5.6. Measurement Devices

To record the interstage pressure data from the compressors Scanivalve DSA3018 pressure transducer units where selected which have a facility for sending data via an Ethernet link to a remote PC. These devices were chosen as they are relatively compact and light-weight and could record from multiple inputs. Each unit has 16 inputs and three types were chosen, one with a dual range of 1.0/2.5 psid, and two others with 30 and 50 psid which would cover the range of pressures expected to be seen in the compressor components. An example of one of the Scanivalve units is shown in figure 5-14. This particular model also incorporates a temperature-compensating piezo-resistive pressure sensors with a pneumatic calibration valve and a 16-bit Analogue-to-Digital converter and microprocessor, which enables the unit to compensate for temperature changes and carry out online calibration checks; extremely useful if several consecutive experiments are carried out and the environment warms due to heat given off by the rig. This calibration capability provides a long-term accuracy of ±0.05% full scale reading.

Having carefully considered the location of the test rig and the position of the turbochargers the best location for the pressure transducers needed to be determined. On completion of the test program for this research work, it was intended to install the interstage rig as a permanent feature at the facility. Therefore convenience of access and discreet positioning were deemed the most important aspect of locating the transducers. After discussion with the GEBS engineering team it was recommended to them by the author of this work that suspending the devices from the ceiling as the best option. This enabled ease of access, reduced the risk of particles being ingested and ensured that condensation in the tubes did not cause spurious readings in the pressure data. A frame and fittings were installed to support the transducers and figure 5-15 shows the completed interstage rig with three transducer units connected via tubing to the compressor. The fourth smaller box containing the linking hardware between the Scanivalves and remote PC.
5.6.1. Bulkhead To Pressure Transducer Attachment

3.3 mm internal diameter nylon tubing was selected to connect the transducers to the bulkhead. Two different methods of attachment were tried, to test out the most best method for later use. For compressor 1 the nylon tubing was push-fit over the steel tubing and then glued at the base, shown in figure 5-16. This however meant that the tubing could only be used once. Compressor 2’s bulkhead was fitted with 1/8” BSP fitting and ferrule sets and nuts where attached to both ends of the nylon tubing as shown in figure 5-13. The use of BSP connectors meant that damaged tubing could be replaced and the tubing could be re-used in future experiments. This latter method was recommended to GEBS by the author of this work as the most appropriated installation method for future interstage experimentation.
5.7. Data Acquisition

The data measured by the transducers and the main rig sensors needed to be recorded in a form that could be easily accessed. The simplest format which could be accessed via a spreadsheet for example, was that of a data array which would store the speeds, mass flow rates, and all the temperatures and pressures.

A new PC-based data acquisition program called CIST was found to provide many of the features required for these particular tests and it was considered that a combination of this new program and the existing main rig data acquisition software would provide the best solution to the need. This was decided upon as it required both the minimum of installation and minimum disruption to the existing system i.e. re-routing or new installation of data cables, re-positioning of additional computer equipment etc.

The data acquisition program had to be functional in one of two ways, it needed to be timed to record the data at the same time as the main rig and it needed to produce a big enough array to record all the data.

The following sections describe the data acquisition software and how it operates and some of the modifications that were required to enable it to fit the purpose of these experiments.
The experimental procedure is outlined as well as the approach used to reduce the data into a usable form. Finally the overall compressor maps for compressors 1 and 2 are presented along with an error analysis of the mass flow rate and pressure data. The section concludes with a discussion of the limitations of the experimental procedure.

5.7.1. Data Acquisition Software

The data acquisition program was written using the LABVIEW software package, and contained two main elements; CIST (Computerised Inter-Stage Testing package) and COMPAL (data reduction package). CIST was the only package required for this research as it was decided that data reduction would be done after the experiments were complete. This decision was made so that the accuracy of the data could be checked at every stage of the reduction. Figure 5-17 shows the front screen of the CIST package.

Some modifications to the CIST code were required as it was not set up with the necessary location headings for this particular type of testing (see the ‘station choice’ column shown on figure 5-18). To rectify this an understanding of LABVIEW had to be gained. The author of this work attended a course to familiarise herself with this code and gain experience from which to modify the program. This modified version of CIST was then installed on the interstage rig.
5.7.2. Operation Of CIST

Firstly connection tests between the PC and the transducers were carried out to ensure correct connection via the Ethernet link. CIST automatically checks each connection by sending a sequential signal to the transducer Ethernet card to test each input. The ‘configuration’ screen is shown in figure 5-18, and enables the user to configure the pressure transducers. The screen consists of a table of colour-coded stations representing locations in the compressor. Alongside is a display representing the three pressure transducer units each with sixteen sockets. Un-used transducer sockets are represented as grey dots. To allocate a transducer socket to a station, a coloured code is selected from the table and then a grey dot is selected. The dot changes to the colour of the station chosen, the acquisition system then ‘knows’ the location of each tapping. When the location of the tappings has been selected this can be saved as a template for repeat testing.

![Figure 5-18 pressure tapping configuration screen](image)

A data ID screen (not shown) allows the test number, user initials and barometric pressure to be input. This information is then shown on the ‘pressure monitor’ screen, figure 5-19, which displays the list of stations the raw data from the transducers and the averaged data (time averaged over 5 seconds). The two buttons directly under the central table must show; ‘in range’ and ‘stable’ before a reading can be taken.
Figure 5-19 CIST screen showing tapping readings and ID numbers

Figure 5-20 shows the CIST code displayed on the interstage data PC which was located in the main control room of the test facility, the main rig computer being situated just behind it (top left).

Figure 5-20 The Interstage data PC showing the CIST data acquisition display & behind (top left) the main rig computer
5.8. Experimental Procedure

The experimental procedure was as follows:

- The combustor is ignited.
- The turbocharger turbine is run up to speed in a series of time steps using the main computer.
- When first speed is reached the mass flow is increased by opening the compressor delivery valve until choke flow is reached.
- The system captures readings once a thermally stable operating criterion of twenty-four seconds is met.
- The mass flow is reduced until surge is reached. Note: The inception of surge is determined by the thermal instability of the rig, the fluctuating mass flow and a qualitative assessment of the noise and vibration levels.
- Five intermediate readings are taken between surge and choke with a single sample of interstage data is taken at each point.

Four runs of each turbocharger were carried out to ensure consistent results and no modifications to the compressor were made between experiments. Five speed lines were recorded for compressor 1 and four for compressor 2. As the main rig recorded each point the interstage data was recorded. This required the compressor temperature on the main computer to be constantly monitored so that when the temperature became stable, a reading was taken.

5.9. Limitations Of The Experiment

Due to time limitations, an automatic trigger between the main rig and the interstage rig was not installed. A manual method of logging had to be used, which involved recording the interstage data just as the main rig registered it was about to store its data points. This was done by pressing the ‘sample’ button on the display (figure 5-19). Taking a reading when the main system is storing its data could result in an error in the interstage data, as the main automatic system would have already started to move to its next data point. This may be viewed as an unreliable method of data logging and the potential for errors is highly likely.
However by comparing the data between tests, an error would be clearly seen and therefore only the inconvenience of this logging method could be criticised.

As discussed, the difference in size and shape of the two compressors required different methods of fitting the instrumentation, however, these changes made no difference to the data acquisition process itself.

The primary use of the main rig is to test small automotive turbochargers. Due to the size of turbocharger 2 (manufactured for a truck engine) the existing combustor (the biggest available at that time) was unable to produce enough gas to drive the turbine up to its maximum speed. Also the standard volute scroll for this compressor was not available and a slightly larger volute was fitted. Thus, only four speed lines were achievable with this compressor.

5.10. Extraction of Data from the Experimental Results

Raw data from the main rig was automatically reduced and corrected by the data-logging PC into an array containing only compressor data. This included rotational speeds, the static pressures up and down stream of the compressor, inlet and exit temperatures and the differential pressure across the orifice plate. Additionally, the raw ‘interstage’ data was reduced into two forms; one containing each tapping value and the other containing the averaged values of the total number of tappings at each station for every point measured.

Several computer programs were written to process the data and figure 5-21 shows a flow chart of the program structure. Two programs converted the data into SI units, reduced and averaged it (datareduc & interdata). A second program (flocalc) read both files produced by datareduc & interdata and calculated the mass flow rates. Three other programs determined the averaged 1-D flow characteristics at each station (impcalcs, diffcalc & volutcalcs) and four others determined the 1-D flow characteristics at every tapping at each station (imptap, difhtap, difextap & voltap).
The stagnation pressures, temperatures, velocities, densities and gas angles were calculated by applying 1-D flow assumptions as presented in chapter one. Other values calculated included impeller loss coefficient, equation (1-14), impeller efficiency, equation (1-8) and impeller work, equation (1-15). The diffuser pinch and exit calculations included the stagnation pressure loss coefficients, equation (1-19) and static pressure recovery coefficients, equation (1-17) and the volute calculations included the volute loss between diffuser exit and scroll exit and flow conditions around the volute scroll.

The programs were designed to run sequentially so that impeller data could be used in the diffuser program and so on. Data identification was automatically created so that all files pertaining to a particular compressor were marked with the interstage data file name, ensuring that data was not wrongly labelled.
Figure 5-21  experimental data reduction and interstage data processing programs
5.11. Experimental Compressor Maps

Maps from compressors 1 and 2 are presented in figures 5-22 and 5-23. These maps are presented in terms of pressure ratio $R_c$ and mass flow function ($\dot{m}\sqrt{T_{01}}/(P_{01}/1e^3)$. The overall efficiencies are presented as an efficiency ratio ($\eta_{t-c}/\eta_{t-peak}$), where $\eta_{t-peak}$ represents the peak efficiency for a state-of-art centrifugal compressor, and will be used throughout this work.

![Figure 5-22](image)

compressor 2 experimental characteristic map

0.85, 0.87, 0.89, 0.89, 0.87, 100%

81%

62%

43%

Figure 5-22 compressor 2 experimental characteristic map
5.12. Error Analysis

It was considered that with the number of tests carried out on each compressor an average of the tests could be used which would represent a typical characteristic. To validate and determine the accuracy of the data a quantitative analysis was carried out.

5.12.1. Reading Errors

The static pressures in the interstage section of the compressor were recorded to four decimal places, giving a reading error of ±0.0001bar. The static pressures measured on the main rig were recorded to three decimal places and recorded a reading error of ±0.001bar. The temperatures gave a reading error of ±0.1°C and the speed a reading error of ±1rpm.

5.12.2. Static Pressure Error Analysis

As discussed in section 4.7, the interstage data was recorded manually via a P.C. and this resulted in varying reaction times for recording each data point, so introducing random errors into the interstage data. To determine the effect of these errors, a set of data from randomly selected tappings at three stations, the inlet casing, diffuser exit and volute scroll,
were chosen. A mean value of the static pressures at each mass flow rate, for all the tests was calculated. The standard deviation (SD) was calculated to determine the spread of the measured data and gives an indication of how far any one pressure measurement was from the average. The standard deviation was defined as:

$$SD = \sqrt{\frac{(d_1^2 + d_2^2 + \ldots + d_n^2)}{(n-1)}}$$

Where $d$ is the deviation from the mean and $n$ is the number of data points.

The SD for compressor 1 was approximately ±4000Pa (for three tests) and for compressor 2 was approximately ±400Pa (for four tests).

The standard error (SE) of the mean was also determined and this shows how far the mean value is from the measured data across each test and is defined as:

$$\text{Mean} = \frac{\text{sum of the data}}{n}$$

$$SD = \sqrt{\frac{(d_1^2 + d_2^2 + \ldots + d_n^2)}{(n-1)}}$$

$$\text{SE} = \frac{\text{Standard deviation}}{\sqrt{n}}$$

Where $n$ is the number of data points.

The standard error, at each of the tappings examined, for compressor 1 was approximately ±2300Pa or ±1.5% from the measured value and for compressor 2 the standard error was approximately ±160Pa or ±0.1% from the measured value. Further sets of tapping data were examined and it was found that the percentage error values stated above were representative for all the tappings. The standard error of the speeds across all the tests for compressor 1 was 0.14% of the mean values and for compressor 2 was 0.12% of the mean value.

5.12.3. Mass Flow Function Error Analysis

Figure 5-24 shows the deviation of the experimentally-determined mass flow function at each test point from the mean, across all the tests for both compressors. The experimentally-determined mass flow function is shown to deviate from the mean value by less than ±0.0008 (With 84% of the experimentally-determined values lying within a
±0.0004 band). This translates into an experimentally-determined mass flow rate deviation of ±0.005kg/s from the mean, for all these data.

![Figure 5-24 experimental mass flow deviation from the mean for compressors 1 and 2](image)

5.12.4. Conclusions

It is concluded that the small variation in mass flow rate would have little effect on the overall map and that it would be acceptable to use either one set of experimentally-determined mass flow rates or a mean mass flow value to represent a typical set of data for each compressor. Likewise, the pressure variations were deemed small enough to have little effect on the accuracy of any predictions made using these data. It is recognised that there will be a certain level of inaccuracy due to random errors generated in recording the data and the cumulative errors in the subsequent calculations, however, it must be remembered that 1-D performance prediction is an imprecise procedure and is intended to give an overall 'impression' of performance not an accurate result. Nevertheless, this should not give way to complacency when analysing the data, merely an acceptance of the inherent errors.
5.13 Summary

An experimental approach has been created, based on sound experimental practices. This has enabled the extraction of interstage data from two automotive turbocharger compressors. The method of construction of the test rig and instrumentation of the compressors has been presented and shown to be a reliable and repeatable method of testing. The development of this interstage rig has also provided Garrett Engine Boosting Systems with a new test facility and procedures for collecting component data.

Raw data in the form of pressures and temperatures, speeds and mass flow rates have been reduced into a format which can be used to create correlations for the improvement of the prediction technique CAPRICE. This has been achieved by the creation of a series of data processing programs developed especially for this research work.

The following chapter looks at the reduced forms of this data and determines its usefulness in developing suitable correlations. Correlations of the data are also presented and comparisons to CAPRICE predictions are made. Improvements to the program are discussed using empirical models based on the experimental data recorded here. It will be shown that further improvements have been made to the CAPRICE2 program using these data.
6 Overall Characteristics, Pressure Variations & CFD Validation

Synopsis

This chapter presents the experimental data - discussed in chapter five - in graphical form. An examination of the static pressure variations in the compressor between surge and choke is presented and illustrates the inherent interaction between the components. The 'No Loss' assumption discussed in chapter five is justified and shown to be acceptable within the limitations of the experimentation. Characteristic maps are presented and compared with CAPRICE2 predictions, along with the overall and impeller efficiency ratios. This chapter also introduces a CFD model of one of the tested compressors. The mass-averaged data from this model and the experimentally-determined flow conditions shows good agreement and validate the accuracy of the experimental data.
6.1 Introduction

The overall performance data and correlations made from the ‘inverse’ program in chapter four produced a substantial improvement to the CAPRICE prediction program however, it was clear that further improvements could be made and that this could be achieved using the experimental data from chapter five.

An assessment of the accuracy of the experimental data showed that, whilst some scatter exists, the variations are small enough to enable the data to be used to create correlations and models for the improvement of the CAPRICE program. It is however, prudent at this stage to analyse the data still further and determine the overall characteristics of the two compressors. The data extracted also provides a unique opportunity to assess the interaction and pressure variations, which occur in each component over the range of speeds and mass flow rates tested.

It was suggested in chapter five that the conditions at the impeller tip could be determined by using a ‘No Loss’ assumption between the impeller and diffuser pinch. The approach used to determine the tip data from this assumption is presented here. It will be shown that whilst the approach is sound, the error between the actual and ‘no loss’ static pressures is relatively high and that adjustments to compressor 1’s data would be necessary to account for this.

It will also be shown that as a result of the experimental programme, contra to the usual assumption of axial flow beyond the exit section of the volute casing, the flow continues to swirl well down stream of the compressor. It will be shown that gas angles in excess of 50° are associated with this phenomena.

This chapter also presents a comparison between the experimental and CAPRICE2 predictions for both compressors tested as well as differences between the overall and impeller efficiency ratios. Finally, the accuracy of the experimental data is also verified by comparing data from compressor 2 to the results from a CFD model of the same compressor. It will be shown that there is excellent agreement between the two sets of data.
6.2 Pressure Variations Inside the Compressor

The pressures inside the compressor are constantly changing as the mass flow rate and speeds vary. It is therefore important to be able to visualise these variations and to understand how they will effect the performance prediction. This section presents the static pressure variations around the compressor in graphical form and highlights some of the typical characteristic behaviour and interactions of the components between choke and surge. The determination of the impeller conditions using diffuser pinch data is presented and resulting effects on impeller efficiency are discussed. It will also be shown that axial flow assumptions down-stream of the volute exit are incorrect and that improvements to the prediction technique are required to compensate for swirling flow in this duct.

6.2.1 Determining the Impeller Characteristic from the Diffuser Pinch Characteristic

As discussed in section 4.4.1.2, the radial distance between the impeller tip (2) and diffuser pinch (3) of compressor 1 was extremely small and therefore tappings were only located at station (3). By assuming that mixing out had occurred by the time the flow reached the pinch diameter, it was assumed that the data recorded at station (3) could be used to determine the flow conditions at the impeller tip based on a ‘no loss’ assumption (\(P_{02} = P_{03}\)). This assumption was validated by using experimental data from compressor 2, where both impeller tip and diffuser pinch data had been recorded. From this, the ‘no loss’ model could be compared with the experimental conditions and the differences compared, giving an indication of how suitable the assumption was.

Since \(P_{02}\) was assumed to be equal to \(P_{03}\), \(P_{02}\) was greater than that determined experimentally \(P_{02}\). The experimentally-determined values of absolute and relative velocity \((V_2\) and \(V_{22}\)) were therefore higher, than the ‘no loss’ values, giving smaller values of static pressure \(P_2\), with very little change in density \(\rho_2\) and static temperature \(T_2\). The averaged difference between the ‘no loss’ and experimental static pressures was approximately 3.5% and for the stagnation pressures was 2.8%. These variations in pressure will in turn effect the efficiency prediction and consequently the overall compressor map. It was therefore important to determine by how much the impeller efficiency would be effected when applying the ‘no loss’ model.
It can be seen in figure 6-1 that there are significant differences between the experimentally-
determined impeller efficiency ratio \((\eta_{\text{imp}}/\eta_{\text{imp peak}})\) and the ‘no loss’ impeller efficiency 
ratio (‘no loss’ in red and experimental in blue). The diffuser pinch efficiency ratio \((\eta_d/\eta_d \text{ peak})\) is also shown (in green) to emphasise the differences between the data. The two 
impeller efficiencies were found to have an averaged difference of 7%, but it can be seen 
that the difference increased with larger mass flow rates. The largest difference between the 
efficiencies occurred at the highest mass flow rate and speed, and was found to be 28%.

Since the impeller and diffuser geometry of compressor 1 is different to that of compressor 
2 (table 4-1 section 4.3.3) it would be inappropriate to directly apply the error values for 
found here, onto compressor 1. It must therefore be accepted that by applying a ‘no loss’ 
assumption to compressor 1 the calculated static pressure values will be slightly lower than 
their true values and the relative and absolute velocities will be slightly higher than those 
determined from experimental data. This will result in the impeller efficiency being over 
predicted and will need to be addressed in the prediction program.
6.2.2 Compressor Static Pressure Variations

Figures 6-2 to 6-7 show the variation in static pressure at surge, choke and mid-range at stations {2} to {5} for compressors 1 and 2, (74% and 81% of maximum speed respectively). The static pressures are represented as a ratio of measured pressure to impeller exit dynamic head to allow the data from different sections of the compressors to be compared with each other and with data from figures 2-16, 2-18 and 2-20, section 2.2.3.[48][53][56][78].

Figures 6-5 to 6-7 show the static pressure variations for compressor 2. These figures show a significant variation in static pressure particularly between 45° and 90° (the volute tongue), suggesting that the flow is highly swirling between these tappings. It is possible that the effects of this swirling flow are transported up-stream into the diffuser, hence the repeated distortion in the flow at similar points up-stream. As stated in section 4.7, the volute scroll of compressor 2 was slightly larger than the standard casing, which could be the reason for the increased distortion, particularly at the volute tongue (as discussed in section 1.3.4.). Included in the pressure variation figures for compressor 2 are the ‘no loss’ model static pressure values (solid green lines). Here it can be seen that the pressure variations at the impeller tip are considerably ‘smoothed out’ by the ‘no loss’ assumption.

Compressor 1, shows reasonably uniform static pressure around the compressor suggesting that across the flow range the angle of the volute tongue and the angle of the flow are well matched and that any turbulence carried up-stream is minimal.[46][53]. Indeed, compressor 1 shows a relatively uniform pressure across the whole flow range, but it must be remembered that the impeller static pressure has been calculated not measured.

