The design and performance of a 1.9m x 1.3m indraft wind tunnel

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

Additional Information:

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

Metadata Record: https://dspace.lboro.ac.uk/2134/7194

Publisher: © Guru Johl

Please cite the published version.
This item was submitted to Loughborough’s Institutional Repository (https://dspace.lboro.ac.uk/) by the author and 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/
The Design and Performance of a 1.9m x 1.3m Indraft Wind Tunnel

by

G. S. Johl

A Doctoral Thesis

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

06/04/2010

© Guru Johl 2010
DEDICATION

To my wonderful wife Stacy and my Dad
ABSTRACT

This Thesis has endorsed employing a novel indraft configuration for a severely spatially and financially constrained wind tunnel aimed at undergraduate and postgraduate aeronautical and automotive instruction.

The novel ‘horseshoe’ indraft configuration employed may be considered to either bend a traditional open circuit or remove corners 3 and 4 from a traditional closed circuit. By connecting the inlet and exit to atmosphere the new configuration prevents pressure loading of the surrounding building; eliminates the problem of exhausting a jet within a laboratory; and eliminates costs associated with a heat exchanger. The modest budget (£350,000) is commensurate with the financial means of a University or small enterprise.

Aerodynamic performance data suggests future designers should not shy away from an indraft tunnel by default: Velocity uniformity in the working area of jet has been shown to vary by less than 0.3% of the mean in the presence of ambient gusts up to 11.5% of the test velocity. Lift and drag coefficients derived from a 27% scale Davis automotive model (5.9% frontal area blockage) repeated to 6 units (0.6%) and 2 units (0.2%) respectively in the presence of ambient gusts up to 13% of the test velocity. Axial turbulence intensity was measured to be in the region of 0.15% (negligible ambient gusts) and 0.35% (ambient gusts up to 16% of the test velocity). This data compares favourably to that for the significantly larger NASA Ames 80ft x 120ft open circuit wind tunnel. Maximum test section velocity has been shown to be in excess of the desired 40m/s. The test section boundary layer closely follows the profile for a 1/7th power law turbulent boundary layer, which suggests the contraction is free from separation.

This Thesis contributes to the body of knowledge by publishing performance data for a new type of wind tunnel configuration. It also augments existing design guidelines and ‘rules of thumb’ by providing a complete reference point (including design flowcharts) for the design of comparable low speed wind tunnels. The Thesis offers the following specific conclusions and implications:

**Screens:** Whilst the inlet filer mesh is effective at damping ambient gusts it suffers the worst correlation to the governing equations (significant under prediction of loss), likely due to wire-wake coalescence. This highlights the importance of performing pipe rig tests for screens with open areas significantly less than 57%. Safety screen loss was under-predicted (assumed drag coefficient, $C_{D}$ of 1.0 due to treatment as isolated wires). Whilst measurements suggest a $C_{D}$ of $\sim$1.25 designers are advised to conduct pipe rig tests.  

**Contraction:** To allow pressure gradients to decay prior to the working section, it is advised that the parallel duct at end of the contraction be 1 hydraulic diameter rather than the 1 hydraulic radius proposed by the major texts.  

**Working section:** To allow for model wake recovery (and hence reduce the effect of non-uniformity on the downstream diffuser), a working section length-to-diameter ratio of 2.5 is suggested rather than 2 proposed by the established texts. Additionally, the static ports of tunnel pitot-static should be at least 0.55 hydraulic diameters upstream of the model leading edge to position them away from the static pressure signature of the model.  

**Diffusers:** Whilst the safety screen would ideally have to be removed to prove the hypothesis - it is suggested that turbulent mixing aft of the safety screen (located at the end of the working section) appears to offer a $\sim$10% $C_{p}$ improvement to the first diffuser.  

**Corner cascades:** Whilst the established texts focus on corner loss coefficient ($K_{L}$) this Thesis has shown that $K_{L}$ should not be the sole metric used to select the space-to-chord ratio (s/c) of corner cascades. Uniformity far downstream of a test cascade has been shown to improve with more closely spaced vanes (s/c of 0.190 rather than 0.237) despite $K_{L}$ being similar. Improvements to inlet boundary layer quality have also been shown to reduce $K_{L}$.  

**Fan:** The fan static pressure rise was measured to be less than predicted due to smaller than expected leakage losses. A leakage loss of 2.5% is therefore proposed rather than the 10% suggested by the major texts.
ACKNOWLEDGEMENTS

To my supervisors:

Martin – thanks for picking up the phone and asking me (on several occasion) where my PhD was, and also for the late meetings!

Pete – thanks for offering me my first real job in the summer of 1997 (modernising the Department’s old open-jet) and for your continued support since then.

To you both – thanks for allowing me to be part of a really rewarding project. I think we did something quite interesting and will be pleased if this Thesis is allowed to stand as a record of that.

Thanks also to Professor Stan Stevens for the idea of bending an open circuit...and also to Dr Jon Cole and Dr Dachun Jiang for their advice at various stages of the project.

Well done to the build Team: Tony Eyres, Keith Coulthard, Grenville Cunningham, Rob Hunter, Pradip Karia, Geoff Knowles, Norman Randall, Peter Stinchcombe and the late Kevin Springthorpe.

To my close friends (and ex-office mates!) Dr James Lloyd and Dr Duncan Priestley - your advice and encouragement throughout have been invaluable – particularly during my (intermittent!) writing up period after I had left the Department. Thanks also to James’ dad Peter Lloyd for proof reading my final draft.