The mid-flow pressure distribution for both compressors is presented in figures 6-3 and 6-7. The mid-flows were chosen to correspond to the peak efficiency point, although it is worth remembering Cumpsty’s warning regarding the relationship between flow uniformity and peak efficiency, (section 2.3). In this case, it is simply coincidental that the mid-range flow corresponded to the peak efficiency and it was not necessarily the most uniform distribution.
compressor 1: Static pressure variations at each component at surge mass flow rate

compressor 1: Static pressure variations at each component at mid flow rate
**Figure 6-4**  
Compressor 1: Static pressure variations at each component at choke flow rate

**Figure 6-5**  
Compressor 2: Static pressure variations at each component at choke mass flow rate
Figure 6-6  compressor 2: Static pressure variations at each component at surge mass flow rate.

Figure 6-7  compressor 2: Static pressure variations at each component at mid flow rate.
Figure 6-8 shows a representation of the static pressure distribution around the compressor at the impeller exit and volute scroll at surge, choke and mid-range for compressor 2 (where 45° represents the volute tongue). This is similar to the iso-contour map of figure 2-18 presented by Sideris\textsuperscript{[79]} in section 2.3 and enables the pressure variations around the compressor to be better visualised\textsuperscript{[121]}. At surge, the volute is too large for the low mass flow passing through it. It slows the flow in the circumferential direction and increases the static pressure around the volute, the slower moving air impinges on the tongue, causing the pressure to drop severely. At choke, the increasing volumetric flow fills the volute causing the flow to accelerate towards the exit resulting in a fall in static pressure. The speed of the flow around the tongue area forces the pressure to rise. At mid-flow the static pressure is relatively constant around the volute; the flow is undistorted and the tongue angle and flow angle are well matched. Figure 6-5 showed a fall in pressure in the volute at 45°, with a similar fall in static pressure up-stream in the impeller but a further 45° around the circumference. This is clearly visible in figure 6-6 and would suggest that turbulence caused by swirling flow at the tongue is being carried up-stream into the impeller\textsuperscript{[121]} and that the angle shift could be associated with the log spiral path of the flow from the impeller exit, through the diffuser and around the volute. Further analysis of the log spiral phenomenon will be discussed in chapter six.

![Figure 6-8 static pressure distribution around compressor 2](image-url)
6.2.3 Swirl Between Volute Exit and Down Stream Ducting

It is generally assumed that at station \(8\) (downstream of the compressor exit, figure 5-5) the flow was sufficiently developed and could be described as an axial flow. However, it was found, in the case of compressor 2 that the stagnation pressure at station \(8\) was greater than at the volute exit \(7\). This could only be possible if there was some form of energy input into the flow, which was clearly not the case. It was therefore assumed that the flow at \(8\) had to be highly swirling and that the swirling nature of the flow had to be taken into account in the calculations.

Several issues arose when trying to determine if the flow was swirling in the exit ducting, Firstly, without the use of a yaw or cobra probe, a true measurement of the swirl angle, \(\alpha_8\) could not be made. This made it impossible to determine the actual velocity vectors at both \(7\) and \(8\). There was also no measurement of stagnation pressure on the test rig at either \(7\) or \(8\) as the introduction of total pressure probes would have resulted in disruption to the flow. Without a true value of stagnation pressure, it was not possible to determine the dynamic head at \(7\) and \(8\) accurately and therefore determine the loss in the pipe. Stagnation pressure had to be calculated and it was assumed that the calculated value of stagnation pressure would be considerably lower than the actual.

Also, there was no measurement of the static temperature on the exit ducting of the compressor and since there is no truly reliable way to measure it the static temperature had to be calculated. Only a single static pressure tapping was used at \(8\) with no auxiliary, and whilst the calibration of the static pressure tappings was deemed within acceptable limits it could not be validated. The same could be said for the static pressure tapping at \(7\), however due to the number of tappings around the volute, all with a similar range of values, the reading from this tapping was considered to be reasonably accurate. Calculations were therefore reliant on the measured static pressures at \(7\) & \(8\).

For a diffusing pipe section, the equations for continuity and constant angular momentum can be used to determine the absolute and swirl velocity components of the flow. An assessment of the loss between \(7\) and \(8\) was made with a range of swirl angles \(0^\circ\leq\alpha_7\leq80^\circ\) to find values for the stagnation pressures \(P_{07}\) and \(P_{08}\), using the experimental static pressures and isentropic flow assumptions. The results showed that the loss between
{7} and {8} remained negative between $0^\circ \leq \alpha_7 \leq 60^\circ$, above which it became positive. This was due to the change in stagnation pressure, $\Delta P$ being predominantly negative up to $60^\circ$, whilst the dynamic head remained positive. The gas angle $\alpha_8$ varied by an average of $1.45^\circ$ from $\alpha_7$, with the maximum and minimum variations being $+2.8^\circ$ and $-2.7^\circ$ respectively.

Based on a 'no loss' assumption ($P_{07} = P_{08}$) the exit velocity conditions were determined using an iterative method. Firstly an initial value of swirl angle, $\alpha_7$ is selected and, using the measured value of static pressure $P_7$ the conditions at station {7} are determined. Values of absolute velocity and density are calculated, from which the static temperature $T_7$ and finally the stagnation pressure, $P_{07}$ is determined. The value of density is adjusted, and the iterative process repeated until the calculated static pressure $P_8$ matches the measured static pressure. From this, a value for swirl angle $\alpha_8$ is determined. A range of $\alpha_7$ angles from $10^\circ$ to $80^\circ$ were applied. It was found that for the 'no loss' model, where the best match between the measured and experimental static pressures was found, the gas angle $\alpha_7 = 52^\circ$. The variation in gas angle between $\alpha_7$ and $\alpha_8$ was approximately $0.56^\circ$, with the maximum and minimum variations being $+0.72^\circ$ and $+0.36^\circ$ respectively.

A similar iterative scheme was applied based on a 'loss' assumption. The exit conditions were determined at station {8} by determining the head lost to friction between {7} to {8}. From these calculations it was found that where the measured and experimental static pressures $P_8$ closely matched, the gas angle $\alpha_7 = 58^\circ$. The variation in gas angle between $\alpha_7$ and $\alpha_8$ was approximately $2.6^\circ$, with the maximum and minimum variations being $+4.3^\circ$ and $+1.5^\circ$ respectively.

From the analysis of this data it was clear that an axial flow model in the down-stream ducting could not be assumed and a swirling flow model needed in the CAPRICE program to account for this. This model will be addressed in chapter eight.
6.2.4 Conclusion

The pressure variations inside the two tested compressors have been presented. It has been shown that:

- The conditions in the impeller exit can be determined from the diffuser pinch static pressure if a 'no loss' model is assumed. But it is recognised that this assumption can result in significant over prediction of the impeller efficiency and subsequently the overall compressor map, particularly at higher speeds.

- Figures 6-2 to 6-7 have been shown to agree well with data presented in section 2.3 (figures 2-16, 2-18 and 2-20). Figure 6-8 highlighting the pressure distortions, in the form of a contour plot (supported by data from figure 2-18, Sideris\textsuperscript{[79]}), in the volute at varying flow rates and showing that these distortions can be carried through the compressor components in the form of a log spiral flow path.

- It has been shown that the generally accepted assumption of axial flow in the downstream ducting of the compressor continues to swirl well beyond the volute exit. This phenomenon needs further analysis and possible inclusion in the CAPRICE program to improve the performance prediction technique.
6.3 Comparison of Experimental and CAPRICE2 Data

The following section compares the CAPRICE2 predictions for the two tested compressors against their experimental overall compressor maps and overall efficiency values as well as those for the impeller efficiency. It will be shown that significant differences still exists and that the need for improved modelling is vital.

6.3.1 Overall Performance Maps

Figures 6-9 and 6-10 show the characteristic maps for compressors 1 and 2, predicted using CAPRICE2 (in red) and the geometric data in table 4-1. The overall efficiency ratio ($\eta_{t_{\text{peak}}}/\eta_{t_{\text{peak}}}$), where $\eta_{t_{\text{peak}}}$ represents the peak efficiency for a state-of-art centrifugal compressor, are shown as hashed lines. These maps have been over-laid with the experimental characteristics in green.

The prediction for compressor 2 in figure 6-10, shows good agreement with the experimental; with wide flow ranges and a satisfactory surge prediction. However, the prediction for compressor 1 was particularly poor. This it was assumed was due to two reasons. Firstly compressor 1 was a newly designed compressor with significantly different geometry to compressor 2, and secondly, since the modifications made to the CAPRICE2 program, in chapter 3, were based on compressors with similar geometric features as compressor 2, the prediction could not accurately determine the compressor map using compressor 1’s geometry. This further strengthen the case for improving the models in CAPRICE2 so that ‘any’ compressor performance could be predicted as was originally specified in the brief. With this in mind, analysis of the flow conditions was necessary to determine suitable correlations from which new models could be created.
Overall Characteristic Data

Figure 6-9 compressor 1: experimental and CAPRICE2 predicted performance characteristic

Figure 6-10 compressor 2: experimental and CAPRICE2 predicted performance characteristic
6.3.2 Overall Efficiency Ratio

The CAPRICE2-predicted and experimental overall efficiency ratio $\eta_{t}/\eta_{t_{\text{peak}}}$ (where $\eta_{t_{\text{peak}}}$ represents the peak efficiency for a state-of-art centrifugal compressor) are shown in figures 6-11 and 6-12. There is poor agreement between the predicted and experimental values across the ranges of both compressors and the reasons for this were discussed in section 4.6.2. However, the predicted and experimental efficiency ratios for compressor 1 are noticeably worse than compressor 2.

![Figure 6-11: Compressor 1: Experimental and CAPRICE2 Predicted Overall Efficiency Ratio](image)

---

*Overall Characteristic Data*  
Chapter 6  
Page 190
6.3.3 Impeller Efficiency Ratio

The experimental and predicted impeller efficiency ratios ($\eta_{\text{imp}}/\eta_{\text{imp\ peak}}$) are presented in figures 6-13 and 6-14. (where $\eta_{\text{imp\ peak}}$ represents the peak efficiency of a state-of-art impeller, which will be used throughout this work). It can be seen that there is a significant difference between the predicted and experimental; the predicted efficiency ratio values being very flat and almost constant across the mass flow range. Compressor 2 shows particularly poor efficiency at low speeds. It was speculated that this was due to the oversized compressor casing, where the likelihood of flow re-circulation occurring along the shroud at low speeds was increased, resulting in increased losses and hence a fall in the expected efficiency. The flat curves of the prediction were a direct result of the empirical model used in CAPRICE2 and this clearly needed improving so that a curve prediction more suited to the shape of the experimental data could be produced.
Figure 6-13  compressor 1: experimental and CAPRICE2 impeller efficiency ratio

Figure 6-14  compressor 2: experimental and CAPRICE2 impeller efficiency ratio
6.4 Computational Analysis

The difficulties of adequately modelling the geometry and the complexity of the flow field in turbomachinery means that adequate validation is essential, to this end a CFD model of the impeller and diffuser of compressor 2 was constructed. Analytical results of the compressor were obtained using the turbomachinery CFD solver, CFX TASCflow 2.7.2 and its associated pre-processing software. The purpose of utilising such a CFD technique in the development of a 1-D prediction method was to enable the experimental data to be validated. A detailed understanding of the 3-D flow phenomena was not the main objective of this investigation; rather it was how these CFD data could be utilised for comparison with the 1-D model.

A numerical mesh for both the impeller and vaneless diffuser was constructed, although the volute was not modelled. A total mesh size of approximately 120,000 nodes was used to model one complete main vane and splitter vane passage with a corresponding sector of the diffuser passage. Blade tip clearance was modelled in the impeller vanes. Inlet conditions (mass flow rate, inlet temperature and impeller speed), measured from the experimental test, were used to define the boundary conditions for the model. Flow direction was defined as axial to the impeller inlet. Ambient total pressure and temperature were used for the inlet boundary conditions (standardised to 298K and 101.3 kPa), and mass flow rate for the outlet boundary condition (outlet of the diffuser). Turbulence was modelled using the standard k-ε model.

Seven stage operating points were modelled corresponding to experimental test points (four from speed line 3 and three from speed line 4). Numerical data - densities, mass-averaged static and stagnation temperatures and pressures - were extracted from the model at four planes corresponding to locations on the compressor as defined in chapter four. Flow parameters such as velocity and gas angle were then calculated.

The major difficulty in comparing the CFD analysis and experimental data was the difference between the 3-D and 1-D modelling techniques. Slight differences in numerical values such as geometry and boundary conditions contribute to differences between the experimental and CFD results. Material properties defined by the CFD such as surface roughness and material type may also have contributed to differences in the results.
However, taking these things into account in general the CFD and experimental data show good agreement, the 3-D model supporting the 1-D experimental work. The graphical results of these comparisons are presented in the following section, where appropriate.

### 6.4.1 Comparison of CFD & Experimental Characteristic

Figure 6-15 shows the comparison of the overall stagnation pressure ratio for the CFD and experimental data. The ratio was defined as the impeller inlet to diffuser exit, \( R_{\text{cfd}} = \frac{P_4}{P_0} \). Figure 6-15 shows good agreement between the two sets of data with the CFD data points following the curve of the experimental data extremely well. An average error of -0.06 for speed line at 81% of maximum speed and an average error of -0.1 for speed line 100% of maximum speed (shown as error bars in figure 6-15).
6.4.2 Comparison of CFD & Experimental Overall Efficiency

The overall efficiency was calculated between the impeller inlet and diffuser exit from the equation:

\[
\text{Overall efficiency} = \frac{T_{01}(P_{04}/P_{01})^{\gamma - 1}/(T_{04} - T_{01}) - 1}{T_{04} - T_{01}}
\]

Figure 6-16 shows a comparison between the CFD and experimentally-determined efficiency between stations {1} and {4}, for the two speed lines. The CFD data points for speed line at 100% of maximum speed show slightly lower values of efficiency, with an error of +0.025. Speed line at 81% of maximum speed shows remarkably good agreement with the experimental data, with a maximum error of +0.01. Since the overall efficiency is related to the performance of the components, it is necessary to determine their characteristics in order to improve the overall prediction.

![Figure 6-16 experimental and CFD predicted overall efficiency](image)
6.5 Summary

This chapter of the thesis has served to present some of the overall data from the experimental programme, in graphical form. It has shown that:

- The volute has a significant effect on flow distortions in the up-stream components of the compressor which are off-set by approximately 45° which describes the log spiral path around the volute.
- It is possible to determine the conditions at the impeller tip from the diffuser pinch based on a no loss assumption but that considerable errors will be incurred and must be accounted for.
- That the assumption of axial flow in the down-stream section of the compressor is incorrect and must be accounted for in the prediction model.
- That experimental data has proved to be suitable for validating the experimental data.

The issues arising from this initial study of the experimental data will be addressed and where possible incorporated into the CAPRICE prediction program in later chapters of this work. The key focus now turns to the component experimental data and how this can be used to develop new models for the CAPRICE code. Chapters seven and eight will look at these data in more detail and it will be shown that a ‘second stage’ improvement to the program is achieved.
7 CAPRICE2 Model Validation

Synopsis

This chapter presents a validation of the models presented in chapter four using the available experimental data. A new set of equations for the prediction of the impeller characteristic are presented as well as improvements to the new impeller work model by the addition of experimental data. The surge correlation is also improved and validation of the Mach number correlations of chapter three is also presented. Separation of the diffuser and volute characteristics is achieved and two equations to predict the characteristics are presented as a first stage to developing models for the CAPRICE program. New models for the prediction of surge in the diffuser pinch are presented as well as a correlation for the prediction of Mach number at the volute tongue.
7.1 Introduction

The analyses presented in chapter three demonstrated that with simple correlations significant improvements could be made to the CAPRICE code. With the addition of experimental data the validation of those models can be carried out.

The use of empirical modelling has been well proven over many decades and it is believed that the same approaches can be applied here. By applying the techniques and flow assumptions presented earlier in this thesis it will be shown that correlations of geometry and performance can be created and that these correlations can be incorporated as models into the CAPRICE code.

This chapter will demonstrate that:

- Improvements to the impeller model have been achieved by the development of a new correlation based on experimental data. This has improved on the previous model by better predicting the mass flow rate at peak efficiency, \( \frac{m_{\text{peak}}}{m_{\text{max}}} \) and by more accurately predicting the shape of the impeller characteristic over varying speeds.
- The experimental data has enabled the work equation given in chapter three to be validated and enabled this to be developed to accommodate impellers of different backsweeps.
- Additional improvements to the impeller surge predictions have been made.
- The experimental data has enabled the diffusion system to be separated into its constituent components and two correlations have been produced for the diffuser and volute, relating efficiency and impeller exit angle. These correlations are a first step to developing two separate models for the two components.
7.2 Impeller Model Improvements

Using the experimental data an analysis of the impeller characteristics has been undertaken. Whilst it was stated earlier that the impeller model in CAPRICE had been well tested and was assumed to be accurate the model was written some years ago and its accuracy could therefore be questioned. It was prudent to assess this accuracy against the available data and the following section presents the findings of this work.

7.2.1 Impeller Loss Coefficient

The impeller loss coefficient \( \zeta_s \) for the four compressors from figure 4-23 is presented (green points) in figure 7-1, along with the experimentally-derived loss coefficient data from compressors 1 and 2 (red points). Figure 7-1 shows the experimentally-determined \( \zeta_s \) to be highly scattered, but there is a distinct distribution to the data which is shown by the curve-fit and the expression relating incidence and loss coefficient:

\[
\zeta_s = 0.00033i^2 - 0.006913i + 0.164319 \quad \text{eqn. (7-1)}
\]

![Figure 7-1](image) compressors 1 and 2 impeller loss coefficient for varying inducer incidence
The majority of experimentally-derived \( \zeta \) are lower than those predicted from figure 4-23, but there are seven data points which are slightly higher and these correspond to compressor 2's lowest speed-line. From this comparison it was clear that the original impeller model did not accurately represent the shape of the impeller characteristic and this supported the need to improve the impeller loss model.

### 7.2.2 Impeller Correlations

The impeller characteristic in CAPRICE is presented in the form of an efficiency ratio against mass flow rate ratio as derived by Swain\(^9\) in section 3.2.1. For consistency, the experimental data was presented in the same form and is shown graphically in figure 7-2.

![Graph of impeller efficiency ratio vs mass flow rate ratio](image)

**Figure 7-2** compressors 1 and 2 impeller efficiency ratio for varying mass flow rate ratio

Correlating the experimental impeller efficiency ratio \( (\eta_{\text{imp}}/\eta_{\text{imp\ peak}}) \), (where \( \eta_{\text{imp\ peak}} \) is defined as the peak efficiency at each specified speed), against the proportion of flow to choke flow \( (m/m_{\text{max}}) \), for both compressors produced a distinctive shape to the data. It was believed that by predicting this shape the impeller model in the CAPRICE program could be improved.
Two methods of determining this impeller characteristic were evaluated:

- **A simple curve-fitting technique** – which only requires the blade speed and \( \dot{m}/\dot{m}_{\text{max}} \) from which to determine the efficiency ratio, \( \eta_{\text{imp}}/\eta_{\text{imp peak}} \) curves. The results of this correlation are shown in figure 7-3 and the full correlation method is presented in appendix 1, section I-vii.

- **Twin Efficiency Equation Technique** - presented by Swain in section 3.2.1., which divides the efficiency ratio curve into two equations either side of the peak efficiency and is shown in figure 7-5.

It was found that the model based on Swain’s technique, produced the more accurate results. This technique was however more complex, requiring the blade speed \( (U_2/\sqrt{T_{01}}) \), \( \dot{m}_{\text{peak}}/\dot{m}_{\text{max}} \) and \( \dot{m}/\dot{m}_{\text{max}} \) as well as equations (7-3) and (7-4), to find the efficiency ratio. However, the use of two efficiency equations enabled the prediction to be better matched to the experimental data. The approaches used in determining this result are presented in the following sections.

### 7.2.2.1 Simple Curve-fitting Technique

The predicted and experimental efficiency ratios for four speed lines (two for each compressor) are presented in figure 7-3, using a simple curve-fitting technique. The prediction produced the change in curve shape with speed as was required and the peak efficiencies also varied from low to high speeds in concert with the experimental data. However, there was poor correlation between the two sets of data, particularly at surge and this was believed to be as a result of:

1. Poor correlation of coefficient data points, as shown in figure I-3.
2. The application of parabolic curve-fits which did not match the experimental data accurately enough.
1. It could however, be argued that the model is not required to produce an exact prediction, but purely a representation of the expected characteristic shape and range. Although the predicted efficiency ratio curves were based on curve-fits of experimental data and highly scattered coefficients, it is believed that this technique made some improvement to the previous impeller characteristic prediction. This is supported by the inclusion of a predicted speed line at $U_2/\sqrt{T_0} = 14$ (blue dashed line) using equations 3-2 to 3-4 from section 3.2.1. It can be clearly seen that the original model produced an extremely flat curve across the whole range of mass flow functions and at the extremes of choke and surge the model was very poorly matched compared to the newer correlation.

### 7.2.2.2 Twin Efficiency Equation Technique

The second correlation method applied the technique used by Swain\textsuperscript{[9]}. This method divided the characteristic into two equations, one for either side of the efficiency at peak mass flow. As pointed out previously, the efficiency ratio curves are not truly parabolic and by splitting the curve in two allows different shapes to be better predicted.
By determining the position of the peak efficiencies, the blade speed \( \frac{U_2}{\sqrt{T_0}} \) to the mass flow rate at peak efficiency, \( \frac{m_{\text{peak}}}{m_{\text{max}}} \), can be found. Figure 7-4 shows the position of peak efficiency correlation (red and green points) for both compressors and a curve-fit shown as blue squares, up to a 'limiting value' of 0.89 (where any value of \( \frac{m_{\text{peak}}}{m_{\text{choke}}} \) above \( \frac{U_2}{\sqrt{T_0}} = 35 \) is limited to 89% peak efficiency). Two of the data points \( \frac{U_2}{\sqrt{T_0}} = 10 \) and 15 did not correlate well with the curve-fit and it was expected that the efficiency curves for these two speeds would be under predicted.

The empirical equation for this curve-fit line was of the form:

\[
\frac{m_{\text{peak}}}{m_{\text{max}}} = 0.01305092(U_2/\sqrt{T_0}) + 0.46329031 \quad \text{eqn. (7-2)}
\]
Figure 7-5 shows the efficiency ratio $\eta_{\text{imp}}/\eta_{\text{imp peak}}$ (where $\eta_{\text{imp peak}}$ is defined as a constant value) for varying mass flow ratios. The experimental data for four speed lines are shown as points. It can be seen that the impeller efficiency ratio model produced good correlations both in curve shape and speed range. It should be pointed out that the prediction of efficiencies at blade speeds, $U_2/\sqrt{T_{01}} = 10$ and 15 (not shown) were poorly predicted but this had been anticipated. It was believed that this prediction model could be incorporated into the CAPRICE program and its accuracy at predicting the impeller efficiency will be assessed in chapter eight.

Two equations were produced using combinations of $\dot{m}_{\text{peak}}/m_{\text{max}}$ and $m/m_{\text{max}}$ to find $\eta_{\text{imp}}/\eta_{\text{imp peak}}$, these being:

Above the peak efficiency flow rate:

$$\frac{\eta_{\text{imp}}}{\eta_{\text{imp peak}}} = 1 - \left( \frac{\dot{m}}{m_{\text{max}}} - \frac{\dot{m}_{\text{peak}}}{m_{\text{max}}} \right) \left( \frac{1.9 - \frac{\dot{m}_{\text{peak}}}{m_{\text{max}}}}{3.5} \right)$$

eqn. (7-3)
Below the peak efficiency flow rate.

\[
\eta_{\text{imp}} = 1 - \left( \frac{m_{\text{peak}}}{m_{\text{max}}} - \frac{m}{m_{\text{max}}} \right) \left( 1.7 + \frac{m}{m_{\text{max}}} \right)
\]

\text{eqn. (7-4)}

7.2.3 Impeller Work

Figure 7-6 shows the experimental work for compressors 1 and 2 and their respective curve-fit equations.
The impeller work for compressor 1, whilst conforming to the a power curve-fit presented in section 3.5.3, has significantly lower values of work than compressor 2 over the same flow range. This can be attributed to the difference between the impeller backsweep angles of the two compressors, (discussed in section 3.5 and supported by Wiesner\textsuperscript{[20]}), and accounts for the difficulty CAPRICE2 had in predicting the work of compressor 1, shown in figure 7-7.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{figure7-7.png}
\caption{experimental and CAPRICE2 predicted work}
\end{figure}
7.2.3.1 Comparison of CFD and Experimental Work

Figure 7-8 shows the comparison between the experimental and CFD impeller work. The characteristic shape of the work curve cannot be clearly seen in the CFD data, since only 7 data points were modelled but, the experimental and CFD data show good agreement, with a maximum error of only ±0.03.

![Graph showing comparison of CFD and experimental work](image)

Figure 7-8 experimental and CFD predicted impeller work

7.2.3.2 Improved Work Model

The data from figure 7-6 were used to improve the work correlation produced in chapter three, equation (4-9) and were incorporated into the tan beta/power relation shown in figure 4-29. Using this relationship, equation (4-9) was modified to:

\[
\frac{\Delta H}{U_2^2} = 10 \left( \frac{V_{r_2}}{U_2} \right)^{-0.275 \tan(\beta_2) - 0.163}
\]

eqn. (7-5)

Figure 7-9 shows the difference between the predicted power curve, based on equation (4-9) (dashed lines) and the newly predicted power curves (solid lines) based on equation (7-2), along with the experimental data (points). As can be seen there is some improvement to the
original work prediction, and although compressor 1 is of a different impeller type, the prediction holds well with the experimental. It is suggested that this improved work prediction could be applied with some success to other compressors with different impellers and that further improvement to the work prediction would be gained with more experimental data.

![Figure 7-9 power curve fits with CAPRICE2 work eqn. 3-26 and new prediction eqn. 5-2](image)

### 7.2.4 Impeller Surge Correlations

The inducer incidence and impeller exit angle at surge were determined from the experimental data and are shown in figures 7-10 and 7-11 as blue and red points. These data were combined with the two sets of curve-fits from figures 4-30 and 4-31 and are shown as dashed brown and green lines. Even though the two compressors show similar surge ranges over the specified speeds to those shown in chapter 3, it can be seen that there is a significant difference between the experimental data points and the curve-fits, particularly for compressor 1. The error between the experimental data points and the curve-fits increased as the speed increased. For example, below $U_{1_{rms}}/\sqrt{T_{01}} = 10$ the averaged error was ±0.8°, but this increased to ±1° above $U_{1_{rms}}/\sqrt{T_{01}} = 10$. This was also the case for the impeller exit, where the averaged error below $U_{2}/\sqrt{T_{01}} = 19$ was ±0.5° but above this speed the error increased to ±1.5°.
Figure 7-10  compressor 1 and 2 inducer incidence at surge points and curve-fits, including improved curve-fits using combined experimental and fig 3-43 data

Figure 7-11  compressor 1 and 2 impeller exit angle at surge points and curve-fits, including improved curve-fits using combined experimental and fig 3-44 data
Curve-fits of the experimental data are shown as solid green and brown lines in both figures, with only two data points (in figure 7-10) being in error >2°. The equations which describe these curve-fits are presented below and will be assessed in the new CAPRICE program in chapter 9.

Inducer incidence:

Below \( U_{1rms}/\sqrt{T_{01}} = 10.7 \)

\[ i_s = -0.8186(U_{1rms}/\sqrt{T_{01}}) + 30.064 \]  
\[ \text{eqn. (7-6)} \]

Above \( U_{1rms}/\sqrt{T_{01}} = 10.7 \)

\[ i_s = -2.6437(U_{1rms}/\sqrt{T_{01}}) + 49.539 \]  
\[ \text{eqn. (7-7)} \]

Impeller exit:

Below \( U_2/\sqrt{T_{01}} = 19.5 \)

\[ \alpha_{2s} = -0.3758 \left( U_2/\sqrt{T_{01}} \right) + 84.64 \]  
\[ \text{eqn. (7-8)} \]

Above \( U_2/\sqrt{T_{01}} = 19.5 \)

\[ \alpha_{2s} = -0.8765 \left( U_2/\sqrt{T_{01}} \right) + 93.424 \]  
\[ \text{eqn. (7-9)} \]

7.2.5 Inducer and Diffuser Pinch Mach Number at Surge

Figure 7-12 shows the relationship of inducer throat relative Mach number (blue points) and the diffuser pinch diameter absolute Mach number (red points) along the surge line for the two tested compressors, including curve-fits of the data from figure 4-32 (solid lines). The two sets of data correlate well with the curve-fits, showing similar values of Mach number at the specified speeds. However, compressor 1 shows a slightly lower range of relative Mach numbers and is confirmed by the fact that the impeller inlet velocity is lower than that of compressor 2. It should also be noted that the absolute Mach number of compressors 1 and 2 do not follow the inducer curve-fit at lower speeds, instead they match more closely the diffuser curve-fit. This would suggest that the compressors tested had greater levels of instability in their diffusers at low speeds than the four compressors from figure 4-32. Also,
the inducer and diffuser Mach numbers for compressor 1 cross over at $U_2/\sqrt{T} = 17$, whilst compressor 2 crosses just below $U_2/\sqrt{T} = 14$. This would suggest that there is a 'grey area' for each compressor in which the instability moves from the impeller to the diffuser as speed increases and in which it would be difficult to determine which component was limiting the mass flow rate.

The curve-fit equations for the two sets of data were:

Inducer throat Mach $N^o$, $M_{rel} = 0.03761(U_2/\sqrt{T_0}) - 0.010884$

Pinch diameter Mach $N^o$, $M_{abs} = 0.02317(U_2/\sqrt{T_0}) + 0.194696$

![Graph showing Inducer and diffuser Mach number for compressors 1 and 2](image-url)
7.3 The Diffusion System

The diffusion system efficiency ratio ($\eta_d/\eta_{d\text{ peak}}$) was determined from the experimental data of both compressors and is shown in figure 7-13, along with the curve-fit from figure 4-21. As expected, compressor 2 shows reasonable agreement with the curve-fit but compressor 1 shows a much broader characteristic over a wider impeller exit angle range, with little drop-off in efficiency between the peak and the extremes of surge and choke. Some scattering can be seen at the choke end, but this is not as pronounced as that of compressor 2. The findings of Ludtke\textsuperscript{[72]} in section 2.2.2., support these observations and suggest that the diffusion system of compressor 1 was better suited to the range of operating speeds and mass flow rates than compressor 2.

![Figure 7-13: Diffusion system efficiency ratio for compressors 1 & 2 with curve fit from fig. 3-34](image)

Dean\textsuperscript{[24]} suggested that the optimum velocity ratio for stable operation of a vaneless diffuser would be in the range $V_{w2}/V_{r2} = 2 - 3$ (tan $\alpha_2 = 63.4° - 71.6°$). Compressor 2’s averaged $V_{w2}/V_{r2} = 2.75$ ($\approx 70°$) at peak efficiency ratio and compressor 1’s averaged $V_{w2}/V_{r2} = 1.73$ ($\approx 60°$) at peak efficiency ratio, confirming that the two compressors diffusion systems were operating in a stable range. It was also thought by Brown\textsuperscript{[30]} that the diffuser efficiency could be effected by a reduction in the diameter ratio $D_4/D_2$. Since the $D_4/D_2$ of compressor
was smaller than that of compressor 1 and compressor 2's efficiency ratio was therefore smaller, this would seem to substantiate this assumption.

7.3.1 Comparison Of CFD And Experimental Component Efficiency

The experimental and CFD diffuser efficiency ratios are shown in figure 7-14 at two speed lines. It can be seen that the CFD-predicted efficiency is slightly lower than the experimental data with minor variations in exit gas angle. It could be argued that since the CFD model did not include the volute, the up-stream effects of this component have not been assimilated into the CFD diffuser efficiency result. Combine that with the differences associated with 1-D and 3-D models and the discrepancy between the experimental and CFD predicted efficiencies can be justified. Since there were only seven points modelled by the CFD, it is not possible to determine the shape of the efficiency curves or to determine the peak efficiency, however, it is clear that the CFD prediction does follow the shape of the curve fit of the experimental data.

![Figure 7-14: Compressor 2 CFD & experimental diffuser efficiency](image-url)
7.3.2 Separating the Diffusion System Components

The diffuser and volute characteristics have to be separated using the experimental data from the volute and diffuser and the programs discussed in section 5.2. The flow conditions inside the vaneless diffuser and volute scroll were determined and the following section looks at the experimental characteristics and efficiencies of the diffuser and volute separately and compares them with the diffusion system correlation of chapter three.

The volute and diffuser efficiency ratios for both compressors are shown in figures 7-15 and 7-16. The diffuser efficiency ratio \( \eta_d/\eta_d\text{ peak} \) was defined between stations {3} to {4} and the volute efficiency ratio \( \eta_v/\eta_v\text{ peak} \) was defined between stations {4} and {7}. The peak diffuser efficiency \( \eta_d\text{ peak} \) is a constant value and represents the peak efficiency of a state-of-art diffuser. Likewise, the peak volute efficiency \( \eta_v\text{ peak} \) is a constant value. Both sets of data have been correlated against impeller exit gas angle \( \alpha_2 \) and for comparison and the curve-fit of diffuser system efficiency ratio from figure 4-21 (purple line) has been included in figure 7-15.

In chapter three section 3.4.2. it was postulated that the volute would be the dominant component of loss in the diffusion system, particularly at high speeds and mass flow rates, due to flow separation around the tongue, resulting in a narrowing of the characteristic. It is clear in figure 7-16 that the two volutes have lower efficiency ratios than the diffusers, for both compressors, from which it can be assumed that there is a greater level of loss in the volutes, particularly at choke flows \( (\alpha_2 < 65^\circ) \). The blue lines in figures 7-15 and 7-16 represent the curve-fits of the experimental data for both compressors and the equations were expressed:

**Diffuser efficiency ratio.**

\[
\eta_d/\eta_d\text{ peak} = -0.0002144\alpha_2^2 + 0.0212867\alpha_2 + 0.3990245
\]

**Volute efficiency ratio.**

\[
\eta_v/\eta_v\text{ peak} = -0.0003017\alpha_2^2 + 0.0447386\alpha_2 - 1.0344745
\]
Figure 7-15  compressors 1 & 2 diffuser efficiency ratio

Figure 7-16  compressors 1 & 2 volute efficiency ratio
Whilst these two expressions go some way to describing the range and level of efficiency in the two components they do not adequately describe the change in shape of the curves with speed and the significantly large spread of volute efficiency data. It was therefore necessary to consider developing a correlation which would describe the diffuser and volute characteristic curve shapes and ranges. A more detailed analysis of the volute and diffuser using a 'loss coefficient' approach is presented in chapter eight.

### 7.3.3 Diffusion System Surge

A relatively small quantity of work has been produced on the prediction of surge in the vaneless diffuser and volute. However, it is clear that the prediction of this phenomenon, in both cases, is of great importance in determining the operating range and characteristic map of the compressor. The effects of surge in these two components is not taken into account in the original CAPRICE program, resulting in a straight surge line on the compressor maps and not the distinctive 'kinked' surge line denoting the transition of surge from the impeller to the diffusion system. Using the experimental data, the flow conditions in the diffuser and volute at surge have been determined and these have been correlated with gas angle and incidence angle receptively.

Unlike the vaned diffuser, where flow separation has been shown to occur around the vane tip, (section 2.2.3), the location of flow separation and the onset of surge is difficult to locate in the vaneless diffuser. The most likely location is its narrowest point, i.e. the pinch diameter, where at low mass flow rates the flow may separate on the shroud side. The variation of diffuser pinch exit angle at surge with blade speed was determined from the experimental data and is shown in figure 7-17. As can be seen there are some differences between the gas angles at surge for the two compressors, over the range of blade speeds. It was speculated that the difference between the range of diffuser pinch gas angles was due to the difference in component geometry, but, with only two compressors tested it was not possible to verify this hypothesis. It was however, possible to produce two straight-line curve fits of these data (brown dashed line $10 < U_2/\sqrt{T_{01}} \leq 20.3$ and green dashed line $20.3 < U_{rms}/\sqrt{T_{01}} \leq 28$). Whilst two of the data points for compressor 2 correlated poorly with these curve fits, particularly at speeds of $U_2/\sqrt{T_{01}} = 10$ and 24, the majority of data points were within $\pm 1.5^\circ$ of the curve fits.
The equations for the curve fits are given as:

Below $U_2/\sqrt{T_{01}} = 20.3$

$$\alpha_{3s} = -0.1865(U_2/\sqrt{T_{01}}) + 79.761 \quad \text{eqn. (7-10)}$$

Above $U_2/\sqrt{T_{01}} = 20.3$

$$\alpha_{3s} = -0.5751(U_2/\sqrt{T_{01}}) + 87.638 \quad \text{eqn. (7-11)}$$

Section 1.3.4 gave a brief explanation of the affect the volute tongue had on the scroll flow conditions, however there is little documentary evidence to demonstrate this effect. At low mass flow rates the exit angle from the diffuser is large, making the tangential angle smaller than the tongue angle. It was believed that by determining the incidence at surge between the tongue and the gas angle it would be possible to determine the surge condition in the volute. The tongue angle was calculated using the technical drawings provided.

Figure 7-18 shows the incidence angle at surge for varying blade speeds. The incidence at surge between highest and lowest speeds was no greater than $9^\circ$. ($21^\circ$-$26^\circ$ for compressor 1 and $15^\circ$-$23^\circ$ for compressor 2). Again, two straight-line curve fits of these data were
determined (brown dashed line $10 < U_2/\sqrt{T_{01}} \leq 22.2$ and green dashed line $22.2 \leq U_{1rm}/\sqrt{T_{01}} \leq 28$). The majority of data points were within $\pm 1.7^\circ$ of the curve fits even though compressor 2 again correlated poorly with these curve-fits at speeds of $U_2/\sqrt{T_{01}}=10$ and 23.

The equations for the curve fits are given as:

Below $U_2/\sqrt{T_{01}}=22.2$

$$i_{s, \text{vol}} = -0.0711(U_2/\sqrt{T_{01}}) + 26.65 \quad \text{eqn. (7-12)}$$

Above $U_2/\sqrt{T_{01}}=22.2$

$$i_{s, \text{vol}} = -0.4245(U_2/\sqrt{T_{01}}) + 33.797 \quad \text{eqn. (7-13)}$$
7.3.4 Volute Mach Number at Surge

The volute tongue absolute Mach number $M_{\text{abs vol}}$ was also determined and was correlated along with the diffuser pinch absolute Mach number $M_{\text{abs dif}}$ over the range of blade speeds, and is shown in figure 7-19. The two compressor's $M_{\text{abs vol}}$ varied by an average of 0.05 across the whole range. This was shown to be due to the difference in absolute velocity for compressor 2, which was slightly higher than that of compressor 1 for similar values of static temperature.

The volute curve-fit equation for the two sets of data was:

$$M_{\text{abs vol}} = 0.012907(U_2/\sqrt{T_{01}}) + 0.15453 \quad \text{eqn. (7-14)}$$

An assessment of the Mach number prediction and its effects on performance will be presented in chapter nine.
7.4 Summary

The following correlations have been presented using the experimental data:

- Impeller efficiency ratio for varying mass flow ratio and blade speed
- Impeller work for varying flow coefficient and given backsweep angle
- Inducer and impeller exit surge for varying blade speed
- Impeller, diffuser and volute Mach number for varying blade speed
- Diffuser and volute efficiency ratio for varying impeller exit gas angle
- Diffuser and volute surge for varying blade speed

Algebraic equations have been created from these correlations and are to be incorporated into the CAPRICE prediction program.

An analysis of the component efficiencies derived from the experimental data revealed that the impeller model was poor at predicting the efficiency of the impellers tested. Two techniques were tested and the second, using the relationship between mass flow to choke mass flow and efficiency ratio, resulted in an improved prediction which more accurately predicted the shape of the experimental data.

Additionally, the impeller work correlation produced in chapter three was improved by the inclusion of the experimental data from compressors 1 and 2. The impeller surge predictions were also up-dated using the experimental data and new forms of the expressions produced in chapter three were developed.

The experimental data enabled the diffusion system to be divided into its constituent components namely the vaneless diffuser and volute casing. Expressions describing the diffuser and volute efficiency ratios against impeller exit gas angle were produced. It will be shown in chapter eight that further sub-dividing the diffusion system will enhance these new correlations still further.

Correlations of blade speed with diffuser pinch gas angle and volute incidence at surge were produced. The resulting curve-fits produced reasonable agreement with the experimental data resulting in an averaged error of ±1.5°.
7.5 Review of Presented Work

So far, points 1 to 5 of the research plan presented in section 4.8 have been carried out. These being:

1. Development of an experimental test program
2. Analyse the data.
3. Determine the accuracy of the data.
4. Establish a series of suitable correlations using the experimental data.
5. Verify the data and correlations against recognized techniques and results.

The final stage of improving the 1-D performance prediction technique is to develop a set of correlations for the diffusion system components using the experimental data. Chapter eight will show in some detail the different techniques which have been applied to the data to determine the most appropriate method for predicting the diffuser and volute characteristics. It will be shown that this can be achieved by using a loss coefficient correlation. Finally, an assessment of the correlations is presented in chapter nine and predicted compressor maps for the new CAPRICE program, using the new correlations which have been incorporated into the code, are presented.
8 Developing The Diffuser And Volute Loss Model

Synopsis

This chapter focuses on the diffuser and volute flow conditions and determines the characteristics in terms of non-dimensional coefficients. Available techniques, such as simple pipe models and methods proposed by Rogers, Van Den Braembussche and Runstadler have been compared to the experimental data and an assessment of their suitability for inclusion into the 1-D performance prediction program is presented. Two techniques have been selected; one for each of the components, correlating the geometric features with the non-dimensional coefficients.
8.1 Introduction

The central focus of this research was to produce a set of correlations to improve the prediction program CAPRICE. The final stage of this development process is to determine the vaneless diffuser and volute casing performance characteristics using their main geometric features in conjunction with the experimental data from chapter five. To achieve this, relationships between the geometry and flow conditions needed to be established.