Thanks to my examiners Professor Kevin Garry and Dr Zhiyin Yang for a rigorous, interesting and enlightening technical discussion.

Finally, a very special thank you to my lovely wife, Dr Stacy Clemes for helping me in more ways than I can remember or recount - and to my parents-in-law Carole and Tony for allowing me to clutter the dining table for days on end!
PUBLICATIONS ARISING FROM THIS THESIS

Journal Papers


Conference Papers

CONTENTS

Dedication
Summary
Acknowledgements
Publications arising from this Thesis

Journal Papers
Conference Papers

Contents
Nomenclature

1 Introduction .................................................................................................................1-1
  1.1 Motivation..................................................................................................................1-1
  1.2 Thesis Objectives......................................................................................................1-3
  1.3 Design Specification and Constraints .......................................................................1-4
  1.4 Types of Wind Tunnel .............................................................................................1-5
  1.5 Preliminary Closed Circuit Design ..........................................................................1-8
  1.6 Indraft Design ........................................................................................................1-9
    1.6.1 General description of the proposed indraft tunnel ........................................1-12
    1.6.2 Construction materials and techniques ..........................................................1-14

2 Design Methodology ..................................................................................................2-1
  2.1 Bellmouth intake .....................................................................................................2-1
  2.2 Settling chamber .....................................................................................................2-3
    2.2.1 Inlet filter mesh .................................................................................................2-3
    2.2.2 Honeycomb selection .......................................................................................2-5
    2.2.3 Loss and turbulence reduction across mesh screens ......................................2-6
    2.2.4 Screen selection ...............................................................................................2-8
  2.3 Contraction .............................................................................................................2-10
    2.3.1 Contraction ratio ..............................................................................................2-11
    2.3.2 Velocity under-shoot and over-shoot .............................................................2-11
    2.3.3 Corner fillets .....................................................................................................2-12
    2.3.4 Length ..............................................................................................................2-12
3 Calibration and Commissioning ......................................................... 3-1

3.1 Boundary Layer .............................................................................. 3-1
3.2 Velocity Uniformity ....................................................................... 3-4
  3.2.1 Method: .................................................................................. 3-4
  3.2.2 Still air baseline ...................................................................... 3-5
  3.2.3 Repeatability of still air baseline ............................................. 3-7
  3.2.4 Effect of ambient winds ......................................................... 3-9
3.3 Turbulence Intensity ...................................................................... 3-12
  3.3.1 Method .................................................................................. 3-12
  3.3.2 Results and discussion ........................................................... 3-14
3.4 Distribution of Roof Static Pressure Coefficient .............................. 3-16
3.5 Repeatability of Aerodynamic Coefficients .................................... 3-20
4 Comparison of Measured and Predicted Losses .........................4-1

4.1 Comparison of Overall Tunnel Performance .................................................4-2

4.2 Screen Losses .........................................................................................4-6

4.2.1 Inlet filter mesh ..............................................................................4-6

4.2.2 Screen 2 .........................................................................................4-7

4.2.3 Safety screen / Exit guard mesh .....................................................4-7

4.2.4 Reynolds number scans .................................................................4-8

4.3 Turning Vane Losses ........................................................................4-10

Corner 1 .............................................................................................................4-11

Corner 2 .............................................................................................................4-12

4.4 Diffuser Losses ...................................................................................4-16

4.4.1 First diffuser ..................................................................................4-16

4.4.2 Second diffuser ..............................................................................4-19

4.4.3 Third diffuser .................................................................................4-20

4.5 Graphical summary of circuit total pressure variation .......................4-22

5 Conclusions and Implications .................................................................5-1

References

Appendix A (Low Speed Indraft Wind Tunnel Design Flow Charts)
NOMENCLATURE

$\rho$  air density  \( \text{kg/m}^3 \)

$\theta$  diffuser included angle and momentum thickness  \( \text{degrees} \)

$\eta$  efficiency

$\alpha$  kinetic energy coefficient

$\psi$  screen aperture  \( \text{mm} \)

$\beta$  screen fractional open area

$A$  area  \( \text{m}^2 \)

$A/D$  analogue to digital

$A_r$  area ratio

$B$  Aerodynamic blockage ratio

$c$  chord  \( \text{mm} \)

$C_D$  drag coefficient

$C_L$  lift coefficient

$C_{MX}$  rolling moment coefficient

$C_{MT}$  pitching moment coefficient

$C_{MZ}$  yawing moment coefficient

$C_{p_s}$  static pressure recovery coefficient

$CR$  contraction ratio

$CTA$  Constant Temperature Anemometer

$C_Y$  side force coefficient

$D$  diameter or hydraulic diameter  \( \text{mm} \)

$ER$  energy ratio

$f_a$  axial turbulence reduction factor

$H$  stagnation or total pressure  \( \text{N/m}^2 \)

$J$  local height or width of contraction measured from the centreline  \( \text{mm} \)

$K_L$  local total pressure loss coefficient

$K_o$  component of screen loss coefficient due to open area

$L$  length  \( \text{mm} \)

$M$  screen mesh width  \( \text{mm} \)

$n$  number of screens and power for contraction contour

$P$  power  \( \text{W} \)

$p$  static pressure  \( \text{N/m}^2 \)

$q$  dynamic pressure  \( \text{N/m}^2 \)

$R$  radius or hydraulic radius  \( \text{mm} \)

$Re$  Reynolds number

$RF$  reserve factor

$s$  vane spacing  \( \text{mm} \)

$Tu$  turbulence intensity  \( \% \)