As discussed in chapter one, the most common way to examine performance, particularly between different compressors, is to use non-dimensional functions such as efficiency, static pressure recovery or stagnation pressure loss coefficients. This chapter looks in more detail at the diffusion system and uses the experimental data from the compressors tested, to determine non-dimensional loss and recovery coefficients for the diffuser and volute.

The experimental data has been compared to a number of existing prediction techniques which use non-dimensional models, namely those produced by Rogers\cite{28}, Runstadler\cite{60} and Van Den Braembussche\cite{8284,851,8284}, to determine if they can be applied to this particular prediction technique. It will be shown that whilst some of these techniques are too simplistic, even for a 1-D prediction technique, others give good agreement with the experimental data. These correlations have been incorporated into the CAPRICE prediction program and will be assessed in chapter nine.

8.2 Proportioning Losses

The overall performance characteristic maps and diffusion system efficiency figures presented in chapter seven go some way to describing the behaviour of the vaneless diffuser and volute casing. However, further study of the individual component data was required to create two separate models of the diffuser and volute for use in the CAPRICE program. By proportioning and determining the non-dimensional losses within the components, correlations can be made which enable the two compressors to be compared. This, it was believed, would yield a general method of prediction that could be applied to any diffusion system.
The types of losses discussed in chapter one can be difficult to identify, particularly with limited experimental and geometric data. Typically, the losses are considered as a 'lumped' loss component with the loss due to skin friction believed to be the predominant constituent loss (it being the easiest to determine based on geometry). Skin friction factor can be determined from the stagnation pressure loss between two points, from equation [16]:

\[ \Delta P = (4C_f L/D) \frac{1}{2} \rho V^2 \]

Haarland [122] produced a simple expression for skin friction factor relating not only the Reynolds number but the relative roughness of a pipe as well and this equation has been used to determine the friction factor for both sets of compressor data.

\[ \frac{1}{\sqrt{C_f}} = -3.6 \log_{10} \left( \frac{6.9}{Re} + \left( \frac{K}{3.71D_{eq}} \right)^{1.11} \right) \]

The value of \( C_f \) can vary with compressor size, surface roughness, and Reynolds number between 0.005 and 0.01 and there were small variations in the values of skin friction factor for both compressors (between 0.00698 and 0.007040). It was therefore deemed acceptable to use a constant value of approximately 0.007 which Rogers [128] used to good effect. Using this value for skin friction factor and the equations (1-37) and (1-38), the loss coefficient and static pressure recovery for the diffuser and volute could be determined.

Table 8-1 shows how the compressor losses were divided up. Here, attention will be focused on the station \( \{3\} \) to \( \{4\} \), \( \{4\} \) to \( \{6\} \) and \( \{6\} \) to \( \{7\} \).

<table>
<thead>
<tr>
<th>Impeller Exit {2}</th>
<th>Diffuser Exit {3}</th>
<th>Diffuser Exit {4}</th>
<th>Volute Exit {5}</th>
<th>Volute Exit {6}</th>
<th>Volute Exit {7}</th>
</tr>
</thead>
<tbody>
<tr>
<td>2-3</td>
<td>3-4</td>
<td>4-5</td>
<td>5-6</td>
<td>6-7</td>
<td>7-8</td>
</tr>
<tr>
<td>2-4</td>
<td>3-5</td>
<td>4-6</td>
<td>5-7</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>3-6</td>
<td>4-7</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 8-1

The following section presents the diffuser and volute loss coefficients, extracted from the experimental data, in graphical form and identifies some of the key features.
8.3 Experimental Losses in the Diffuser and Volute

8.3.1 Experimental Diffuser Loss Coefficient

The diffuser stagnation pressure loss coefficient \( \zeta_d = (P_{03}-P_{04})/(P_{03}-P_3) \), between stations \{3\} and \{4\} was determined from the experimental data using equation (1-19). Figure 8-1 shows \( \zeta_d \) for varying diffuser pinch gas angles, over the range of rotational speeds for both compressors. Since the stagnation pressure was calculated and not measured, it was accepted that \( \zeta_d \) would not be an exact value. Never-the-less, it was believed that a correlation of this slightly scattered data could be made with some level of accuracy.

Both compressors showed only a small variation in \( \zeta_d \) from surge to choke with the majority of values lying between 0.08 to 0.25. Dean et al.\(^{123}\) suggested that for minimum \( \zeta_d \) the velocity ratio \( V_{w2}/V_{r2} \) should be approximately 2 (\( \alpha_2 \approx 63.44^\circ \)). The minimum value of \( \zeta_d \) for compressor 1 was at a gas angle \( \alpha_3 \approx 46.33^\circ \) (\( V_{w3}/V_{r3} \approx 1.07 \)) and for compressor 2 was \( \alpha_3 \approx 62.14^\circ \) (\( V_{w3}/V_{r3} \approx 1.99 \)). This slight variation in angle between those suggested by Dean at \( \alpha_2 \) and those found experimentally at \( \alpha_3 \) were to be expected and are acceptable within the bounds of the 1-D flow assumptions.
8.3.1.1 Comparison of CFD & Experimental Loss Coefficient

The CFD data presented in chapter six for compressor 2 were used to determine loss coefficient values and validate the accuracy of the experimental data for the mass flow rates tested. Figure 8-2 shows a comparison of the two sets of data at two speeds. The CFD points and experimental data correlated well even though the experimental data exhibits some fluctuations; small errors being attributed to the mass-averaging of the CFD data and the inherent experimental errors. The minimum experimental values of $\zeta_d$ were approximately 0.107 at $U_2/\sqrt{T_{01}} = 24$ and 0.124 at $U_2/\sqrt{T_{01}} = 20$ with $\alpha_3 = 49^\circ$ and $66^\circ$ respectively, but the limited number of points from the CFD analysis meant that it was not possible to determine the position of minimum $\zeta_d$. The error between the experimental and CFD data was approximately ±0.01 at $U_2/\sqrt{T_{01}} = 24$ and ±0.05 at $U_2/\sqrt{T_{01}} = 20$.

![Figure 8-2 comparison of the CFD and experimental stagnation pressure loss coefficient](image-url)
8.3.2 Experimental Volute Loss Coefficient

The losses associated with the volute are a result of turbulent mixing as the flow enters the scroll from the diffuser. Figures 8-3 and 8-4 show the proportioning of losses between the diffuser exit {4}, volute scroll throat {6} and volute scroll exit {7} for compressors 1 and 2 and how they contribute to the overall volute loss coefficient $\zeta_v$, between {4} to {7}.

The loss coefficients are divided up as follows:

- Volute scroll loss coefficient $\zeta_{\text{scroll}}$ {4} to {6} hollow points
- Whole volute loss coefficient $\zeta_v$ {4} to {7} solid lines
- Exit diffuser loss coefficient $\zeta_{\text{exit}}$ {6} to {7} solid points

Although the size of the two compressors is different, it can be seen that the overall volute loss coefficient $\zeta_v$ for both compressors is similar, within a region of 0.4 to 0.8. However, the two contributing loss coefficients ({4} to {6} and {6} to {7}) are distinctly different.

![Figure 8-3 compressor 2: volute stagnation pressure loss coefficient between stations 4 to 6, 4 to 7 and 6 to 7](image-url)
The exit stagnation loss coefficient $\zeta_{\text{exit}}$ (solid points) of compressor 2 is extremely small relative to the whole volute loss coefficient $\zeta_V$, suggesting that the majority of energy dissipation occurs in the scroll section. This would be consistent with the comments made in sections 4.7 that the volute casing for compressor 2 was larger than average. Closer inspection of $\zeta_{\text{scroll}}$ between 4 and 6 shows that from the minimum loss value to choke ($\alpha_4 = 50^\circ$ to $63^\circ$), $\zeta_{\text{scroll}}$ is almost constant for compressor 2. Although the $\zeta_{\text{exit}}$ is small, it is a contributing factor to the total stagnation pressure loss coefficient, producing the typical loss curve shape (shown as in solid lines).

Compressor 1 has a much higher level of $\zeta_{\text{exit}}$ between stations 6 and 7 than compressor 2, and the total loss coefficient $\zeta_V$ is dominated by $\zeta_{\text{exit}}$ (between choke and the minimum loss values $\alpha_4 = 35^\circ$ to $60^\circ$). As the compressor moves towards surge the $\zeta_{\text{scroll}}$ increases and becomes the dominant factor. $\zeta_{\text{scroll}}$ of compressor 1 is not constant between minimum loss and choke like that of compressor 2, instead, it continues to fall in the choke region, suggesting that at choke the fluid in the exit diffuser is highly swirling.

Figure 8-4 compressor 1: volute stagnation pressure loss coefficient between stations 4 to 6, 4 to 7 and 6 to 7.
8.3.3 Conclusion

Section 8.3 has presented the experimentally-determined stagnation pressure loss coefficients for the diffuser between (3) and (4) and volute from (4) to (6) and (6) to (7). The diffuser data has been validated against CFD data and has been found to be in good agreement. It has been shown that proportioning the losses within the two components allows the performance of the components to be easily identified.

The volutes of both compressors showed a significantly higher level of total loss than that of the diffuser which was to be expected from these components. The key benefit from the experimental data was keenly highlighted in figures 8-2 and 8-3 where the loss coefficients from several sections of the volute where presented. These clearly show that whilst the overall volute loss coefficient for both compressors are relatively in the same range, the loss coefficients that make up that overall value clearly behave in distinct ways. No corroborating evidence has been found in other works to support this attribute. But it is postulated that subdivision of the volute characteristic into two sections would be beneficial to the prediction program as it would provide a greater degree of accuracy to the performance prediction.

It was also noticeable that compressor 2's loss coefficient values were higher than that of compressor 1 and this was believed to be related to the differing geometries of the two diffusers. To ascertain whether or not this assumption was correct an analysis of the effects of diffuser geometry on performance was carried out. The following section discusses the findings of this analysis.

8.4 Variation of Diffuser Exit Gas Angle with Diffuser Geometry

One of the main features of the turbocharger compressors supplied by GEBS is the single impeller designed to operate with a range of volute casings of varying sizes. This provides a valuable commercial benefit, in that the compressor can be used over a broader operating range without the need for a range of individually-optimised compressors. Due to reasons of availability, the volute casing used for compressor 2 was slightly larger than the 'optimum' one for that particular range of compressors. Unfortunately, there were no experimental data available for the optimised version of the compressor and therefore no
direct comparison with compressor 2 - with respect to the affects of casing size on the performance and flow conditions - could be carried out. It was however, believed possible to speculate on these affects based on the experimental data from the tests and the application of a series of simple assumptions. This section looks at the variation of diffuser exit angle with changes in diffuser geometry to assess the affects on loss.

Brown et al\textsuperscript{31} suggested that the affects on performance were only slight when mismatches occurred between the diffusion system and the impeller. Since changes in the size of the volute casing would result in variations in the diffuser width, it was surmised that this would influence the flow conditions through the diffuser. These changes in flow conditions would affect the diffuser performance and hence could be quantified in the form of a loss coefficient.

Initially, three assumptions were made:

- The path traced out by the flow through the diffuser was a logarithmic spiral.
- The conditions at the impeller exit were constant.
- The diffuser width, $b_3$ was the only variable.

### 8.4.1 The Log Spiral Path

In the vaneless diffuser the fluid does not travel in a straight line - as is assumed in the original CAPRICE program - but follows a log spiral path, as discussed section 1.3.3.1. It was surmised that by determining the length of the flow path based on a logarithmic spiral, a more realistic value of the LWR could be found.

Firstly, consider a small section of the curve between the diffuser pinch and diffuser exit. The log spiral curve is $s$ long and travels an angle $\phi$ from its point on the inner arc to the outer arc at an angle $\theta$ to the tangential, $(90^\circ - \alpha^\circ)$.

$$\delta s^2 = r^2 \delta \phi^2 + \delta r^2 \quad \text{as} \delta \phi \longrightarrow 0$$
\[ \left( \frac{ds}{d\phi} \right)^2 = r^2 + \left( \frac{dr}{d\phi} \right)^2 \]

\[ \frac{ds}{d\phi} = \sqrt{r^2 + \left( \frac{dr}{d\phi} \right)^2} \]

\[ r_4 = r_3 e^{\tan \theta_3} \]

And differentiating gives:

\[ \frac{dr}{d\phi} = \tan \theta_3 r_3 e^{\tan \theta_3} \]

Thus for a small section of the curve:

\[ \frac{ds}{d\phi} = \sqrt{\left( r_3 e^{\tan \theta_3} \right)^2 + \left( \tan \theta_3 \cdot r_3 e^{\tan \theta_3} \right)^2} \]

Integrating with respect to $\phi$:

\[ s = \int r_3 e^{\tan \theta_3} \cdot \sqrt{1 + \tan^2 \theta_3} \ d\phi \]

\[ s = (\sec \theta_3) \cdot r_3 \int e^{\tan \theta_3} \ d\phi \]

\[ s = \left( \frac{r_3}{\sin \theta_3} \right) \left[ e^{\tan \theta_3} \right]_0^\phi \]

\[ s = \cosec \theta_3 r_3 \left\{ e^{\tan \theta_3} - e^{-\tan \theta_3} \right\} \]

At the inner arc, $\phi_3 = 0^\circ$ and therefore the equation can be written:

\[ s = r_3 \cosec \theta_3 \left\{ e^{\tan \theta_3} - 1 \right\} \]

\[ s = r_3 \cosec \theta_3 \left\{ e^{\tan \theta_3} - 1 \right\} \quad \text{eqn. (8-3)} \]

From $r_4 = r_3 e^{\tan \alpha}$, $r_4 = \frac{D_4}{D_3} = DR$ and solving for $\phi$, $s$ can be written:

\[ s = \cosec \theta_3 r_3 \left\{ DR - 1 \right\} \]

\[ s = \cosec \theta_3 r_3 \left\{ DR - 1 \right\} \quad \text{eqn. (8-4)} \]

This method of determining the length of the flow path only requires the inner and outer diameters of the diffuser and the inlet gas angle to be known\[126]. The relationship of the
angle $\theta_3$ to the log spiral flow path length $'s'$ is shown in figure 8-5 and is a power curve of the form $y = ax^{-b}$.

![Figure 8-5 relationship of gas angle to log spiral flow path length](image)

It was noted that there was a small difference in the experimentally-determined inlet gas angles at station \{3\} and exit gas angles at station \{4\}. For compressor 1 the difference was approximately 2.3° and approximately 1.5° for compressor 2. It could be argued that the assumption of constant angle through the diffuser does not correctly describe the actual flow path, but it was found that this simplified model had only a minimal effect on the diffuser exit velocity vectors. Even at the extremes of flow, i.e. at choke, where differences between the experimentally and numerically-determined exit angles were greatest, the averaged difference between them was approximately 0.07° (the majority were below 0.05°), with the log spiral method producing slightly larger angles at exit than the experimentally-determined. This model is used later to determine a correlation between stagnation loss coefficient and LWR.
8.4.2 Variations in Diffuser Width

The geometry of compressor 2’s diffuser was used to determine the effects of changing width on the performance. Equation (1-17) and (1-18) were used to calculate the static pressure recovery coefficient from impeller exit $t_2$ to diffuser exit $t_4$. The skin friction factor was calculated using equation (8-2) and the impeller exit gas angle was assumed constant for each change in diffuser width. Since the gas angle and the diffuser radii were constant, the log spiral path length also remained constant for each change in width. The diffuser width was adjusted by ±40% of that used on the tested compressor and the recovery coefficient $C_{pr}$ was calculated using equation (1-20). Figure 8-6 shows the variation in recovery coefficient with flow coefficient $V_{r2}/U_2$ for varying diffuser width.

![Figure 8-6 Variation of static pressure recovery with flow coefficient for varying diffuser widths](image)

It can be seen that for a given flow coefficient, as the diffuser width increases the static pressure recovery increases. The greatest increase in recovery being between $V_{r2}/U_2 = 0.1$ and 0.5 (surge to mid flow). The averaged static pressure recovery coefficient changed by approximately 20% between the original width and -40%, compared with only an averaged 9% change between the original width and +40% of the original width. It was also noted that the percentage difference between the experimentally-determined and numerically-determined exit angles also increased by approximately 0.2° as the width of the diffuser reduced.
8.4.3 Conclusion

The variation in pressure recovery coefficient suggests that as the diffuser is widened the net gain in pressure recovery reduces, and that for a particular geometry there will be a limiting diffuser width for which no useful gain in recovery will exist. From this analysis it was believed that even though a slightly larger volute casing had been used, the values of loss in the diffuser would not be excessively greater than those extracted from an 'optimised' casing. It was therefore concluded that using the experimentally-determined flow conditions for compressor 2 to determine the diffuser characteristic was acceptable within the limits of a 1-D prediction technique, whilst recognising that they would be slightly higher than would normally be expected.

8.5 Assessing Different Types of Loss Modelling Techniques

Having divided up the volute and vaneless diffuser into sections, a method of predicting the characteristics in each of section was required. Six methods for predicting the volute and diffuser characteristic are presented here, each being compared to the experimentally-determined values to assess its suitability for use in the CAPRICE program. Firstly, three simple models were considered.

- A straight pipe model of mean diameter assuming a linear flow through the pipe.
- A straight pipe model of mean diameter assuming a swirling flow through the pipe.
- An equivalent geometry of $LWR_{eqv}$, $AR_{eqv}$ and $20_{eqv}$, based on the straight-walled diffuser model of Runstadler.

The reasons for assessing these three methods is as follows:

- The simple geometric shape of a pipe is easily visualised.
- Data on pipe flow models is widely available, which provides a means of validation of the determined losses.
- The advantage of modelling the vaneless diffuser and volute using the Runstadler technique enables the experimental data to be validated against existing empirical evidence.
It will however, be shown that the first two of these models prove to be insufficient at producing comparable data, mainly as a result of their over simplicity.

Additionally, a diffuser model for determining static pressure recovery, presented by Rogers, is assessed. This produced favourable results in predicting the diffuser loss compared to the pipe loss model. As a consequence of this, a more suitable value of diffuser width was determined. The only draw-back to this technique was that it was unable to distinguish between the shapes of the loss coefficient curves at varying speeds.

A volute loss model presented by Van Den Braembussche et al[83] is also presented, but this was also found to poorly predict the shape of the curves at varying speeds and was particularly poor at choke prediction.

Two empirical models were also determined for the volute and diffuser. Both were found to have limited success in determining the loss coefficients, but scattered data, particularly in the diffuser’s case, resulted in poor predictions at several speed lines.

The following sections describe the approaches used and the results and discussion of fitness for purpose of each technique, culminating in the selection of the most appropriate method which will be incorporated into the CAPRICE program for evaluation.

8.5.1 Runstadler Equivalent Model

It was postulated that if a vaneless diffuser and volute equivalent geometry could be defined in terms of LWR, AR and 20, it would be possible to compare the experimentally-determined losses relative to straight-walled diffuser data provided by Runstadler[60]. This type of validation has been used to good effect in the past[54-61], particularly in vaned diffuser analysis. It was speculated that it could provide a simple way to determine the performance of the vaneless diffuser and volute. The experimental data was used as the inlet conditions and pressure recovery was calculated to allow for comparisons with the Runstadler data.
8.5.1.1 Diffuser Equivalent Geometry

The geometry of the straight-walled diffuser as defined by Runstadler\cite{runstadler}, is shown in figure 2-3. The main geometric features of all vaneless diffusers include the pinch and exit radii and the width, shown in figure 1-17. Consideration has been given to ensure that the geometric features chosen could be applied to any radial vaneless diffuser. The following equations were used to define an equivalent set of geometry:

- The length to width ratio $LWR_{eqv}$ was defined as the difference between the vaneless diffuser pinch and exit diameters $l = (D_4 - D_3)/2$ to the vaneless diffuser width, $b_3$:
  \[ LWR_{eqv} = l/b_3 \]  
  eqn. (8-5)

- The area ratio was defined as the inlet area to outlet area where $A = \pi D_b$:
  \[ AR_{eqv} = A_4/A_3 \]  
  eqn. (8-6)

- An expression for an equivalent divergence angle of the vaneless diffuser, as derived by Runstadler, was chosen as:
  \[ 2\theta_{eqv} = 2\tan^{-1}\left[ (AR_{eqv} - 1)/(2(LWR_{eqv})) \right] \]  
  eqn. (8-7)

The equivalent geometries for both compressors are shown in table 8-2. By comparing these data with figure 2-3 the two vaneless diffusers were found to lay in the flow region of 'no appreciable stall'.