$u$  instantaneous velocity (x-component)  \( \text{m/s} \)

$U$  time averaged axial velocity  \( \text{m/s} \)

$x$  axial length co-ordinate  \( \text{mm} \)

$X$  location of contraction matchpoint or point of inflection

$y$  local boundary layer height  \( \text{mm} \)

$Y$  local width measured from the centreline of the working section  \( \text{mm} \)

$Z$  local height measured from the floor of the working section  \( \text{mm} \)

$\delta$  boundary layer thickness  \( \text{mm} \)

$\delta^*$  displacement thickness  \( \text{mm} \)

$\theta$  momentum thickness  \( \text{mm} \)
indices and subscripts

∞ freestream
1 upstream of component
2 downstream of component
C contraction
HC honeycomb
L local
W wire
WS working section
1 INTRODUCTION

“…Wind tunnel design lies somewhere between an art and a science, with occasional excursions into propitiatory magic…”

P Bradshaw, and RC Pankhurst
The Design of Low Speed Wind Tunnels
National Physical Laboratory
1964

1.1 MOTIVATION

This Thesis arose from the belief that, in light of severe spatial and financial constraints, the design specification (section 1.3), for a new, low-speed wind tunnel at the Department of Aeronautical and Automotive Engineering (AAE), Loughborough University, UK, might best be met by a previously untried indraft configuration.

AAE had decided that it required a new wind tunnel to support its teaching and research objectives. Consequently, a literature survey and set of preliminary calculations were performed to determine whether the design specification was realistic and whether a traditional closed circuit tunnel would be suitable.

During this process, it became apparent to the author that whilst very valuable design guidelines and rules of thumb from the likes of Bradshaw and Pankhurst (1964), Barlow (1999), Mehta and Bradshaw (1979) and Wolf (1993) did exist, the dataset was by no means exhaustive. As an example, the guideline for separating turbulence reduction screens by 0.05D (Bradshaw and Pankhurst, 1964), where D is the hydraulic diameter, falls down when space is severely limited. This rule of thumb is also purely a function of duct geometry rather than accounting for the configuration of the screen.

This observation is by no means new. During the design of the NASA Ames 80ft x 120ft indraft wind tunnel, Ross (1989), noted that:

“…The design of closed circuit wind tunnels has historically been performed using ‘rules of thumb’ which have evolved over the years into a body of useful guidelines…”
Necessary details required for the pragmatic design of each tunnel component, such as equations which define a suitable contraction contour in the absence of access to CFD (Su, 1991); and those which quantify axial turbulence reduction achieved by mesh screens (Loehrke, 1972); were either buried as references within the design guidelines highlighted above or required specific searches of the literature.

There appeared, therefore, to be a place alongside the established guidelines for a complete reference point, which presents the information required to design each component of a low speed wind tunnel. Effectively a document, which the author himself would have found useful at the start of the design process.

In addition to the above, it became apparent to the author and AAE that the spatial constraints would lead to an inadequate version of a traditional closed circuit tunnel (discussed further in section 1.4). The capital and running costs associated with a heat exchanger to maintain a consistent working section air temperature during a test were also a concern. A traditional open circuit design (ingesting and exhausting into a room) was deemed not to be feasible, given the consequent pressure loading of the surrounding building.

A novel ‘indraft’ layout was therefore conceived, which effectively bent a traditional ‘straight-through’ open circuit tunnel into a ‘U’ and connected the inlet and exit to atmosphere through two apertures in the external wall of the building. A further complication was that a building located opposite the inlet and exit was likely to create a recirculation zone. The exit was sufficiently far from surrounding workspaces that noise was not a concern.

It became clear from a search of the literature that no indraft tunnels of comparable layout and size had been built; primarily due to concerns over achieving acceptable test section flow quality as a consequence of variations in ambient winds.

In his background research for the design of the NASA Ames 80ft x 120ft indraft wind tunnel, Ross (1989), noted that:

“...the development of indraft wind tunnels however, has not been well documented. The design of indraft wind tunnels is therefore generally performed using a more intuitive approach, often resulting in a facility with disappointing performance. The primary problem is a lack of understanding of the flow in the inlet...”
Although significantly larger and of different layout to the tunnel presented in this Thesis, the NASA Ames 80ft x 120ft (the World’s largest wind tunnel), depicted in Figure 1-4, was of the indraft type and had been successfully designed to achieve flow quality targets that were not dissimilar to those requested by AAE. The designers, (Ross et al, 1986) found that the use of a vertical cascade at the inlet plane (to redistribute the entrained flow), with a high loss screen placed immediately downstream of the cascade (to maximise the mixing length downstream), produced the desired test section flow quality in the presence of ambient winds up to 15% of the test section velocity.

Therefore, although no calibration and commissioning data existed for an indraft tunnel directly comparable to the one presented in this Thesis, the results from Ames and further preliminary calculations, suggested that there was sufficient provision within the pressure loss and financial budgets, for the inclusion of inlet cascades/louvers, exit vanes (to direct exit flow away from the inlet) and a high-loss inlet mesh to help achieve the desired working section flow quality from the indraft wind tunnel outlined in section 1.5.

1.2 THESIS OBJECTIVES

In conclusion, this Thesis aims to contribute to the body of knowledge in the following ways:

- Present an argument for the selection of an indraft configuration to satisfy the design specification.
- Discuss the methodology used to conduct the pragmatic design of each tunnel component. Since many of the components are not unique to an indraft tunnel, the discussion is relevant to the design of low speed wind tunnels in general.
- Present calibration and commissioning data since published data for a comparable indraft wind tunnel does not currently exist. Also to quantify how these data are affected by ambient conditions.
- To compare how the wind tunnel as a whole, and individual modules in particular, performed relative to theoretical predictions in order to help future designers tackle unconventional circuit configurations with greater confidence.
- Produce a Thesis that will stand alongside the established (but by no means exhaustive) design guidelines and rules of thumb, and act as a complete reference point for information pertaining to the pragmatic design of each component of a low speed wind tunnel.
1.3 DESIGN SPECIFICATION AND CONSTRAINTS

AAE required that the working section be closed throat and accommodate as a minimum, a 25% scale model of a typical automotive saloon (full scale frontal area \( \approx 2m^2 \)) at approximately 5% blockage (based on model frontal area). Aeronautical half-models up to a semi-span of 1m and chord of 0.3m were also to be accommodated.

Whilst it is acknowledged that a scale of 25% is considered small for vehicle development, the facility was proposed for research using simplified bluff body models. Consequently, for this purpose, the scale and length based Reynolds numbers are sufficient.

In order to achieve suitable test Reynolds numbers \((Re)\), a working section velocity in excess of 40m/s was requested. The velocity variation in the working area of the jet was required to be less than 0.3% deviation from the mean velocity at 40 m/s (Barlow, et al, 1999). Since the turbulence level in the upper atmosphere is significantly lower than when close to the ground, it was considered desirable for the purposes of aeronautical testing, for working section axial turbulence intensity to be in the region of 0.1% at 40 m/s. Although the difficulty inherent in confidently measuring flow angularity (due to rig deflection) meant that no specific flow angularity target was specified, it was acknowledged that this ought to be within +/- 0.5deg of zero for pitch and yaw angle.

Whilst tunnel health, operational and pressure measurement software would be written in-house (partly by the author and partly by final year project students), the fan and underfloor balance were to be outsourced. The use of an elevated ground plane rather than a boundary layer suction system was favoured. This suited the indraft design that was eventually adopted since the test section static pressure would be below atmospheric and hence complicate sealing around a moving ground.

The available space envelope for the wind tunnel measured 18m (length) x 10m (width) x 7.5m (height), and fan power consumption needed to be less than 140kW. The budget for the complete project was £350,000.
1.4 TYPES OF WIND TUNNEL

An erudite discussion of different wind tunnel types has already been presented by Barlow et al. (1999), and so only a brief overview is needed here. At an holistic level, wind tunnels are classified as being either high or low speed. Barlow qualifies the latter as having maximum speed capability of up to Mach ≈ 0.4 (134 m/s). The wind tunnel discussed in this Thesis is therefore of the low speed variety.

As proposed by Pankhurst and Holder (1968), the main requirement of a wind tunnel is to produce a parallel, steady, spatially uniform stream of high velocity air for models undergoing tests in the working section. A simple wind tunnel might therefore be a constant area ring, where the fan is solely required to overcome the frictional losses associated with moving the air around the circuit, and the drag produced by the model.

Unfortunately, this simplified tunnel lacks a means of eliminating the non-uniformities introduced into the airstream by the fan and model, and boundary layer development along the walls. Its constant area design also means that all sections of the wind tunnel are subjected to the same, high, test section velocity. Any flow conditioning devices (such as screens, honeycomb or turning vanes), introduced into the circuit in an attempt to improve flow quality upstream of the model would create additional drag, and consequently a loss of total (or stagnation) pressure, $\Delta H$, across the device. This is commonly defined as:

$$\Delta H = K_L q_1$$

Where, $K_L$ is the total pressure loss coefficient across the device and $q_1$ is the inlet dynamic pressure defined as:

$$q_1 = \frac{1}{2} \rho U_1^2$$

Where, $\rho$ is air density and $U_1$ is the mean bulk inlet velocity. Furthermore, since:

$$P = \Delta HQ$$

Where, $P$ is power and $Q$ is the volume flow rate through the device, which is defined as:

$$Q = A_1 U_1$$
Substitution gives:

\[ P = \frac{1}{2} K \rho A U_1^3 \]

It is apparent from this analysis that the power required to overcome drag is proportional to the cube of the local velocity. Introducing flow conditioning devices into a constant area wind tunnel circuit would therefore require substantial increases in fan power in order to maintain the same working section velocity. By way of example, for a tunnel with a constant cross section of \(2.5\text{m}^2\) and a velocity of \(40\text{m/s}\), adding a single turbulence reduction screen (\(K=1\)) would require a power increase of \(98\text{kW}\).

One obvious method of reducing losses across flow conditioning devices would therefore be to locate them in a region of large cross sectional area and hence lower velocity. Methods for achieving such area changes around a wind tunnel circuit (principally through the use of diffusers and contractions) have been refined over the past century; spurred on by the desire for a greater number of more efficient wind tunnels, with improving levels of flow quality, to support the rapid advances made in the aeronautical (summarised in Figure 1-1 below) and automotive industries.

Figure 1-1: Wind tunnel hours required for new aircraft development (http://history.nasa.gov)
Today, wind tunnel design has developed to the point where most new tunnels employ a common series of modules, arranged in one of two ways: In a closed circuit (Gottingen) layout, or an open circuit (Eiffel) configuration.