<table>
<thead>
<tr>
<th>Compressor number</th>
<th>Compressor 1</th>
<th>Compressor 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>$WDR_{eqv}$</td>
<td>0.0519</td>
<td>0.0432</td>
</tr>
<tr>
<td>$LDR_{eqv}$</td>
<td>0.3218</td>
<td>0.2180</td>
</tr>
<tr>
<td>$LWR_{eqv}$</td>
<td>6.1955</td>
<td>5.047</td>
</tr>
<tr>
<td>$AR_{eqv}$</td>
<td>1.436</td>
<td>1.644</td>
</tr>
<tr>
<td>$2\theta_{eqv}$</td>
<td>4.03°</td>
<td>7.30°</td>
</tr>
</tbody>
</table>

Table 8-2

Since the $AR_{eqv}$ was constant for each compressor it was not possible to plot a pressure recovery contour map. However, using the equivalent straight-walled diffuser geometry from table 8-2 and the contour map shown in figure 2-5 it was possible to estimate the range of possible recovery coefficients. The averaged recovery ratio, $C_{prave}$ for compressor 1
0.52±0.039 and for compressor 2 was 0.37±0.028. Using the map and geometry compressor 1 fell in the range $C_{prave} = 0.5$ to 0.55 and compressor 2 fell in the range $C_{prave} = 0.4$ to 0.5.

It should be pointed out however, that the two compressors presented here have extremely low aspect ratios (AS) compared to that of the straight-walled diffuser. At such low AS, the affects of boundary layer growth and Reynolds number become significant factors in the performance of the diffuser causing a reduction in pressure recovery $^{[31]}$. The Runstadler data for the straight-walled diffuser supports that determined from the equivalent vaneless diffusers however, it would be inappropriate to suggest this would support all vaneless diffusers. Further study would be required and is outside the remit of this project.

### 8.5.1.2 Comparison of CFD & Experimental Blockage

Blockage and Mach number were however extracted from the experimental data; the blockage was determined from equation (I-9) in appendix I, where the effective area was found from $\frac{m}{\rho_3 V_r3}$ and the geometric area was calculated from $\pi D_3 b_3$.

The Mach number was calculated using the velocity $V_r3$ and the temperature $T_3$. Figure 8-7 shows the blockage variation with Mach number for CFD and experimental data at two
speeds. The experimental data varies considerably over the small Mach number range, however, direct comparison of these data with a corresponding Runstadler plot was not possible because the vaneless diffuser produced a higher value of blockage than the straight-walled diffuser for similar values of Mach number. Figure 8-7 shows that for very low aspect ratios a small change in velocity can result in a significantly large change in blockage. The CAPRICE prediction of blockage for compressor 2 produced a range of blockage values between 0.9 and 0.94. Although the experimental blockage is over a small range, the shape of the data (falling blockage with increasing Mach number) shows good agreement with the Runstadler data.

8.5.1.3 Volute Equivalent Geometry

It was surmised that the same technique used on the vaneless diffuser could be applied to the volute casing. The following equations were used to define an equivalent set of geometry:

- The length to width ratio was defined as the total length of the scroll from \{4\} to \{7\}, \( L_{\text{tot}} \) as defined in 6.4.1 to the equivalent vaneless diffuser diameter \( D_{\text{4eqv}} = 2\sqrt{(A_4/\pi)} \):

\[
\text{LWR}_{\text{eqv}} = \frac{L_{\text{tot}}}{D_{\text{4eqv}}} \quad \text{eqn. (8-8)}
\]

- The area ratio was defined as the inlet area to outlet:

\[
\text{AR}_{\text{eqv}} = \frac{A_7}{A_4} \quad \text{eqn. (8-9)}
\]

- An expression for an equivalent divergence angle of the vaneless diffuser, as derived by Runstadler, was chosen as:

\[
\theta_{\text{eqv}} = 2.\tan^{-1}\left[\frac{(\text{AR}_{\text{eqv}} - 1)/(2(\text{LWR}_{\text{eqv}}))}{(\text{AR}_{\text{eqv}} - 1)/(2(\text{LWR}_{\text{eqv}}))}\right] \quad \text{eqn. (8-10)}
\]

The equivalent geometries for both compressors are shown in table 8-3.

<table>
<thead>
<tr>
<th>Compressor number</th>
<th>Compressor 1</th>
<th>Compressor 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \theta_{\text{eqv}} )</td>
<td>0.643</td>
<td>0.15</td>
</tr>
<tr>
<td>( \text{LWR}_{\text{eqv}} )</td>
<td>11.275</td>
<td>11.854</td>
</tr>
<tr>
<td>( \text{AR}_{\text{eqv}} )</td>
<td>1.126</td>
<td>1.031</td>
</tr>
</tbody>
</table>

Table 8-3
Since the divergence angles were so small it is not possible to compare the experimental data with the Runstadler data. However, the volute $C_{pr}$ for compressors 1 and 2. The $C_{prave}$ for compressor 1 was $0.24\pm0.15$ and for compressor 2 was $0.26\pm0.12$. These values are consistent with the expected pressure rise in the volute, being relatively smaller than that in the diffuser.

8.5.1.4 Summary
By determining the equivalent geometry of the two vaneless diffusers a comparison has been made with data provided by Runstadler, for similar shaped straight-walled diffusers. It has been shown that it is possible to use this existing data to determine whether the losses in vaneless diffuser and volute are within acceptable bounds given a specific set of geometry. Whilst this method does not provide a range of values from which to produce a contour map, it does provide a quick method of determining whether the two components are operating within an appropriate region.

8.5.2 Simple Pipe Loss Models

8.5.2.1 Pipe Loss of the Vaneless Diffuser
The pipe diameter was defined as the mean diameter between stations {3} and {4} and the diffuser inlet and exit areas were determined from:

$$A_3 = \pi D_3 b_3 \text{ and } A_4 = \pi D_4 b_3$$

The average pipe diameter was:

$$D_{ave} = 2 \sqrt{\frac{(A_3 + A_4)}{2\pi}}$$

The equivalent length of diffuser, from {3} to {4} was:

$$L_{diff} = \frac{D_4}{2} - \frac{D_3}{2}$$

The Reynolds number was calculated using the diameter $D_{ave}$ and the experimental absolute velocity and density, $V_3$ and $\rho_3$. The surface roughness coefficient was assumed constant at 0.15mm and the friction factor was determined from equation (8-2). The head lost to friction was then determined from equation (8-1).
The pipe loss analysis of the vaneless diffuser for compressors 1 and 2 was done as shown in Figure 8-8 and 8-9. The loss data was obtained from the fuel consumption and the handling of the diffuser. The data was presented in the form of a graph showing the stagnation pressure loss coefficient on the y-axis and the angle of attack (alpha) on the x-axis. The data was divided into two models: one for each compressor. The red squares represent the experimental data, and the blue diamonds represent the pipe loss model.

Figure 8-8 shows the stagnation pressure loss coefficient for compressor 2's vaneless diffuser pipe loss model. The data points are scattered across the graph, indicating a range of values for different angles of attack.

Figure 8-9 shows the stagnation pressure loss coefficient for compressor 1's vaneless diffuser pipe loss model. The data points are also scattered, but with a different distribution compared to Figure 8-8.

Page 240
The pipe loss analysis of the vaneless diffuser for compressors 1 and 2 did not produce favourable results as shown in figures 8-8 and 8-9. The $\zeta_d$ in both cases barely rose above 0.013 across the whole range. This was due to the assumption of mean diameter and constant velocity and density through the pipe.

### 8.5.2.2 Pipe Loss of the Volute

The pipe diameter was defined as the mean diameter between stations {4} and {7} and the diffuser exit area and the exit diffuser area were determined from:

$$A_4 = \pi D_4 b_3 \quad \text{and} \quad A_7 = \pi (D_7/2)^2$$

The average pipe diameter was:

$$D_{\text{ave}} = 2 \sqrt[2]{\frac{A_4 + A_7}{2\pi}}$$

The equivalent length of scroll, from {4} to {6} was:

$$L_{\text{sc}} = 2\pi pc$$

Where the mean perimeter, $pc = ((R_4+R_6)/2)$ was found from $R_6 = (A_6/(A/R))$ and $R_4 = D_4/2$.

The total length of pipe $L_{\text{tot}}$, was the equivalent scroll length plus the exit diffuser length. The Reynolds number was calculated using diameter was $D_{\text{ave}}$ and the absolute velocity and density $V_4$ and $\rho_4$. The surface roughness coefficient, friction factor and head lost to friction were then determined in the same way as the diffuser pipe model. Again, the pipe loss model of the volute for compressors 1 and 2 produced poor results as shown in figure 8-10. The $\zeta_v$ in both cases did not rise above 0.32 across the whole range.
It is clear from the results that the use of a simple pipe model to determine the loss in either the diffuser or volute is wholly unrealistic and too over-simplified even for a 1-D performance prediction technique.

8.5.2.3 Volute Pipe Model With Helical Flow

It is known that the flow entering the volute is spiralling as shown in figure 1-22. It was hypothesised that the flow path length would, due to swirling flow be longer than that assumed in the straight pipe case. This would result in an increase in friction in the pipe and therefore an increase the losses. Based on this assumption a helical flow path model was devised. The helix was chosen because of its straightforward geometry which it was believed would provide a simple representation of the flow in the volute. Since the inlet angle and the absolute velocity from the diffuser can be determined, and again assuming a mean diameter pipe, the helical flow path can be determined from.

\[
\text{hypotenuse} = \frac{\pi.D_{\text{ave}}}{\cos \alpha_4}
\]

\[
\text{Pitch} = \pi.D_{\text{ave}}.\tan(\alpha_4)
\]
Number of pitches = \( \frac{L_{\text{tot}}}{\text{pitch}} \)

Length of helical flow path = number of pitches x hypotenuse

The remainder of the calculation was carried out as in section 8.5.2 where the Reynolds number was calculated using diameter \( D_{\text{ave}} \) and the experimental absolute velocity and density, \( V_4 \) and \( \rho_4 \). The friction factor was determined from equation (8-2) and the head lost to friction was then determined from equation (8-1).

Figure 8-11 shows the experimental and predicted stagnation pressure loss coefficient, \( \zeta_v \) between \{4\} and \{7\} for compressor 2. \( \zeta_v \) is highest at the choke end of the flow (\( \alpha_4 = 50^\circ \) to \( 63^\circ \)) which was expected since at choke the inlet angle \( \alpha_4 \) would be small and the number of pitches would be high, resulting in a long path length and a higher loss due to friction. However, as the flow nears surge (\( V_{\text{w4}}/V_{\text{rd}} = 63^\circ \) to \( 75^\circ \)) \( \zeta_v \) becomes almost constant at around 0.33. Whilst there is a slight improvement over the straight pipe model, the helical path model still does not adequately predict the \( \zeta_v \). The averaged difference between the experimental and predicted loss coefficients was 0.15, with the stagnation pressure predicted using the helical model being approximately 1.6% greater than that determined from the experimental.

Figure 8-11 compressor 2: volute experimental stagnation pressure loss coefficient and helical flow pipe loss prediction
8.5.2.4 Conclusion
The simple pipe and helical path loss models present in section 8.5.2 have been shown to be too simplistic in their approach and are unsuitable for predicting the loss coefficients in the diffuser and volute. Their assumptions of simple geometry and linear flows do not adequately describe the complex shape and highly swirling flows associated with the diffusion system components. It was therefore necessary to find a method which would enable the prediction, not only of the losses, but of the changing shape of the loss curves as well.

8.5.3 Rogers Vaneless Diffuser Loss Model
Rogers\textsuperscript{[28]} presented a technique for determining the static pressure recovery of a radial vaneless diffuser with a backswept impeller. He produced a correlation between recovery coefficient and diffuser geometry over a range of gas angles; where the skin friction factor was determined from an empirical relationship $C_f = 0.046\Re^{-0.2}$ and this is presented in equation (1-20).

The vaneless diffuser experimental data is presented in terms of a static pressure recovery coefficient $C_{pr}$, against inlet gas angle $\alpha_3$, shown in figure 8-12 as red and green squares along side the Rogers prediction of $C_{pr}$ determined from equation (1-20). A relationship between $C_{pr}$, DR and $\alpha_3$ was found by Watson et al\textsuperscript{[124]}, who demonstrated that $C_{pr}$ increased with falling velocity ratio, $V_w/V_r$ (falling gas angle), for a fixed DR and constant skin friction factor. It can be seen in figure 8-12 that there is significant difference in $C_{pr}$ between the two compressors, which is a function of their geometries.

The Rogers prediction and experimental data correlated reasonably well at the higher gas angles between 65° and 70° (surge), but compressor 2 showed a particularly poor correlation below 63°. This was believed to be due to the larger than optimum diffuser width. Using equation (1-20) – and with all other geometries held constant – the diffuser width was adjusted until a suitable match with the experimental data was found. $b_3$ was found to be 29% smaller than that of the tested diffuser, for this new curve-fit which is shown in figure 8-12 as a hashed line.
From equation (1-20) it can be seen that the relationship between $C_p$ and $\zeta_d$ is determined by the diameter ratio, DR. By rearranging this equation a relationship for loss coefficient can be found based on the geometric features of the diffuser and the inlet gas angle. Equation (1-21) was used to determine $\zeta_d$ and was correlated with $\alpha_3$, shown in figure 8-13.
Again, with the original geometry, the compressor 2 correlation is poor at low gas angles. Since, for a specified set of geometry, $\zeta_d$ is proportional to $\frac{1}{\cos \alpha_3}$, the experimental and predicted loss coefficients were correlated against $\frac{1}{\cos \alpha_3}$ (shown in figure 8-14) to determine a linear relationship between gas angle and loss coefficient, where the geometry of each compressor determines the gradient of the straight line.

![Graph showing linear relationship between diffuser loss coefficient and inlet gas angle for varying diffuser geometry](image)

The Rogers correlation provided a simple technique for the prediction of diffuser performance and was a significant improvement on the pipe loss models previously presented.
8.5.4 Vaneless Diffuser Empirical Model

Whilst a simple model such as the Roger's technique can provide an adequate prediction in terms of the spread of data, it provides no clear delineation of the losses from one speed line to the next. It was speculated that an empirical analysis of the experimental data would yield a model which would not only predict the level of loss but also the shape of the loss curve at varying speed. The following section demonstrates how this correlation was developed and establishes that whilst it shows promise, the limited and scattered experimental data resulted in poor correlations between the experimental and predicted curves for some of the speeds.

Figure 8-15 shows stagnation pressure loss coefficients $\zeta_d$ for varying gas angles at diffuser pinch $\alpha_3$, for four speed lines, two from each compressor. Also shown are the curve-fits of each set of data. This figure illustrates how the shape and spread of the data varies between speed lines and it is this changing shape that required predicting.

![Figure 8-15](image.png)

Figure 8-15 experimental stagnation pressure loss coefficients and corresponding best fit curves

It is clear that the performance of the diffuser is affected by the diffuser geometry and therefore it is necessary to correlate $\zeta_d$ with one or more of the diffuser's geometric..
parameters such as pinch or exit radii and/or width. However, correlating with one or more geometric features makes it difficult to compare different compressors directly, this can be overcome by selecting a non-dimensional geometric parameter.

It has been shown in section 8.4.1. that the flow path of the air through the diffuser varies with inlet gas angle and that the path length also dictates the value of diffuser loss coefficient. Furthermore, the diffuser width has also been shown to play a part in determining the loss coefficient value. The Length to Width ratio (LWR) was therefore selected as a generic feature of the diffuser with which a correlation could be made. The following section presents a technique for predicting the diffuser performance using loss coefficient $\zeta_d$ and LWR.

The steps describing the technique are as follows:

- Apply equation (8-4) to determine the log spiral flow path (for each mass flow rate at all speeds), from which the LWR can be found.

- Correlate the experimental loss coefficient against LWR and establish a set of quadratic algebraic expressions, (using the method of least squares) from the curve-fits of these data, for all specific values of rotational speed.

- Using these expressions, determine the loss coefficient at any speed.

8.5.4.1 Correlation of Experimental Data with Geometry and Shape Function

The log spiral length to width ratio (LWR$\text{spiral}$) can then be described as the 'length of the logarithmic flow path to the width of the diffuser passage':

$$LWR_{\text{spiral}} = \frac{s}{b}$$

eqn. (8-11)

If $WDR = \frac{b_3}{D_3}$ then

$$LWR_{\text{spiral}} = \frac{s}{b_3} = \frac{s}{D_3} \cdot \frac{D_3}{b_3}$$
Applying equation (8-4)

\[ LWR_{spiral} = \frac{\cos ec\theta_3 \cdot (DR - 1)}{2 \cdot WDR} \quad \text{eqn. (8-12)} \]

Where:

\[ \theta = \sin^{-1}\left( \frac{(DR - 1)}{2 \cdot LWR_{spiral} \cdot WDR} \right) \]

\[ \theta_{eqv} = 2 \cdot \tan^{-1}\left[ \frac{(AR - 1)}{(2 \cdot LWR_{spiral})} \right] \quad \text{eqn. (8-13)} \]

Equation (8-12) was used to determine the experimental LWR over the range of experimental gas angles for both compressors and figure 8-16 shows the relationship between LWR and \( \zeta_d \).
8.5.4.2 Determining the Speed Function

Figure 8-17 shows the experimentally-determined stagnation pressure loss coefficient with varying LWR for four speed-lines (two from each compressor) and their corresponding curve-fits.

The coefficients for each curve-fit ($x^2$, $x$ and $c$) where plotted against blade speed ($U_2/\sqrt{T_{01}}$). The $x^2$ coefficient is shown in figure 8-18 as a straight-line fit (using a method of least squares) and is expressed in algebraic form as:

$$x^2 = 0.0000479118 \left(\frac{U_2}{\sqrt{T_{01}}}\right) - 0.0004008863$$
From which the stagnation pressure loss coefficient $\zeta_d$ can be determined from the expression:

$$\text{loss} = x^2 (\text{LWR})^2 + x (\text{LWR}) + c \quad \text{eqn. (8-14)}$$

With the limited number and scattering of the stagnation pressure loss coefficient data points shown in figure 8-18, it was clear that the $x^2$, $x$ and $c$ curve-fit coefficients would also be scattered and that an accurate correlation would be difficult to achieve. The averaged error of the $x^2$ coefficients was approximately ±0.00015.

Using these coefficient functions a predicted $\zeta_d$ against LWR was compared with the experimentally-determined data and four of the speed lines are presented in figure 8-19.
It was anticipated that since the predicted $\zeta_{td}$ curves were based on best-fit curves of experimental data, and straight-line fits of coefficients, the accuracy of the prediction would be poor. The averaged error across the range of curves was found to be +/- 0.04, with $U_2/\sqrt{T_{01}} = 24$ of compressor 2 showing the best correlation, with an error of +/- 0.01.

8.5.4.3 Conclusion

This empirical model showed some success at predicting the shape and range of the loss coefficients for each speed but due to the scatter of the coefficient data several of the predictions were very poor. It is believed that for this technique to be successful a large amount of experimental data would be required.
8.5.5 The Van Den Braembussche Volute Model

It was proposed that the 1-D prediction technique for off-design conditions within the volute scroll, described by Van Den Braembussche et al\[^{83}\] (VDB) in section 2.3.2, provided a suitable model of the prediction of the volute loss characteristic.

The volutes were divided up into seven segments as described by VDB, which corresponded to the spaces between the static pressure tapings around the volute. The cross-sectional areas and radii to the centre of each area, at each tapping location, were provided by GEBS and the model was constructed as defined in VDB\[^{83}\]. A full description of the model is provided in appendix I section I-viii.

Firstly, several assumptions had to be made.

- The diffuser exit conditions were used as inlet conditions for each segment, i.e. a constant inlet velocity and density was assumed around the circumference of the diffuser.
- The calculated inviscid stagnation pressure was assumed to be the diffuser exit stagnation pressure.
- The experimentally-determined static pressure around the volute was assumed to be the wall static pressure at the outlet of each section.

Two programs were written; the first one to calculate the core flow conditions using the experimental static pressures, which represented the wall static pressures. The second to predict the flow at the wall and the core, as defined in VDB\[^{83}\]. Using these two sets of data the predicted wall and core flow conditions could be compared with the experimental data.

Since the VDB model was only concerned with the scroll section of the volute (stations \{4\} to \{6\}) the exit diffuser was ignored for this analysis.

The experimental and predicted loss from both compressors are shown in the figures 8-20 and 8-21. The VDB model shows some similarity to the experimental $\zeta_v$ however, at the extremes of flow, the $\zeta_v$ predictions are poor, notably at choke where the predicted $\zeta_v$ is shown as being much greater than the experimental $\zeta_v$. 
Figure 8-20 Experimental and VDB predicted loss ratio for compressor 2

Figure 8-21 Experimental and VDB predicted loss ratio for compressor 1
8.5.6 Volute Empirical Model

A similar technique to that used on the diffuser loss correlation was applied to the experimentally-determined volute stagnation pressure loss coefficient data, $\zeta_v$. Figure 8-22 shows $\zeta_v$ for varying inlet gas angles $\alpha_4$, for both compressors. In general, the values are very similar, ranging between 0.4 – 0.7. The spread of the loss data is also very similar, being parabolic in nature and it was considered that a good empirical correlation of the volute data could be made to determine both the range and shape of the data for varying speeds.

![Figure 8-22 experimental volute loss ratio for compressors 1 and 2](image)

For clarity, four speeds (two from each compressor) are shown in figure 8-23. The figure shows that at surge the curves share a common range of gas angles and loss coefficient values. However, as the curves move past the minimum loss coefficient and begin to rise towards choke, the curves separate, varying with inlet gas angle over a similar range of $\zeta_v$; compressor 1 producing lower values of loss than compressor 2. Each of these curves can be predicted using the same method as demonstrated in section 8.5.6.
A series of quadratic expressions were determined from figure 8-23, again using a least squares method, at each speed line. An equation for each of the coefficients against blade speed was produced and figure 8-24 shows the \( x^2 \) function (solid blue squares) which was expressed as:

\[
x^2 = 0.0000100582 \left( \frac{U_2}{\sqrt{T_{01}}} \right)^2 - 0.00004053944 \left( \frac{U_2}{\sqrt{T_{01}}} \right) + 0.00085035754
\]
Three of the coefficients (hollow blue squares) were not used in this correlation as they were approximately 40% in error of the rest of the data points. This could be criticised as it would inevitably lead to poor prediction of the losses at these three speeds. It can be argued however that the majority of data points provided a good correlation and that with such limited data it must be accepted that some points would not agree with the majority. Using these coefficient functions, $\zeta_v$ could be determined with varying exit gas angle at any speed from the equation:

$$\text{loss} = x^2(\alpha_4)^2 + x(\alpha_4) + c$$
eqn (8-15)

Figure 8-25 shows the predicted and experimental $\zeta_v$ for four speeds. The shapes are well predicted for three of the four sets of data, with averaged errors of 1.5%. As was expected from the correlation presented in figure 8-24 the predicted loss for $U_2/\sqrt{T_{01}} = 20$ was poor, particularly at the choke end of the curve. The predicted curve was shifted to the left, resulting in much lower values of exit angle for the same loss coefficient values (a 35% difference in the worst case).

![Figure 8-25](image_url)
8.5.6.1 **Sub-dividing the Volute**

Using the data for the volute presented in section 8.3.3., correlations between stations {4} to {6} and {6} to {7} were made against diffuser exit gas angle $\alpha_4$ and are presented in figures 8-26 and 8-27.

![Graph showing stagnation pressure loss coefficient between station {4} and {6}](image)

**Figure 8-26** Volute stagnation pressure loss coefficient between station {4} and {6}

$$\text{Loss}_{4,6} = 0.00009476(\alpha_4)^2 - 0.0009736(\alpha_4) + 0.142797 \quad \text{eqn. (8-16)}$$

Whilst the loss coefficient data of compressor 2 was spread over a narrower range of gas angles than compressor 1 it was believed that a single parabolic curve fit could be applied to both sets of data to produce a simple prediction for stagnation pressure loss coefficient between the diffuser exit and volute tongue. Since the majority of experimental data points fell well within ± 0.1 of the curve-fit, this was deemed acceptable within the bounds of the limited data.

The same approach was applied to the stagnation pressure loss coefficient data between stations {6} and {7}, shown in figure 8-27. Again compressor 2's data is spread over a much smaller range of gas angles than compressor 1, but in essence both sets of data produced a linear relationship.
It could be argued that a single curve-fit through these data does not adequately describe the differences between the two sets of experimental data, and that it is clear that variations in stagnation pressure loss coefficient exist which are likely caused by upstream flow conditions as well as the volute geometries. With such limited data as these, it was deemed the best possible method of determining the volute loss. The difference between the two sets of experimental data in figure 8-27 was ±0.1 and between the curve-fit and experimental data the difference was reduced to ±0.07. This was judged to be an acceptable error considering the limited available data from which to make a correlation. These two curve-fits have been incorporated into the CAPRICE program to assess their ability to predict the volute characteristic. It was speculated that a more accurate prediction of loss would be produced using the two correlations from figures 8-26 and 8-27 than that produced from figure 8-25.

8.5.6.2 Conclusion
Two techniques for modelling the stagnation pressure loss coefficient between diffuser exit and volute exit have been presented. It was found that both the VDB and empirical methods produced only limited success in predicting the volute characteristic. This was primarily due to the nature of the experimental data and the compromises that had to be made in the overall experimental and computational procedure. It was found that a more accurate prediction could be obtained by analyzing the overall experimental and computational results and developing a model that takes into account the loss in the overall characteristic curve. This was achieved through several experimental and computational techniques and a better understanding of the flow physics involved.

\[
\text{Loss}_{6-7} = -0.010797(\alpha_4) + 0.766733 \quad \text{eqn. (8-17)}
\]
due to the scattering of the experimental data and the compromises that had to be made in order to produce a suitable prediction. It was found that a more accurate solution was to divide the volute into two sections; the scroll and exit diffuser, and determine the conditions in each of these components from which the overall characteristic could be determined. Whilst this method also produced errors between the overall experimental and predicted stagnation pressure loss coefficient, the inherent error was reduced.

8.6 Summary

This chapter has set out to analyse and determine a method of predicting the losses in the compressor vaneless diffuser and volute as a means of extending the 1-D prediction technique. Various methods of predicting the losses in the diffuser and volute have been investigated and the following findings made:

- The simple pipe loss and helical flow path models produced extremely poor correlations with the experimental data and did not truly reflect the conditions in the two components.
- The comparison with Runstadler data showed that the diffuser and volute could be described using an equivalent geometry and that existing pressure recovery contour maps could be used to give an initial idea of the losses in the two components.
- The Rogers model produced good results and provided a method for determining a more ‘optimised’ diffuser width.
- The VDB models produced reasonable correlations with the experimental data, but did not fully produce an adequate representation of the changes in shape of the loss curves with variation of speed.
- Two empirical correlations were produced for the vaneless diffuser and volute, with the intention of not only determining the loss, but also determining the shape of the loss curves at varying speeds. The expression presented only required the impeller exit gas angle and the diffuser width and inlet and exit radii to find the loss coefficient. Some improvement were achieved but several loss curves were poorly predicted due to the scattering of coefficient values.
Overall the two most suitable techniques found to produce the best correlations were the Roger's recovery model for the vaneless diffuser and the empirical model for the subdivided volute. These models, have been incorporated into the CAPRICE program, and the results of their predictions along with the overall accuracy of the new program are presented in chapter nine.
9 Assessing the Accuracy of the New Correlations and CAPRICE3 – The Improved Prediction Technique

Synopsis

The correlations presented in chapters seven and eight have been incorporated into the CAPRICE prediction program. This chapter presents the improved prediction technique CAPRICE3 and assesses the accuracy of the newly developed models. A comparison of the CAPRICE3-predicted compressor characteristics & the original prediction technique is presented in the form of compressor maps.
9.1 Introduction

The correlations produced from the experimental data in chapters seven and eight have been incorporated into a new version of the CAPRICE program called CAPRICE3. A comparison of the CAPRICE3-predicted and the actual compressor maps has been made along with comparisons of the impeller, vaneless diffuser and volute component characteristics.

The following sections present the final program structure in the form of a flow chart and discuss the accuracy of the new models, their deficiencies and advantages. The experimental and predicted compressor maps of the two tested compressors tested also presented and it is shown that the data and prediction were in good agreement, despite the limitation of the available data and accuracy of some of the empirical correlations.

9.2 CAPRICE3 Program Structure

There were six main correlations produced from the experimental data which were incorporated into the component models of the CAPRICE program, these were:

- Impeller Work correlation
- Impeller efficiency ratio
- Impeller Surge correlations at inlet and exit
- Diffuser stagnation pressure loss coefficient using a log spiral path
- Volute stagnation pressure loss coefficient using the diffuser exit gas angle
- Diffuser and Volute Surge predictions at pinch and tongue respectively

The flow chart figure 9-1, shows how these new correlations (in red boxes) have been incorporated into the CAPRICE program.
CAPRICE3 – The Improved Performance Prediction Technique

Start

Read geometric data

Calculate initial gas properties

Calculate inducer and throat parameters

Read in speeds

Calculate inducer choke flow

Calculate inducer entry conditions

Adjust \( \alpha \) as defined in eqns. (7-6) & (7-7)

Compare \( \alpha \) to \( \alpha_{2s} \)

Adjust \( m \) as defined in eqns. (7-8) & (7-9)

Compare \( m_i \) & \( m_o \) select the larger of the two values

Calculate range between choke & surge

Calculate diffuser conditions using diffuser loss coefficient equation

Compare \( \alpha \) to \( \alpha_{3s} \)

Adjust \( m \) as defined in eqns. (7-10) & (7-11)

Compare \( m_o \) with \( m_{mp} \) & select the larger of the two values

Calculate overall characteristics

Calculate Volute conditions using volute loss coefficient equations

Output data

Figure 9-1 CAPRICE3 flow chart of operation

Page 264
The program functions in the following way:

- The gas properties and choke flow are calculated based on the geometric parameters and speed input by the user. These particular models are unchanged from the original.

- The inducer flow conditions are calculated

- The inducer mass flow at surge is determined by adjusting the mass flow rate until the inducer incidence value matches the inducer incidence value at surge, based on equations (7-6) and (7-7).

- The impeller flow conditions are calculated and the impeller exit angle at surge is determined by adjusting the mass flow rate until the exit angle value matches the exit angle at surge, using equations (7-8) and (7-9).

- The mass flow rates are compared and the larger of the two is selected to determine the diffuser conditions.

- The diffuser pinch flow conditions are determined.

- The surge in the diffuser pinch is determined by adjusting the mass flow rate until the gas angle matches the surge gas angle found from equations (7-10) and (7-11).

- A comparison between the impeller and diffuser mass flow rates is made and the larger of the two is selected as the surge value.

- The diffuser exit conditions are determined.

- The overall characteristic is divided into 21 points, between the surge and choke values and determines the flow conditions at each point for the impeller, diffuser and volute. The impeller characteristic is determined from equations (7-2) to (7-4). The diffuser characteristics is determined by applying equations (1-37). The volute characteristics are determined using equations (8-16) and (8-17) from which the exit conditions are found.

- When the choke mass flow rate is achieved the data is output to a file and the program moves on to the next speed or ends.

The overall compressor maps produced by CAPRICE3 are shown in figures 9-10 and 9-11 and are overlaid with the maps produced from the experimental data as a comparison.
9.3 Assessing The Correlations

Chapters seven and eight presented an analysis of the experimentally-determined flow conditions in the three components of the centrifugal compressor. From this it was shown that correlations could be produced using these data, from which predictions of the performance could be made. Section 9.2 has introduced the general structure of the new CAPRICE3 program and the following section compares the results of the predictions made for compressors 1 and 2. Also, the accuracy of the models and their ability to predict the characteristics of each component is presented and the flexibility of the CAPRICE3 program to overcome some of the inherent errors associated with empirical predictions is discussed.

### 9.3.1 Accuracy of the Impeller Work Correlation

The experimental data curve-fits were incorporated into the empirical equation and a new equation for impeller work was presented in section 7.2.3.2, equation (7-5). The CAPRICE3-predicted work for both these compressors compared well with the experimental data and the error between predicted and experimental work was ±0.045.

![Figure 9-2: CAPRICE3 prediction and experimental Impeller Work](image-url)
9.3.2 Accuracy of the Impeller Efficiency Ratio Model

Initially an improvement to the impeller model was not thought to be necessary since the model had been used for several years to good effect. However, it was shown in figures 6-13 and 6-14 that the existing model poorly predicted the shape of the impeller characteristic and that some improvement to the prediction was required.

A correlation of the experimental impeller efficiency ratio \( \eta_{\text{imp}}/\eta_{\text{imp peak}} \) and mass flow rate ratio \( m/m_{\text{max}} \) with varying speed was presented in figure 7-4, which showed that the scattering of the two sets of data were similar, with some slight variations at surge. Using these data a set of equations was produced based on those derived by Swain\(^9\).

The correlation presented in figure 7-5 showed good agreement between the experimental and predicted efficiency ratios and the model was incorporated into the CAPRICE3 code. A prediction of the two compressors using the program, is presented in figure 9-3. Whilst there is some variation between the mass flow functions, the efficiency values correlated very well. The shape of the predicted curves is much improved from the original and more accurately represents the experimental impeller efficiency.

![Figure 9-3: Experimental and CAPRICE3 predicted Impeller efficiency ratio for both compressors.](image-url)
9.3.3 Accuracy of the Diffuser Loss Coefficient and Log Spiral Model

The scattering of the experimental data points in figure 8-16 - especially those determined from compressor 2 - were the main obstruction in producing an accurate prediction for the diffuser loss coefficient model. Whilst compressor 1’s data was closely matched in both loss coefficient and LWR over the range of speeds, compressor 2 produced a significant variation in loss, even though the range of LWR’s was similar to compressor 1. This resulted in a compromise between the two sets of data, resulting in higher values of loss in compressor 1 and lower values in compressor 2.

The flow path length was derived from an assumption that the gas angle from diffuser inlet to exit was constant. This assumption does not describe the ‘actual’ particle path, but provides a simple model from which to derive the flow path length and was certainly an improvement on the existing model which assumed a straight-line flow path. Typically the velocity ratio of the diffuser should not exceed 4, which is equivalent to an inlet gas angle, $\alpha_3$ of $76^\circ$ (or $\gamma = 14^\circ$). Beyond this, the length - determined from equation (8-4), and the LWR equation (8-12) - tends to increase exponentially for small changes in angle. At surge, where the flow path is longest, a small decrease in gas angle results in a proportionally larger increase in flow path length than that for the same angle change at choke. However, the prediction of LWR for both compressors, across all the speeds tested, was not adversely effected by this assumption as shown in figure 8-19.

9.3.4 Accuracy of the Diffuser Loss Coefficient Sins Rogers Technique

It was shown in figure 8-12 that the technique using the Rogers equation (1-21) correlated very well with the experimental data. The predicted and experimental data for compressors 1 and 2 is shown in figure 9-4. Compressor 1’s prediction gave an error no greater than $\pm 0.02$, however compressor 2 gave an error of approximately $\pm 0.03$, with one speed line being in error of $\pm 0.05$. overall this diffuser characteristic proved very successful.
9.3.5 Accuracy of the Volute Loss Coefficient Correlation

A comparison of the losses predicted in the scroll and exit diffuser equations (8-16) and (8-17) are presented in figures 9-5 and 9-6. It was found that at surge the two compressor’s data matched closely, however at choke they showed marked differences. In general, and considering the scatter of the experimental data the prediction shows good agreement with the experimental values. With the majority of data points being in error by only ±0.05 for the loss coefficient between {6} and {7}. However the error between the experimental and predicted data between {4} and {6} was much greater; being ±0.08 for compressor 1 but in excess of ±0.2 for compressor 2. This error significantly effected the predicted flow conditions in the compressor and were reflected in the overall compressor map prediction for compressor 2 presented in section 9.5.
Figure 9-5  Experimental and predicted volute loss coefficient in the scroll section (stations {4} to {6})

Figure 9-6  Experimental and predicted volute loss coefficient in the exit diffuser section (stations {6} to {7})
9.3.6 Accuracy of the Surge Correlations

The inducer and impeller exit surge equations (7-6) to (7-9) were shown to produce good predictions against the experimental data. Although the experimental data was scattered, it was found that by splitting the curve-fit equations into two allowed for a more accurate prediction of the incidence and exit angles. The experimental and predicted surge are shown in figure 9-7 and 9-8.

![Graph showing experimental and predicted surge angles for compressors 1 & 2.](image)

**Figure 9-7 Experimental and predicted inducer incidence gas angle for compressors 1 & 2**

The averaged error between the experimental and CAPRICE3 predictions are given in table 9-1.

<table>
<thead>
<tr>
<th>Compressor</th>
<th>Incidence angle</th>
<th>Exit angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compressor 1</td>
<td>-0.73°</td>
<td>-0.68°</td>
</tr>
<tr>
<td>Compressor 2</td>
<td>0.15°</td>
<td>0.56°</td>
</tr>
</tbody>
</table>

**Table 9-1**

The difference between the experimental and predicted data is well within the tolerances of the 1-D prediction. The new prediction for compressor 2 shows some improvement to the that shown in chapter 3, section 3.8.5 and although the prediction was slightly lower than the experimental for compressor 1, due to the nature of the curve-fits, the difference is still very small.
The same technique was applied to the prediction of surge at the diffuser pinch, equations (7-10) and (7-11). Here, however, the scattering of experimental data was even more pronounced than that of the impeller surge prediction, particularly for compressor 2. The difference in the gas angles at the diffuser pinch was thought to be due to the difference in geometry of the two compressors and the wider than optimum diffuser width, but without additional experimental data from diffusers with different geometries, it was not possible to verify this theory. As was expected the surge prediction produced by CAPRICE3 for compressor 2 was poor, the difference being - at the low and high speeds - 6.4° and 9° respectively. The majority of data points were, however within ±2° of the experimental data.

Figure 9-8 Experimental and predicted impeller exit gas angle for compressors 1 & 2
9.3.7 Conclusion

The accuracy of the models discussed above are reliant on two conditions:

- The accuracy of the experimental data
- The accuracy of the correlations

It follows that the correlations and hence the models are only accurate if the experimental data is determined to be accurate. The experimental data itself was shown to be accurate in chapter five and was supported by the CFD data. Inaccuracies in the models have been shown to be due to the issues associated with compressor 2 and its non-standard casing, which caused the compressor to operate below its optimum performance. This led to inaccuracies in the correlations because some of the experimental data was below its expected level. Additionally, the use of simple best-fit assumptions and quadratic expressions resulted in poor predictions particularly at flow range extremes of surge and choke. However, as an initial prediction techniques the agreement is good. A simple relationship between geometry and flow conditions has been established and it is no longer necessary to manipulate the code in order to gain a suitable compressor map.
It should be remembered that the intention of these models was not to predict the experimental data point for point but to provide an impression of the shape and range of the compressor components using only geometric features and operating speeds. This has been achieved and an improvement to the CAPRICE performance prediction technique has been made.

**9.4 CAPRICE3 – The Improved Performance Prediction Technique**

The geometry of the two compressors has been used to predict the performance using the improved prediction technique, CAPRICE3. The compressor characteristic maps are presented in figures 9-10 and 9-11. The experimental compressor maps are shown in blue with a red surge line and the predicted map is shown in green. The new characteristics show some improvement on the original CAPRICE prediction. The shape and range of the predicted maps compares well with the experimental characteristic. Compressor 1 does show some variation between the experimental and predicted ranges particularly at surge, where CAPRICE3 has over-predicted the pressure ratios. This is primarily due to the inaccuracies developed in the models due to severe scattering of some of the experimental data.

![Figure 9-10: Compressor 1 experimental and CAPRICE3 predicted map](image-url)
Figure 9-11: Compressor 2 experimental and CAPRICE3 predicted map.
10 Conclusions & Further Work

Synopsis

This chapter contains the final conclusions drawn from the work presented in this thesis. It highlights points for further work and justifies the research’s contributions to the field of turbomachinery.
10.1 Achievements

The objective of this research project was to improve a 1-D performance prediction technique for automotive turbocharger compressors, paying particular attention to the diffusion system characteristic.

The main objectives were to:

- Understand the behaviour of the flow inside a typical turbocharger compressor and identify key geometric features which relate to performance. This was achieved by investigating previous work carried out in this field and by producing correlations using experimental data from a test rig developed for the purpose of this research.

- Identify areas of needed improvement and, using/adapting existing 1-D techniques, develop new prediction methods. This has been achieved by utilising an existing 1-D performance prediction program called CAPRICE and by identifying areas of weakness within its models in the development of an inverse approach to the models.

- Extract experimental data from typical compressors to formulate new correlations and simple models. This has been achieved by the development of an experimental program and the installation of an interstage rig which is now permanently installed with GEBS.

- Validate the prediction technique against existing techniques. This has been achieved by utilising tools such as CFD and existing performance prediction methods which have shown the correlations produced to be in good agreement with these techniques.

In general terms, an improvement to the original 1-D performance prediction program has been achieved.
10.2 Conclusions

10.2.1 General

Chapter nine presented and discussed the results of the new prediction technique. It was understood from the outset of the work that with such limited experiential data accurate prediction of the two compressors, with differing geometries would be difficult. But, it has been shown that whilst some variations exist between the theoretical and experimental data, particularly for the individual component correlations, the overall prediction has been shown to be a significant improvement on the original CAPRICE code and is robust enough to deal with the variations in the correlations. The following conclusions have been drawn from the work presented in the preceding chapters.

The Extraction of the diffusion system performance from overall compressor maps, using an inverse technique, was proved to be a successful method of isolating the component performances and identifying the problems with the existing code. It was shown in chapter three that simple correlations and modifications to the 1-D code could result in significant improvements to the performance prediction. This analysis illustrates the need for the development of such techniques.

The experimental test programme permitted the interstage conditions from two compressors to be collected. This programme yielded some extremely useful results, which enabled the flow conditions within the compressors to be determined and analysed. The simple construction of the test rig using static pressure tapings combined with the existing test facility proved to be extremely successful over a number of tests and the interstage rig remained installed at the GEBS test facility as a permanent feature. It should be noted however that the determination of stagnation pressure was by calculation only and no measurement of stagnation pressure was made, as the introduction of a total pressure probe would possibly cause blockage and disruption to the flow. Regardless of this, the errors between the experimental and theoretical data where shown to be small.