The Gottingen (or closed-return) design was pioneered by Ludwig Prandtl, who commissioned the World’s first closed return tunnel in 1908 (http://history.nasa.gov). A typical Gottingen design is depicted in Figure 1-2 and has the advantage of high circuit efficiency (since the air is recirculated) at the expense of a rise in internal air temperature caused by the energy put into the airstream by the fan being turned into heat through friction. Temperature control is usually achieved by means of a heat exchanger; albeit at the expense of a large pressure drop and significant capital and running costs.

Figure 1-2: Plan view of the 13 x 9 ft Gottingen wind tunnel (Red Bull Racing) in Bedford, UK.

The Eiffel design is a straight-through wind tunnel, which usually has an axial flow fan located downstream of the working section (as shown in Figure 1-3). If a centrifugal fan is used at inlet, the tunnel is commonly termed a ‘blower’.

The Eiffel design was named after its creator, Gustave Eiffel, who, although probably better known for building the Eiffel tower, devoted personal monies towards a private aerodynamics laboratory and is credited with performing the first tests on a complete aircraft in model form (http://history.nasa.gov).
Figure 1-3: Plan view of an Eiffel wind tunnel (Daimler-Benz Aerospace Airbus), in Bremen, Germany.

1.5 PRELIMINARY CLOSED CIRCUIT DESIGN

The feasibility of employing a traditional closed circuit configuration to fulfil the requirements of the design specification outlined in section 1.2 was initially investigated. The requirement to test a 25% scale model of a saloon, effectively set the working section cross-sectional area at approximately 2.5m$^2$, and preliminary investigations revealed that the resulting closed circuit tunnel was cramped and ill-proportioned, and therefore had a greater risk of experiencing secondary flows.

The contraction ratio ($CR$) was limited to 4.5, and although the desired working section turbulence intensity could have been achieved with sufficient screens, the risk of significant non-uniformity was considerable. Increasing the contraction ratio by using a wide-angle diffuser severely reduced the space available for the remaining tunnel modules, and the additional screens needed to prevent this diffuser from separating (Mehta and Bradshaw, 1979), increased costs.

There were also limited funds available for the capital and running costs associated with the heat exchanger needed to remove the energy put into the airstream by the fan. Excluding the heat exchanger would result in a rise in tunnel air temperature during a test, and complicate measurements, such as Hot Wire Anemometry. For these reasons, the feasibility of employing an indraft configuration was investigated. The author and AAE appreciated that a drawback of the indraft design was that the working section static pressure would be below atmospheric, hence making it difficult (yet not impossible) to install boundary layer bleed devices or a moving ground.
1.6 INDRA斐 DESIGN

Removing the third and fourth corners from the closed circuit design (the first corner being the one immediately downstream of the working section) created a novel ‘horseshoe’ indraft configuration, shown in Figures 1-5 and 1-6. The additional space made for a better-proportioned wind tunnel, and effectively created a long diffuser downstream of the working section. Details of the internal circuit geometry are provided in Table 1-1.

It became clear from a search of the literature that no indraft tunnels of comparable layout and size had been built, primarily due to concerns over achieving acceptable test section flow quality as a consequence of variations in ambient winds.

However, the NASA Ames 80ft x 120ft (the World’s largest wind tunnel) was of the indraft type and had been successfully designed to achieve flow quality targets that were not dissimilar to those requested by AAE (with the exception of axial turbulence intensity – being 0.5% for Ames and 0.1% for AAE).

NASA Ames can be turned into an 80ft x 120ft indraft tunnel by closing the vane sets downstream of the second corner and opening those upstream of the fan as shown in Figure 1-4.

Figure 1-4(a): NASA Ames 80ft x 120ft indraft wind tunnel
Due to financial and spatial constraints, the designers (Ross et al, 1986) were required to limit the tunnel to a contraction ratio of 5:1, with only a short duct located upstream. From their assessment of previous work on indraft tunnels, they concluded that:

“...conventional indraft designs could not provide the desired test section flow quality (less than 0.5% turbulence intensity; angle variations less than +/- 0.5deg; and dynamic pressure variations across the test section less than +/-0.5% of the mean)...”

Through tests on a 1/15 scale model of the Ames tunnel, Ross et al (1986) found that the use of a vertical cascade at the inlet plane, with a high loss screen ($K_L = 1.6$ as defined in section 2.2.3) placed immediately downstream of the cascade, produced the desired flow quality in the presence of ambient winds up to 15% of the test section velocity.

Stronger ambient winds led to an increase of axial turbulence intensity more so than a degradation of dynamic pressure uniformity. The authors found that placing the screen immediately downstream of the cascade maximised the mixing length prior to the contraction.
The function of the cascade was to:

- Redistribute the entrained flow so that the loss incurred by the combination of cascade and screen is constant across the width of the inlet.

The function of the high loss screen was to:

- Reduce the turbulence caused by separation at the rear of the vanes.
- Reduce the scale of turbulence entering the contraction thereby allowing it to decay faster.
- Decrease sensitivity of the tunnel to unsteady ambient winds.

The use of horizontal splitters between the vertical cascades effectively formed a large honeycomb at inlet, which was aimed at providing improved isolation from ambient winds.

Unfortunately, no indraft tunnel of the layout and scale presented in this Thesis had previously been built. There was consequently no published data on the likely working section flow uniformity, turbulence intensity and balance repeatability, nor on the impact on these from ambient winds. In addition, a boiler house located 5.3m opposite the intake and exit, was likely to create a recirculation zone for the jet exiting the final diffuser.

Since the extent of these issues could not be quantified prior to running the tunnel, and following the results from the Ames designers, allowance was made in the pressure loss calculations for the future inclusion of inlet louvres/cascades and exit guide vanes (to deflect exit flow away from the inlet).