As a result of the limited space in the compressor a 'no loss' method of determining the static pressure at the impeller tip had to be developed. The experimental static pressure
distribution around the impeller exit was shown to be more variable than the calculated static pressure. This demonstrated that the impeller was more severely affected by downstream pressure changes than the calculated pressure would suggest. The averaged error between the experimental and calculated static pressure was 3.7% for all the data.

The CFD model constructed for the impeller and vaneless diffuser permitted the validation of the experimental data. Numerical data such as densities, mass-averaged static and stagnation temperatures and pressures were extracted from the model at four data points and the flow parameters such as velocity and gas angle were then calculated from the data. The major difficulty in comparing the CFD analysis and experimental data was the difference between the 3-D and 1-D modelling techniques. Slight differences in numerical values such as geometry and boundary conditions contribute to differences in the actual compressor and CFD results. Considering the difficulties in comparing these data in general the comparison of the CFD and experimental data shows good agreement and supports the experimental work.

10.2.2 Impeller

Whilst the main focus of the research was on the vaneless diffuser and volute casing, several impeller models were identified as requiring improvement. The two key modifications were in the prediction of impeller Work and surge and are presented in chapter four. The new Work equation is based on the relationship of velocity ratio and impeller backsweep angle and is fully supports the findings of Dean[23] and Whitfield[24]. It shows that Weisner's prediction of a linear relationship for Work is not appropriate for modern-day backswept impellers. The new relationship between velocity ratio and Work is of the form of a power equation and better describes the data.

Improvements to the prediction of surge were produced using a correlation between the inducer incidence and impeller exit gas angle for a range of rotational speeds. For the compressors tested the surge incidence values varied over a range of 10° to 27° and the exit flow angle values varied between 70° to 85°. As the rotational speed increased it was shown that the stall moved from the inducer to the impeller exit illustrated as the step-
change in the exit angle at a particular speed. This improved surge prediction enable the
typical 'kinked' surge line to be better predicted.

10.2.3 Diffuser

The improvements made to the 1-D performance prediction technique and the additional
contributions made to the understanding of diffuser behaviour have been considerable. It
was shown in the initial analysis of the CAPRICE code, chapter three, that the existing
diffusion system model was extremely poor at predicting, with any accuracy, the
performance of the diffusion section and completely failed to identify the significant
differences between the volute and diffuser components. An initial improvement to the
empirical equation for the diffuser system efficiency ratio was determined from the inverse
approach and was clearly shown to be more accurate at predicting the diffusion system
characteristic.

Further analysis of the individual components of the diffusion system (using the
experimental data) yielded several new correlations for their performance prediction. In the
case of the diffuser an analysis of the diffuser loss characteristic identified the minimum
loss to be approximately equal to a velocity ratio of 1.7 ($\alpha_3 = 59.5^\circ$) for both compressors.
Compared to Dean et al [78] who found that for minimum loss an optimum velocity ratio = 2
($\alpha_3 = 63.44^\circ$) this confirmed that the diffusers of the two compressors were well within the
operating parameters for typical diffusers of this type.

Considerable time was taken to identify a simple correlation technique, which would link
the component's geometry to the flow parameters associated with it. A series of techniques,
using both existing methods and newly developed ones, were tested against the
experimental data. A straight-walled diffuser model was used to determine the losses in the
diffuser using an equivalent geometry in terms of LWR, AR and 20 and the relationship
between $C_{pr}$, diameter ratio, DR and $\alpha_3$ was determined. It was shown that the vaneless
diffuser $C_{pr}$ values were just inside the band of the straight-walled diffuser of the same
equivalent geometry. It was also shown that for the purposes of an 'initial guess' the use of
Runstadler straight-walled diffuser maps would provide the designer with a quick method of identifying the area of performance of his design.

A correlation using a technique developed by Rogers was shown to be the most suitable method of predicting the diffuser performance. The experimental data correlated reasonably well with the predicted Rogers curve at the higher gas angles, but below 50° the experimental data correlated poorly. Compressor 2 showed a particularly poor correlation with the Rogers pressure recovery suggesting that factors other than skin friction affect the energy dissipation in the diffuser. The correlation between Rogers and compressor 1 however, showed an extremely close relationship suggesting that skin friction was the predominant loss mechanism. The optimum velocity ratio of the prediction was around 1-1.2 (gas angle of 45°-50°) which was however significantly lower than the experimental values of 1.7 for the corresponding speed lines.

In contrast, an empirical model based solely on the experimental data was determined. An equation for the prediction of the log spiral path of the air through the vaneless diffuser was derived that required only the diffuser inner and outer diameters and the inlet gas angle to determine the flow path length. A method to enable the prediction of the shape, as well as the corresponding stagnation pressure loss coefficient was determined and a correlation between the stagnation pressure loss coefficient and LWR was made using the log spiral path equation. However, due to the scattering of the experimental data between the two compressors it was impossible to determine an accurate prediction using this technique. It is believed however that with considerably more experimental data the accuracy of the model could be greatly improved and could replace the Rogers model in the CAPRICE code.

An extensive analysis of the diffuser geometry's effect on performance enabled the prediction of the optimum diffuser width for compressor 2. An assessment of the pressure recovery coefficient with various widths over a range of inlet gas angles was produced. It showed that as the diffuser is widened the net gain in pressure recovery reduces, and for a particular geometry there will be a limiting diffuser width from which no useful gain in recovery will exist.
Whilst this thesis has shown that significant improvements to the diffuser performance prediction have been made, it is believed that the inclusion of data from a range of different turbocharger centrifugal compressors would enable a more accurate prediction to be made and the effects of geometry to be more clearly seen.

10.2.4 Volute
The development of a 1-D performance model for a compressor volute casing has been presented. It has been shown that few models of this type for automotive compressors exist and this work has made a considerable contribution to the understanding of volute behaviour and the development of 1-D performance prediction. Using the experimental data from the volute it was shown that the fall in pressure ratio over the speed range was consistent with an increase in velocity as the mass flow increased, resulting in an increase in losses. The volute efficiency ratios were similar for both tested compressors, with a peak volute efficiency between 0.4 to 0.6.

Several significant phenomena have been identified as influencing the volutes characteristic particularly the differences between the contributing losses of the volute in terms of the scroll loss component and the exit diffuser loss component. These two losses varied significantly between the two tested compressors but it was found that in combination the total losses were very similar. This phenomenon would not have been possible to visualise without the experimental data produced by the interstage rig. Additionally, significant variations in static pressure were identified, which were consistent with severe pressure variations in the volute and was particularly noticeable in compressor 2. It is believed that this was due to turbulent mixing of the flow at the volute tongue, resulting in a drop in static pressure, which was translated up-stream into the diffuser. Compressor 1, showed very uniform static pressure around the compressor suggesting that across the flow range, the angle of the volute tongue and the angle of the flow are well matched and that turbulence carried up-stream is minimal.

Also, the correlations created from the experimental data identified that the flow continues to swirl beyond the volute exit and that the general assumption of most 1-D predictions of axial flow in this region is incorrect. It was however extremely difficult, due to the nature
Conclusions

Chapter 10

of the 1-D prediction, to incorporate this phenomenon into the code and therefore the general assumption of axial flow in the down-stream ducting had to remain. Whilst this flow condition could not be included here it should not be neglected. It is believed to be significant enough for the designer to consider it as an effect that could contribute to the shape of the overall compressor map.

As with the diffuser, correlations of the experimental data against existing techniques enabled the development of a model that could be used in CAPRICE to predict the volute’s performance. An equivalent geometry was formed from the volute and a comparison to the Runstadler straight-walled diffuser data was carried out. An average $C_{pr}$ for compressor 1 = 0.24±0.15 and for compressor 2 = 0.26±0.12. Since no comparable data for volutes was available these recovery coefficients was purely to identify whether or not this ‘quick’ prediction method to could be used. It is believed that additional volute equivalent geometries would enable a database of $C_{pr}$ to be created for a range of compressors.

Additional evaluations were made to model the volute as a pipe using both straight and helical flow paths in an attempt to provide a simple model relating the geometry and performance. The straight pipe model did not produce favourable results for either compressor and the stagnation pressure loss in both cases barely rose above 2 across the whole range. The helical flow path gave a slightly better correlation, showing a noticeable increase in loss, particularly at choke ($V_{w4}/V_{r4} = 1$ to 2) but as the flow neared surge ($V_{w4}/V_{r4} = 2$ to 4) the loss coefficient became constant at value of 2. It was concluded that whilst a simple geometry was preferable when developing a 1-D prediction technique, oversimplification could render the prediction worthless.

More complex methods such as the VDB model showed a reasonably good correlation with the experimental loss coefficient particularly in the mid range flow and it could be argued that the VDB model would have been better suited to determining the relationship between volute scroll geometry and flow conditions. However, it was believed that the complexity of the VDB model and its poor loss predictions notably at choke, would not have produced any further improvements to the volute prediction. It is therefore believed that the empirical model presented as the final solution was the most appropriate technique for CAPRICE.
The experimental data was shown to be relatively consistent for both compressors and few geometric features were required in order to describe the volute's performance.

It can be seen from the work carried out on the volute that the inclusion of an individual volute model has significantly improved the way in which CAPRICE predicts the performance of the compressor. The separation of the diffusion system into its constituent components gives the designer much more flexibility and a greater level of tuning of the component geometries. This work has demonstrated that the application of simple correlations and limited geometry can still result in an accurate method of prediction that the designer can be confident in.

10.3 Further Work

Whilst this research has improved the 1-D performance prediction technique – CAPRICE there are several areas where further work is required:

- More experimental data from a range of compressors is required to improve the correlations presented here. This would necessitate extensive experimental testing and it is recommended that improvements to the experimental rig could be made; including automating the data collection and data reduction and a method of recording stagnation pressure and temperature readings from inside the compressor should also be developed.

- An investigation into the affects of vaneless diffuser width on performance, with a view to predicting a range of stagnation pressure loss curves for varying volute casings. This would improve the diffuser characteristic prediction and enable the designer to determine 'off-optimum' designs for a range of compressors.

- The boundary layer models used in CAPRICE have not been assessed in this research. It is recommended that as part of an experimental program additional work to determine the boundary layer growth in the turbocharger compressor be under taken to improve the existing models further.
• The CAPRICE code is written in FORTRAN which whilst still a valid form of code has been superseded by languages such as C++ and Labview. It is proposed that the CAPRICE program be updated into a suitable language to enable it to be more easily modified.

10.4 Contribution To Research Field

It has been shown that over the years vast amounts of research has been carried out into the field of compressor performance prediction. Much of the research today uses complex tools such as CFD to produce 3-D models of fluid flow and geometry. It is sometimes forgotten that simple 1-D calculations can provide the designer with a good understanding of how his design will operate without the necessity for complex design work. The development of the CAPRICE prediction technique, enables the designer to carry out these initial calculations using only the barest of geometric features and speeds. It is believed that the research carried out here has contributed further understanding to the performance of the automotive turbocharger compressor and its components by the development of new prediction models and has supported the need for continued research into the development of 1-D prediction tools.
References

[1] Honeywell Turbo Technologies Ltd
Company website 2005

Company website 2005
http://www.holset.co.uk/files/2_5_2_1-exhaust%20gas%20recirculation.php

Principles & performance in diesel engineering
Ellis Horwood Ltd

Longman Group Ltd
ISBN 0582236320

Turbocharging the combustion engine.
Macmillan education Ltd. 1982

Turbomachinery: Basic theory & applications
Marcel Dekker inc. 1981
ISBN 0824715098

Evolution and outlook of turbochargers for vehicle engines.
Turbocharging & turbochargers.
Proceedings of the IMechE, C115/86, p.1 – 4

An integrated system for the design of radial compressors and turbines
C499/022/96, pp. 125 – 141

[9] E. Swain
A simple method for predicting centrifugal compressor performance characteristics
C405/040, pp.207 – 218

[10] E. Swain
The initial design of a centrifugal compressor

A method for performance prediction for centrifugal compressors
National gas turbines establishment, Pyestock, ARC, R & M No 3843, February 1980

Design and Performance of a high-pressure Ratio Turbocharger Compressor, Part 1: Design Considerations
References

Design and Performance of a high-pressure Ratio Turbocharger Compressor, Part 2: Experimental Performance

Review – Progress in Numerical Turbomachinery Analysis

[15] Y.A.Cengal, M.A. Boles
Thermodynamics – An Engineering Approach
International edition 1989
ISBN 0071004947

[16] B.S. Massey
Mechanics of Fluids
ISBN 0412342804

Centrifugal Compressors – A Strategy for Aerodynamic Design and Analysis
American Society of Mechanical Engineers Publications
First edition 2000
ISBN 0791800938

[18] E. Swain
Turbomachinery lecture notes. 1997

[19] T. Ferguson
The Centrifugal Compressor Stage
Butterworths & co publishers ltd, 1963

[20] F.J. Wiesner
A review of slip factors for centrifugal impellers

[21] Busemann
Das fOrderhohenverhAltniss radialer kreiselpumpen mit logarithmischspiraligen schaufeln
Zeitschrift fur angewandte mathematik und mechanic vol. 8 Oct 1928 pp. 372-384

[22] G.F. Wislicenus
Fluid Mechanics of Turbomachinery

[23] H.E. Sheets
The flow through centrifugal compressors and pumps
Trans. ASME vol. 72 1950 pp.1009-1015

[24] R.C. Dean, Jr., Y. Senoo
Rotating Wakes in Vaneless Diffusers
Journal of Basic Engineering, Trans. Of the ASME, 1960

[25] A. Whitfield
Slip factor of a centrifugal compressor and its variation with flow rate
Thermodynamics & fluid mechanics group

[26] F. Dallenbach
The aerodynamic design of centrifugal and mixed flow compressors
SAE technical paper 610160, 1961, pp. 2-30
[27] J. D. Denton
Loss Mechanisms in Turbomachines
The 1993 IGTI scholar lecture

[28] C. Rogers
Static pressure recovery characteristics of some radial vaneless diffusers

[29] D. Japikse
Future performance, design possibilities and goals for radial compressors.
C101/86
Proceedings of the IMechE, Turbocharging & Turbochargers. 1986, , p.31 – 50

Design and performance of a family of diffusing scrolls with mixed-flow impellers and vaneless diffusers.
NACA publication, report No 936, 1949

[31] D. Japikse
Opportunities In University Collaboration For Turbomachinery Development --
A Turbomachinery Perspective On Engineering Education.
The 10th International Symposium On Transport Phenomena And Dynamics Of Rotating Machinery
Honolulu, Hawaii, March 07-11, 2004
Concepts NREC Inc.

[32] Texas A & M University
Research project website
http://turbolab.tamu.edu/research/projects.html

[33] Cambridge University — Turbomachinery Research Group
Research project website
http://www.eng.cam.ac.uk/research/div-a/turbomachinery/research.html#unsteadytransition

[34] Canfield University – Power, Propulsion & Aerospace Engineering
Research project website
http://www.cranfield.ac.uk/prospectus/research/research.cfm?id=70

[35] R. Holmes
Turbocharger Vibration — A Case Study
C692/031/2002
IMechE 7th International Conference on Turbochargers and Turbocharging, pp. 91-100
ISBN 1860583830

[36] R.S. Amos
Trends in Turbocharger application
C602/038/2002
IMechE 7th International Conference on Turbochargers and Turbocharging, pp. 385-396
ISBN 1860583830

[37] M. Beecham
The global market for automotive turbochargers - Detailed information on company market shares, product installation forecasts and trends for passenger cars and commercial vehicles to 2010
2005 edition
ABOUT Publishing Group

Page 288
References

[38] F.S. Binder
Some Aero-thermodynamic Considerations for Future Developments of Small Turbochargers
C554/029/98
ISSN 1356-1448 ISBN 10860581501, pp 171-183

Development of Variable Geometry Turbocharger for Diesel Passenger Car
C554/005/98
ISSN 1356-1448 ISBN 10860581501, pp 329-347

[40] C. Arcoumanis, R.F. Martinez-Botas, C.C. Su
Measured and predicted flow characteristics of mixed flow turbocharger turbines
C554/028/98
IMechE 6th International Conference on Turbocharging and Air Management Systems, pp 157-170
ISSN 1356-1448 ISBN 10860581501

[41] N. Karamanis, R.F. Martinez-Botas, C.C. Su
Detailed flow measurements at the exit of a mixed flow turbine under steady flow conditions
ASME-GT-342, 1999, Pages: 1-9
International Gas Turbine and Aeroengine Congress and Exhibition (Indianapolis USA),

[42] N. Bains
A meanline prediction method for Radial Turbine Efficiency
C554/006/98
IMechE 6th International Conference on Turbocharging and Air Management Systems, pp 45-57
ISSN 1356-1448 ISBN 10860581501,

[43] N, Karamanis, R.F. Martinez-Botas
Mixed-flow turbines for automotive turbochargers: steady and unsteady performance
ISSN: 1468-0874

[44] B.M. Hande
Performance prediction of centrifugal compressors during the conceptual engineering phase
ASME, 94-GT-192

[45] B. Engels
Lifetime Prediction for Turbocharger Compressor Wheels – Why Titanium?
C602/037/2002
IMechE 7th International Conference on Turbochargers and Turbocharging, pp. 297-311
ISBN 1860583830

[46] Zhang, Hong, Ma, Chao-Chen
Geometric Parameter Optimizatiopn, Design and Performance Analysaia in a Vehicle Turbocharger
Centrifugal Compressor.
Beijing, Trans. of Institution, Vol 25, No 1, pp. 22-26, Jan 2005, ISSN 10010645

[47] C. Rodgers
Effects of Blade Number on the Efficiency of Centrifugal Compressor Impellers
Proceedings of ASME. TURBO EXPO 2000

[48] P.R.N. Childs, M.B. Noronha
The Impact of machining techniques on Centrifugal Compressor Impeller Performance
97-GT-456
Trans. Of the ASME, 1997

Page 289
References

A Study of Impeller-Diffuser Interaction part I – Influence on Performance
GT-2002-30381
Proceedings of ASME. TURBO EXPO 2002, June 3-6, Amsterdam, The Netherlands.

A Study of Impeller-Diffuser Interaction part II – Detailed Flow Analysis
GT-2002-30382
Proceedings of ASME. TURBO EXPO 2002, June 3-6, Amsterdam, The Netherlands.