However, in light of the aerodynamic benefits outlined above, a high-loss inlet filter mesh was included in the design of the indraft tunnel presented in this Thesis from the outset.
1.6.1 GENERAL DESCRIPTION OF THE PROPOSED INDRAFT TUNNEL

Figure 1-5: 3D Solid Model
Figure 1-6: General arrangement (dimensions in mm).

Table 1-1: Internal Dimensions at Module Inlet

<table>
<thead>
<tr>
<th>Module</th>
<th>Width (mm)</th>
<th>Height (mm)</th>
<th>Length (L) (mm)</th>
<th>Corner Fillet (mm)</th>
<th>Hydraulic Diameter (D) (mm)</th>
<th>Area (A) (m²)</th>
<th>Area Ratio (A_r)</th>
<th>Hydraulic Radius (r) (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bellmouth</td>
<td>4820</td>
<td>4820</td>
<td>600</td>
<td>-</td>
<td>-</td>
<td>23.232</td>
<td>1.274</td>
<td>2719</td>
</tr>
<tr>
<td>Settling Chamber</td>
<td>4270</td>
<td>4270</td>
<td>1390</td>
<td>-</td>
<td>-</td>
<td>18.233</td>
<td>-</td>
<td>2409</td>
</tr>
<tr>
<td>Contraction</td>
<td>4270</td>
<td>4270</td>
<td>4980</td>
<td>-</td>
<td>-</td>
<td>18.233</td>
<td>7.324</td>
<td>2409</td>
</tr>
<tr>
<td>Working Section</td>
<td>1920</td>
<td>1320</td>
<td>3600</td>
<td>150</td>
<td>-</td>
<td>2.489</td>
<td>-</td>
<td>890</td>
</tr>
<tr>
<td>First Diffuser</td>
<td>1940</td>
<td>1320</td>
<td>4450</td>
<td>150</td>
<td>-</td>
<td>2.516</td>
<td>1.584</td>
<td>895</td>
</tr>
<tr>
<td>First Corner</td>
<td>2340</td>
<td>1780</td>
<td>-</td>
<td>300</td>
<td>-</td>
<td>3.985</td>
<td>-</td>
<td>1126</td>
</tr>
<tr>
<td>Second Diffuser</td>
<td>2340</td>
<td>1780</td>
<td>3250</td>
<td>300</td>
<td>-</td>
<td>3.985</td>
<td>1.384</td>
<td>1126</td>
</tr>
<tr>
<td>Second Corner</td>
<td>2630</td>
<td>2180</td>
<td>-</td>
<td>330</td>
<td>-</td>
<td>5.516</td>
<td>-</td>
<td>1325</td>
</tr>
<tr>
<td>Shape Change</td>
<td>2630</td>
<td>2180</td>
<td>1835</td>
<td>330</td>
<td>-</td>
<td>5.516</td>
<td>-</td>
<td>1325</td>
</tr>
<tr>
<td>Fan</td>
<td>-</td>
<td>-</td>
<td>4640</td>
<td>-</td>
<td>2650</td>
<td>5.515</td>
<td>-</td>
<td>1325</td>
</tr>
<tr>
<td>Final Diffuser</td>
<td>-</td>
<td>-</td>
<td>8195</td>
<td>-</td>
<td>2658</td>
<td>5.549</td>
<td>1.744</td>
<td>1329</td>
</tr>
<tr>
<td>Exit Mesh</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>3510</td>
<td>9.676</td>
<td>-</td>
<td>1755</td>
</tr>
</tbody>
</table>
1.6.2 CONSTRUCTION MATERIALS AND TECHNIQUES

The wind tunnel was constructed from timber modules located on steel supports. Timber was chosen rather than steel as it was considered less likely to ‘drum’ and would be easier to work with when inserting instrumentation.

The final diffuser was manufactured from 5mm thick steel, since a timber fabrication would have required a great deal of surrounding support structure. The constant area shape-change was made from fibreglass on account of its complex geometry. The timber diffusers and corner modules were of box construction, typically manufactured from 25mm marine plywood, with 140mm x 70mm end flanges and similarly dimensioned battening spaced every 500mm. The internal surfaces were sanded, cleaned and then varnished.

The support tables and working section carcass were typically constructed from 150mm x 100mm x 5mm, hollow steel box section for rigidity, with adjustable load feet to allow vertical adjustment. After final positioning the support tables were bolted to the laboratory floor. The working section and fan were isolated from the surrounding structure via flexible couplings.

Barlow (1999) proposes that 10% be added to the final figure of predicted circuit losses allow for leaks and joints. To minimise these losses flexible sealant was used between tunnel modules.
2 DESIGN METHODOLOGY

This Chapter discusses the methodology used to conduct the pragmatic design of each component of the wind tunnel circuit. Since many of the components are not unique to an indraft wind tunnel, the discussion is relevant to the design of low speed wind tunnels in general.

To better visualise the workflows involved, flow charts outlining the ‘key decision’ and ‘detailed design’ phases are available in Appendix A.

At the end of the Chapter, a breakdown of the predicted performance for each component of the tunnel is presented.

2.1 BELLMOUTH INTAKE

An open circuit wind tunnel with the fan placed downstream of the working section requires a means of ingesting air and guiding it downstream into the settling chamber without separation. This may be achieved through the use of a bellmouth.

Massey (1994), argued that if air were simply to be ingested abruptly from a large reservoir (such as the atmosphere in the case of an indraft tunnel) into the sharp-edged entrance of an axisymmetric duct (for example the settling chamber of a wind tunnel), the pressure loss coefficient, $K_L$, would tend to a value of 0.5 as the area of the large reservoir approached infinity.

As the stream-tube approaches the inlet of the sharp edged duct, it contracts. Once downstream of the inlet it expands to fill the duct (typifying the formation of a vena contracta). Eddies are formed between the vena contracta and the walls of the duct, which contribute to loss and non-uniformity.

In the case of the settling chamber of a wind tunnel, the cross sectional area is likely to be large and therefore the loss of total pressure, $\Delta H$, will be small. However, it is desirable for these eddies to be minimised prior to flow conditioning devices being located in this region.

Massey (1994) and Hamilton (1929) report that if the inlet to the axisymmetric duct is well rounded to create a bellmouth with a radius of curvature, $r > 0.14D$, where, $D$, is the duct diameter, the stream can follow the boundary without separating from it, thus eliminating the vena contracta, and the loss coefficient across the bellmouth is consequently much reduced. Massey suggests that $K_L$ is negligible in this instance, although it is unclear as to whether this is Reynolds dependent.
The majority of sources deal solely with circular-lipped bellmouths, but ESDU datasheet 80037 provides empirically derived data for incoming flow angles of up to 25 degrees, which shows that elliptic bellmouths are better able to ensure full flow attachment than circular lips. In tests on elliptic bellmouths with axis ratios between 2 to 5, ESDU datasheet 80037 reports no advantage in having axis ratios greater than 2 (where axis ratio is defined as the ratio between the semi major and semi minor axes of the ellipse). This is helpful since a small axis ratio enables the length occupied by the bellmouth to be minimised.

To ease construction, an elliptic bellmouth with a semi major axis of 600mm and semi minor axis of 275mm was chosen, resulting in an axis ratio of 2.2. The bellmouth was constructed from 18mm thick plywood formers overlaid with a 3mm thick plywood skin and then varnished to protect it from weathering.


2.2 SETTLING CHAMBER

The function of the settling chamber is to produce a parallel, spatially uniform, steady stream of air. In general, honeycomb is used to straighten the flow and suppress lateral turbulence, whilst screens are used to reduce spatial variations in the axial velocity and reduce axial turbulence.

Whilst screens do act to turn the flow normal to the plane of the screen (Barlow, 1999), they are not as effective a flow straightener as honeycomb. Consequently, the two are often used in combination, with the screens being placed downstream of the honeycomb.

Whilst Barlow (1999) describes the mechanisms used to achieve the physical processes described above as being one area that “…continues to resist fully rational quantitative design treatment…” some empirical design guidelines do exist. These are summarised below and their application to the wind tunnel presented in this Thesis is described.

2.2.1 INLET FILTER MESH

As described in section 1.5, Ross et al (1986) utilised a high loss screen ($K_L = 1.6$ as defined in section 2.2.3) located immediately downstream of a vane cascade (to maximise the mixing length downstream) to good effect in the NASA Ames 80ft x 120ft indraft wind tunnel. The authors found that such a screen:

- Reduced the turbulence caused by separation at the rear of the vanes.
- Reduced the scale of turbulence entering the contraction thereby allowing it to decay faster.
- Decreased sensitivity of the tunnel to unsteady ambient winds.

In light of these aerodynamic benefits and notwithstanding the associated loss, the tunnel presented in this Thesis employed an inlet filter mesh with twice the pressure loss coefficient used by Ross et al. This was aimed at significantly damping ambient gusts blowing into the intake (hence possibly eliminating the need for inlet cascades) and to prevent ingestion of particles, which might otherwise damage hot wires in the working section.

AAE were not concerned by clogging of the filter mesh since a 5.3m wide service road in front of the intake provided easy access for cleaning.

A high-loss ($K_L = 3.25$), single-piece filter mesh with wire diameter, $D_w = 0.31\text{mm}$, and aperture, $\psi = 0.52\text{mm}$, was therefore located immediately downstream of the bellmouth.
Additionally, since Groth and Johansson (1988), had stated that wake coalescence and consequent flow unsteadiness might occur for screens with open areas < 57%, and the open area of the filter mesh was only 39%, a mixing length of 700mm was allowed between the filter mesh and the downstream honeycomb. This gap was also intended to house cascades or louvers, should initial commissioning show that further damping of ambient winds was necessary. The filter mesh would then be repositioned to the trailing edge of the cascade/louver set and the 700mm mixing length shortened by the chord of the vanes.

Due to the absence of published data for the pressure loss coefficient across such a fine mesh, the equations presented in section 2.2.3 were used to predict the loss coefficient. Whilst it is acknowledged that the actual loss coefficient may differ due to likely wake coalescence, the aerodynamic benefits offered by such a mesh were believed to outweigh the pressure loss penalty.
2.2.2 HONEYCOMB SELECTION

The incoming airstream may be described as a fluctuating velocity superimposed on a mean. Therefore, as the stream passes through the honeycomb, the honeycomb effectively acts as a low pass filter, insofar as the small diameter of each cell, suppresses lateral velocity fluctuations, and to a lesser extent axial fluctuations, which are damped out more effectively by screens. For this reason, the honeycomb cell diameter ought to be less than the integral length scales of the eddies present immediately upstream.