[51] B.M. Hande
Performance Prediction of Centrifugal Compressors during the Conceptual Engineering Phase
International Gas Turbine Conference and Exhibition, ASME, 1994
ISSN 04021215

Numerical Investigation of Flow in Small Radial Turbocharger Turbine Volutes
Loughborough University, Internal Report, 2001

Review for the Commercial Development of a Small high-speed Centrifugal Compressor
C591/059/2001

[54] J.P. Johnston, R.C. Dean Jr.
Losses in vaneless diffusers of centrifugal compressors and pumps

Optimum Design of Straight-Walled Diffusers

[56] E.G. Reid
Performance characteristics of plane-wall two-dimensional diffusers
NACA TN 2888, 1953

[57] B.A. Waitman, L.R. Reneau, S.J. Kline
Effects of Inlet Conditions on Performance of two-dimensional Subsonic Diffusers

[58] C.A. Moore, S.J. Kline
Some effects of vanes and of turbulence in two-dimensional wide-angle subsonic diffusers
Technical notes 4080, NACA 1958

Performance and Design of Straight, two-dimensional Diffusers

[60] P.W. Runstadler, Jr., R.C. Dean, Jr.
Straight Channel Diffuser Performance at High Inlet Mach Numbers

[61] R.M. Halleen, J.P. Johnston
Inlet Mach number effects on subsonic diffuser performance

Flow Regimes in Curved Subsonic Diffusers
Journal of Basic Engineering, Trans. of the ASME, September 1962, pp. 303 - 316
[63] C.J. Sagl, J.P. Johnston
The Design and Performance of two-dimensional, Curved Diffusers

The Influence of Diffuser Channel Geometry on the Flow Range and Efficiency of a Centrifugal Compressor

The Influence of Diffuser Channel Length-Width Ratio on the Efficiency of a Centrifugal Compressor

[66] W. Jansen
Steady Fluid Flow in a Radial Vaneless Diffuser

[67] H. Krain
A study on centrifugal impeller and diffuser flow

A study of diffuser rotor interaction in a centrifugal compressor

[69] Y. Senoo, M. Ishida
Behaviour of Severely Asymmetric Flow in a Vaneless Diffuser

[70] M. Inoue
Radial Vaneless Diffusers: A Re-examination of the Theories of Dean and Senoo and of Johnston and Dean
Journal of Fluids Engineering, 1983

[71] R.L. Elder, M.E. Gill
A discussion of the factors affecting surge in centrifugal compressors

[72] K. Ludtke
Aerodynamic tests on centrifugal process compressors – the influence of the vaneless diffuser shape

[73] Z. Yingkang, S.A. Sjolander
Effect of Geometry on the Performance of Radial Vaneless Diffusers

[74] R.H. Aungier
Aerodynamic design and analysis of vaneless diffusers and return channels
International Gas Turbine and Aeroengine Congress and Exposition, 1993

[75] D. Japikse K. N. Oliphant
Turbo machinery Modeling: Explicit and Implicit Knowledge Capturing (2005a)
GT2005-68099
References

[76] D. Japikse, O. Dubitsky
Vaneless Diffuser Advanced Model (2005d)
GT2005-68130

[77] N.A. Cumpsty
Compressor Aerodynamics
Longman Scientific & Technical Group, 1989

[78] W. Steifel
Experience in development of radial compressors
Von Karman institute for fluid mechanics, lecture series 50, may 1972

[79] M. Sideris, R.A. Van den Braembussche
Evaluation of the flow in a vaneless diffuser followed by a volute

[80] P. Frigne, R. Van den Braembussche
A theoretical model for rotating stall in the vaneless diffuser of a centrifugal compressor

[81] A. Whitfield, D.V. Roberts
Alternative Vaneless Diffusers and Collecting Volutes for Turbocharger Compressors

[82] R.A. Van den Braembussche, B.M. Hande
Experimental and Theoretical Study of the Swirling Flow in Centrifugal Compressor Volutes

[83] E. Ayder, R. Van den Braembussche, J.J. Brasz
Experimental and Theoretical Analysis of the Flow in a Centrifugal Compressor Volute

Improved model for the design and analysis of centrifugal compressor volutes

[85] P. Frigne, R. Van den Braembussche
Distinction between different types of impeller and diffuser rotating stall in a centrifugal compressor
with vaneless diffuser

Meanline performance prediction of volutes in centrifugal compressors
International Gas Turbine Conference and Exhibition, Dusseldorf June 1986, ASME, 86-GT-216

Experimental and Numerical Investigation of the Flow in a Centrifugal Compressor Volute.
99-GT-79.
44th ASME Gas Turbine and Aeroengine Congress, June, 1999, Indianapolis, USA, ASME

Improved impeller/volute combinations for centrifugal compressors
GT2005-68894
50th ASME International Gas Turbine and Aeroengine Congress, Reno, USA, June 5-9, 2005
VKI RP 2005-30

Page 292
References

[R9] R.A. Van Den Braembussche  
Turbomachinery component design by means of CFD  
VKI RP 2002-11

[R10] C. Rogers  
Typical performance characteristics of gas turbine radial compressors  
Journal of Engineering for Power, Trans. of the ASME, April 1964, pp.161 – 175

[R11] G. Varghese, T.C. Mohana Kumar, Y.V.N. Rao  
Influence of Volute Surface Roughness on the Performance of a Centrifugal Pump  

Effects of Wall Shape on Flow Regimes and Performance in Straight, Two-dimensional Diffusers  

[R13] S.J. Kline  
On the nature of stall  
Journal of Basic Engineering, Trans. of the ASME, September 1959, pp.305 – 320

[R14] U. Seidel, M. Rautenberg  
Method for predicting choke flow in vaned radial diffusers based on thermodynamic and gas dynamic criteria.  
International Gas Turbine and Aeroengine Congress and Exposition, Netherlands, June 1994, ASME, 94-GT-157

[R15] W. Jansen  
Rotating stall in a radial vaneless diffuser  

Prediction of choking flow in centrifugal impellers  
Journal of Basic Engineering, Trans. of the ASME, March 1959, pp.29 – 35

[R17] A.R.S. McCutcheon  
Aerodynamic design and development of a high pressure ratio turbocharger compressor, C73/78  

Turbomachinery Performance Models (2005B) IMECE2005-79414  
ASME International Mechanical Engineering Congress & Exposition, November 5-11, 2005, Orlando, Florida

[R19] A. Whitfield  
Review of preliminary design and performance prediction techniques for centrifugal compressors  
part 1: preliminary design. C390/001  
Trans. of the IMechE 1989

[R20] A. Whitfield  
Review of preliminary design and performance prediction techniques for centrifugal compressors  
part 2: performance prediction. C390/034  
Trans. of the IMechE 1989

[R21] A. Whitfield  
Non-dimensional aerodynamic design of a centrifugal compressor impeller  
IMechE, 1991
[102] A. Whitfield, F.J. Wallace
Performance Prediction for Automotive Turbocharger Compressors

[103] A. Whitfield, A.B. Mohd Noor
Investigation of the flow characteristics of radial inflow turbocharger volutes
IMechE, 1990

[104] M.C.M.M. Campos, P.S.B. Rodrigues
Program finds centrifugal compressor operating point
Hydrocarbon Processing, 1990

[105] E.L. Stadler
Understand centrifugal compressor stage curves
Hydrocarbon Processing, 1986

Design of a compressor rotor using personal computers
UAE University 1988, pp.409 - 414

[107] S.N.J. Al-Zubaidy
Towards optimising the design of centrifugal impellers
UAE University 1990, pp. 227 - 233

Performance Prediction Method of Low Pressure Ratio Centrifugal Compressors
SAE Technical Paper, 1992, ISSN 01487191

[109] C. Bosman
An Analysis of three-dimensional Flow in a Centrifugal Compressor Impeller

Performance Evaluation of Centrifugal Compressor Impellers using three-dimensional Viscous Flow Calculations

[111] H.K. Versteeg, W. Malalasekera
An introduction to computational fluid dynamics
1st Edition
Longman scientific and technical group
ISBN 0582218845

[112] B. Lakshminarayana
An assessment of computational fluid dynamic techniques in the analysis and design of turbomachinery
—the 1990 freeman scholar lecture

The Validation Of Rapid CFD Modeling For Turbomachinery
Institution Of Mechanical Engineers, December 13, 14, 1999

[114] P. Schoonmaker
Preliminary Experience with an expert system providing initial centrifugal compressor sizing for performance prediction and analysis
ASME Paper IGTA conference 1991, ISSN 04021215
[115] C. Rogers
Influence of impeller and diffuser characteristics and matching on radial compressor performance, SAE technical paper 610160, 1961, p. 31 – 42

[116] B.S. Stratford, G.S. Beavers
The calculation of the compressible turbulent boundary layer in an arbitrary pressure gradient – a correlation of certain previous methods. ARC R and M No 3207, 1959

[117] J.H. Horlock
Axial flow compressors
New York academic press inc., 1958

[118] S.L. Dixon
Fluid mechanics, thermodynamics of turbomachinery
3rd edition
Pergamon press, 1978

[119] R.P. Benedict
Fundamentals Of Temperature, Pressure And Flow Measurement
2nd Edition
Wiley-Interscience Publication, 1977
ISBN 0471065617

[120] Recommended Practices For Measurement Of Gas Path Pressures And Temperatures For Performance Assessment Of Aircraft Turbine Engines And Components. (AGARD-AR-245)
AGARD
p. 56 paragraph 3.

[121] C.A. Georgakis, P.Timmis, P.C. Ivey
Circumferential Pressure Variations in Centrifugal Compressors C615/050/2003

[122] S.E. Haarland
Simple and explicit formulas for the friction factor in turbulent pipe flow

[123] R.C. Dean, L.R. Young
The fluid dynamic design of advanced centrifugal compressors
Fluid dynamics institute, Dartmouth collage, Hanover, USA, 1976

[124] N. Watson, D.R. Ingham
Compressible flow in a radial vaneless diffuser.
Conference on internal flows, University of Salford, paper 8, 1971

[125] Swain E, Meese H.
Extension of a centrifugal compressor performance prediction technique.
ISSN 1356-1448 ISBN 10860581501, pp33-43

[126] H.M.Meese, E. Swain
Analysis of Two Automotive Turbocharger Compressor Radial Vaneless Diffusers for Use in Improving a Performance Prediction Technique. 2001-GT-0319.
Proceedings of ASME. TURBO EXPO 2001, June 4-7, 2001, New Orleans, Louisiana, USA.
Citations:

[127] N.S. Nathoo, W.G. Gottenberg
A look at performance analysis of centrifugal compressors operating with mixed hydrocarbon gasses

[128] M.V. Casey
The Effects of Reynolds Number on the Efficiency of Centrifugal Compressor Stages
Journal of Engineering for Gas Turbines and Power, Trans. of the ASME,
vol. 107, April 1985, pp.541 - 548

[129] R.I. Lewis
Turbomachinery performance analysis
Wiley & son Inc, 1996

[130] O.F. Balje
Turbomachines: a guide to design, selection and theory
Wiley & son Inc, 1981
ISBN 0471060364

[131] H. Schlichting
Application of Boundary-Layer Theory in Turbomachinery
Journal of Basic Engineering, Trans. of the ASME, December 1959, pp.543 - 551

Some effects of inlet blockage and aspect ratio on diffuser performance
Journal of basic engineering, Trans. of the ASME, September 1969, pp.551 - 553

[133] R.C. Pampreen
A Blockage Model for Centrifugal Compressor Impellers
I. Appendix

I- i Straight Walled Diffusers

Reid[56] investigated the changes in $C_{pr}$ and flow by systematically altering the LWR, AR and $\theta$ for a series of straight-walled diffusers for subsonic flow. The authors results agree well with Kline et al along with two other points being:

- As the divergence angle increases for a fixed length, a steady flow is maintained until the $C_{pr}$ attains it's maximum value. Above this point the flow becomes unsteady and begins to degenerate.

- Improvements to $C_{pr}$ could be made by the addition of a short exit duct of constant cross-section.

Waitman et al[57] developed the theories linking LWR, $\theta$ and $C_{pr}$ by expanding on[58] to include more geometrical changes and inlet conditions. Tests were carried out for symmetrical as well as distorted flows. Velocity profiles across the width of the diffuser were plotted for varying LWR. It was found that the lines separating the no-appreciable stall, large transitory stall and fully developed stall regions changed position depending on whether there was high or low inlet turbulence. It was concluded that $C_{pr}$ was a function of turbulence intensity. Disturbances were inserted in the path of the fluid flow using bluff objects and splitter vanes. It was found that these disturbances to the flow increased the pressure recovery at all values of divergence tested. The authors suggest that the introduction of bluff objects in the flow;

'Verifies the quantitative importance of large-scale mixing at the inlet on diffuser behaviour'

They concluded that $C_{pr}$ is changed if there are flow obstructions in the flow up-stream of the diffuser throat and the magnitude and direction of $C_{pr}$ depends on the type of obstruction and its location.

Reneau et al[59] looked at LWR, $\theta$ and AR against effectiveness $\eta$ (as defined by Kline[55] and shown in figure 2-4), head loss $H_L$ and pressure recovery $C_{pr}$ at both stalled and unstalled conditions and confirmed the findings of Fox et al[62]. They showed that as AR or $\theta$
was increased for a fixed LWR the pressure recovery curve could be divided into sections by the flow regimes discussed by Kline et al.[55] shown in figure I-1.

The authors conclude that:

- $C_p$ is dominated by AR in the un-stalled region.
- $C_p$ is dominated by $2\theta$ in the transitory region.
- $C_p$ is dominated by no geometric parameters in the jet flow region.

I-ii Curved Diffusers

Sagi et al.[63] investigated the effects of geometry and flow on performance and design based on Kline et al.[55] and Moore et al.[58]. The authors state that flow regimes are strongly dependent on the local pressure distribution along the walls of the diffuser (boundary layer growth). To test this theory they carried out a series of experiments on a range of curved diffusers, using work by Carlson et al.[91], to investigate pressure distribution and loading and how these affect the performance. The curved diffuser underwent a higher loading (higher adverse pressure gradient) near the exit of the inner wall whereas the outer wall experiences very little loading.

They showed that this type of passage considerably altered the flow profile producing large changes in the flow regime. The most significant change being the drop off in area ratio with increasing turning angle shown in figure I-2. The authors suggested a reverse design
technique, in that by specifying the loading on the walls and velocity distribution across the passage it would be possible to calculate the curvature of the diffuser passage. They concluded that the pressure loads on the walls must be below stall to optimise the pressure distribution.

I- iii Simple Curve-fitting Impeller Efficiency Model

A simple curve-fitting method was used to predict the individual efficiency ratio curve shapes, as a function of blade speed \((U_2/\sqrt{T_{01}})\). The curve-fits for each set of data were determined from figure 5-29, along with the corresponding quadratic algebraic expressions using the method of least squares for each speed. Using the coefficients of each of these curves, a quadratic expression for each coefficient against blade speed was produced. Figure 5-30 shows the \(x^2\) function as an example. Using these expressions, the impeller efficiency ratio \((\eta_{imp}/\eta_{imp\ peak})\) could be determined at any speed over a range of flow to choke flow \((m/m_{\text{max}})\) values using a single equation, eqn.(5-7).

The process for determining the efficiency ratio is as follows:

- The speed function coefficients are found from:
  \[
  x^2 = -0.0253(U_2/\sqrt{T_{01}})^2 + 0.7566(U_2/\sqrt{T_{01}}) - 6.6236 \\
  x = 0.0413 (U_2/\sqrt{T_{01}})^2 - 1.2546(U_2/\sqrt{T_{01}}) + 10.8938 \\
  c = -0.0170(U_2/\sqrt{T_{01}})^2 + 0.5293(U_2/\sqrt{T_{01}}) - 3.6543
  \]

- the coefficients are input into the efficiency ratio equation:
  \[
  \frac{\eta_{imp}}{\eta_{imp\ peak}} = x^2(\dot{m}/m_{\text{max}})^2 + x (\dot{m}/m_{\text{max}}) + c \quad \text{eqn. (5-7)}
  \]
The volute flow is determined using an iterative technique which adjusts the diffuser and impeller conditions until the diffuser outlet conditions are constant. The model consists of the following components:

- Impeller flow calculations – to determine the velocity triangles using the exit static pressure distribution.
- Diffuser flow calculations – to determine the diffuser exit velocity triangles and static and stagnation pressure distribution.
- Volute flow calculations – to approximate the 3-D flow in the volute.

The volute is divided up into a number of small sections shown in figure I-8. The conservation of mass and momentum enables the calculation of the outlet conditions of each segment as a function of the inlet into that section plus the incoming flow from the diffuser outlet. The fluid entering at the volute tongue is assumed to remain in the centre of the flow with the fluid entering downstream wrapping around it. The inviscid variation in total pressure and swirling velocity can then be related to the total pressure and radial velocity at the diffuser exit \( P_4 \) and can be determined from continuity.

\[
m = \int_0^\theta [\rho_4(\theta) b_4 R_4 V_{r4}(\theta)] d\theta = \int_0^{\theta^*} 2\pi \rho V_r r dr
eqn{I-1}
\]

Where \( r'(\theta^*) \) is the radius of the fluid entering at position \( \theta^* \) figure I-3. The inviscid swirling velocity at \( \theta^* \) can be found from the conservation of momentum taking into account the change in position.
\[ V_{s(\text{inv})}(\theta^*, r') = \frac{V_{r4}(\theta^*) r_w(\theta^*)}{r'(\theta^*)} \]  

Therefore, the inviscid total pressure can be defined as a function of local pressure at vaneless diffuser outlet.

\[ P_{0(\text{inv})}(\theta^*, r') = P_{04}(\theta^*) \]

Thus, the real total pressure at the cross-section is a function of the inviscid total pressure and is found from a difference between the inviscid and real swirl kinetic energy.

\[ P_0(\theta^*, r) = P_{0(\text{inv})}(\theta^*, r) - \frac{1}{2} \rho(\theta^*, r) \left[ V_{s(\text{inv})}^2(\theta^*, r) - V_{s}^2(\theta^*, r) \right] \]

Where

\[ V_s(\theta^*, r) = \frac{V_s(\theta^*, r_w)}{r_w(\theta^*)} \times r \]

From a first approximation, the swirl velocity at the wall near the diffuser exit is assumed to be:

\[ V_s(\theta^*, r_w) = V_{r4}(\theta^*) \]

This will change during consecutive iterations. Static pressure variations over the cross-section are determined from the radial equilibrium between the static pressure gradient and the forces due to swirl.

\[ \frac{dP}{dr} = \rho \frac{V_s^2(\theta^*, r)}{r(\theta^*)} \]

from which the through-flow velocity is determined from this relationship by integration:

\[ \int_r^{r'} dP = \rho \int_r^{r'} \frac{V_s^2}{r} dr \]

The through-flow velocity \( V_t \) is calculated from:

\[ \frac{\rho(\theta^*, r)V_t^2(\theta^*, r)}{2} = P_0(\theta^*, r) - p(\theta^*, r) - \frac{\rho(\theta^*, r)V_s^2(\theta^*, r)}{2} \]

Where \( V_s \) defined above is modified by changing \( V_s(\theta^*, r_w) \) until:

\[ V_t(\theta^*, r_w) = \frac{V_{w4}(\theta^*) R_s(\theta^*)}{R_t(\theta^*)} \]
The static pressure at the wall $p(\theta^*, r_u)$ is assumed to equal the diffuser outlet static pressure $p_4(\theta^*)$ only at the first iteration and will be modified in consecutive iterations to vary $V_t$ and density until $m$ is constant.

Continuity alone cannot explain the static pressure distortion in the volute and therefore the circumferential pressure variation around the volute needs to be determined. Van den Braembussche defined the conservation of angular momentum in the tangential direction from the following equation. Here curvature of the volute walls is considered as a function of the divergence of the walls and the change in volute centre.

$$R_c(\theta_i)A(\theta_i)[p(\theta_i) + \rho(\theta_i)V_t^2(\theta_i)] + dA_{iw} \left[ p - \rho V_t^2 \frac{R_c - R_{ciw}}{R_{ciw}} \right] R_{ciw} + dA_{ow} \left[ p - \rho V_t^2 \frac{R_c - R_{cow}}{R_{cow}} \right] R_{cow}$$

Where $V_t$ is equal to the mass-averaged value between inlet and exit of the section. $dA_{iw}$ and $dA_{ow}$ are the projections of the volute inner and outer walls as a function of the changing volute centre $R_c$. The static pressure at the diffuser outlet can then be adjusted. The static pressure is found to be equal to the average static pressure at the centre of the down-stream cross-section corrected by the pressure difference between $R_c$ and $R_4$ due to the centrifugal forces in the curved scroll.

$$p_4(\theta_o) = p_c(\theta_o) - 2\rho(\theta_o) \frac{V_t^2(\theta_o)}{R_c(\theta_o) + R_4}[R_4 - R_c(\theta_o)]$$

The comparison of the predicted and experimental results shows excellent agreement. However, deficiency in the model is its inability to predict flow separation in the volute.

**I- vi Boundary Layer Theory**

If a fluid flows over a flat surface, it is usual to find a thin layer of flow between the surface and the core flow of the fluid. This thickness is denoted by $\delta$ and is called the 'boundary layer' shown in figure I-12. The relationship between these velocities is generally referred to as the blockage factor and is defined as:

$$B = 1 - (\delta / U) \quad \text{eqn. (I-8)}$$

If a passage of varying cross-sectional area is considered such as a diffuser, then the more diverging the passage the more likely flow is to separate from the walls. With a longer
passage of the same inlet and outlet areas as a shorter one, separation is less likely to occur. If flow separation occurs, the available area through which the flow passes is smaller than the actual geometric area. This relationship can be defined as the ‘ratio of the remaining flow area after some reducing item, to the total geometric flow area’[93][127-130] and can be written:

\[ B = 1 - \left( \frac{\alpha}{U} \right) = 1 - \left( \frac{A_{\text{effective}}}{A_{\text{geometric}}} \right) \quad \text{eqn. (I-9)} \]

Much of the early work on 2-D boundary layer theory[123][131-133] for turbomachinery was carried out by Schlichting[131], predominately on the boundary layer growth on the blades of axial cascades. Schlichting investigated boundary layer theory in terms of aerodynamic effects based on the loss coefficients of lift and drag. He was aware that it was necessary not only to describe the boundary layer in terms of loss coefficients but also the inlet flow angle, the Reynolds number, and all the geometric parameters which describe the cascade. Schlichting showed that the additional tip clearances and interference affects between the blades and sidewalls gave rise to secondary losses and increased the chance of boundary layer separation.

Johnston et al[132] presented work relating to the effects caused by blockage and low aspect ratio on the performance of a straight-walled diffuser. The diffuser blockage factor was defined as in equation (1-13) Where \( A_{\text{eff}} = \frac{m}{\rho V} \) & \( A_{\text{geo}} = 2Db \). The diffuser was designed to ensure that the boundary layer on each wall at inlet was uniform and identical in shape and thickness. Under these conditions, the viscous effects due to corners were negligible and the aspect ratio was found to relate to the blockage by:

\[ B = (2\delta/w)(1 + (1/AS)) \]

The inlet boundary layer displacement thickness, \( \delta \), Mach and Reynolds number was held constant for each test. Five aspect ratios were tested with three flow regimes, unstalled, first stall limit and transitory stall. For the unstalled and first stall limit flows \( C_{pr} \) increased and \( B \) decreased with increasing AS. This compared well with earlier findings[57][62], however, in the transitory stall area both \( C_{pr} \) and \( B \) decreased with increasing AS. The authors suggested this was due to the stalling boundary layers and unsteady core flow in the diffuser. Their conclusion is a warning not to expect \( C_{pr} \) to increase with a reduction in blockage especially if the flow is in the stalled region.