However, since the upstream integral length scales were unknown (but anticipated to be small as a result of the inlet filter mesh), a cell hydraulic diameter, $D_{HC} = 9.53\text{mm}$ was chosen as this had been used to good effect in previous wind tunnels (Wolf, 1993). Producing fully developed exit flow (merging of wall boundary layers so that viscous effects permeate throughout the honeycomb and the inviscid core ceases to exist) was not desired, since such long honeycomb would generate axial turbulence (due to formation of eddies within the developing turbulent boundary layer) which would then have to be removed by the screens.

The honeycomb was located at the end of the 700mm mixing length allowed downstream of the inlet filter mesh (discussed in section 2.2.1). Empirical data from Loehrke and Nagib (1972), and Schieman (1981) suggested that a cell length to diameter ratio ($L_{HC}/D_{HC}$) of between 6 to 12 (12 being for fully developed flow) be employed, although there is no real data to support where it should specifically be within this range.

In order for the turbulence reduction screens (discussed in section 2.2.4) to be mounted to standard commercially available C-Section steel (Figure 2-1 and 2-2), a honeycomb $L_{HC}/D_{HC}$ of 8.7 was employed.

A guillotine was constructed to produce a smooth edge when cutting the blocks of honeycomb to size. A contact adhesive was then applied to this smooth edge to bond the blocks of honeycomb together. Wires were employed to fasten the honeycomb to the first turbulence reduction screen. This method minimised the blockage that could otherwise be produced at joints between blocks, and was an order of magnitude less expensive than the traditional method of applying sheet metal banding around the honeycomb and then riveting each block together.
2.2.3 LOSS AND TURBULENCE REDUCTION ACROSS MESH SCREENS

Prandtl (1935), reports that whilst a standard reference velocity is not obtained with honeycomb, it can be achieved with mesh screens. To use Prandtl’s own words:

“...The flow resistance of a wire screen is approximately proportional to the square of the speed. Consequently, the resistance in a flow, which locally manifests different speeds, is greater at the points of higher speed than at the points of lower speed.... speedier stream filaments expand upon striking the screen, and slower ones contract, and so the speeds become comparable upon passing through the screen. But this is always at the expense of a great pressure drop...a previously existent moderate velocity difference is approximately lowered to \( \frac{1}{1 + K_L} \) ...”

Wolf (1993), Loehrke and Nagib (1972), and Schieman (1981) define the screen axial turbulence reduction factor \( f_u \), as being the ratio between the downstream and upstream root-mean-square (RMS) of the instantaneous velocity, \( u \):

\[
f_u = \frac{\sqrt{\overline{u_2^2}}}{\sqrt{\overline{u_1^2}}} = \left( \frac{1}{(1 + K_L)} \right)^n
\]

It may therefore be seen that multiple screens (\( n>1 \)) are more effective than a single screen of the same overall pressure drop coefficient, \( K_L \). As an example, a single screen with \( K_L = 2 \), results in axial turbulence being reduced to 1/3 of the inlet value, whereas two screens each with \( K_L = 1 \), results in axial turbulence being reduced to 1/4 of the inlet value.

The screen pressure drop coefficient \( K_L \) is defined by Schieman as:

\[
K_L = K_o + \frac{55.2}{Re_w} \\
K_o = \left( \frac{1 - 0.95\beta}{0.95\beta} \right)^2 \\
\beta = \left( \frac{\psi^2}{(\psi + D_w)^2} \right)
\]
Where $K_o$ is the pressure loss coefficient due to screen open area, $Re_W$ is Reynolds number based on wire diameter, $\beta$ is fractional screen open area, $\psi$ is the aperture, and $D_w$ is wire diameter. Schieman warns that although screen pressure loss coefficients calculated using these empirically derived equations show the correct trends with velocity and screen physical characteristics, they differ from those measured experimentally by up to 50% for screens with $57% < \beta < 64%$.

Screens may be designed to operate in either the subcritical or supercritical regime. In the subcritical regime, defined as $Re_W < 40$, turbulence reduces monotonically across the screen (Groth and Johansson, 1988). However, to produce such low wire Reynolds Numbers at the velocities typically seen in the settling chambers of low speed wind tunnel requires very fine wires, which are brittle, expensive and prone to clogging. Since $Re_W$ is so low, the pressure loss coefficient across a subcritical screen is very high. Alternatively, thicker wire diameters may be used, making the screen operate in the supercritical regime. In this condition the screen itself generates turbulence due to vortex shedding off the wires, and the turbulence intensity immediately downstream of the screen is actually higher than that observed upstream. However, this screen-generated turbulence is of very small scale and has been shown by Groth and Johansson to decay rapidly to below incoming levels by around 25 screen mesh widths ($M = \psi + D_w$). All screens used in this wind tunnel were designed to operate supercritically.

For multiple screens, Groth and Johansson also showed that far field turbulence reduction was largely independent of screen separation, provided the screens were separated at least by the length of the initial decay region ($25M$).
2.2.4 SCREEN SELECTION

The wind tunnel is fitted with two, single-piece turbulence reduction screens designed to operate in the supercritical regime. Single piece screens were chosen to avoid nonuniformity and variations in turbulence intensity that may be produced by welded joints. It also facilitated the screens being tension mounted to eliminate sag.

Loehrke and Nagib (1972), proposed that the suppression of honeycomb generated turbulence is most effectively achieved if the screen is placed immediately downstream of the honeycomb, and if the screen mesh width is a sensible ratio of the honeycomb cell diameter ($3 < \frac{D_{HC}}{M} < 5$).

Consequently, a coarse screen ($D_W = 0.6\text{mm}; \psi = 1.9\text{mm}; \beta = 59\%$) was placed immediately downstream of the honeycomb giving $\frac{D_{HC}}{M} = 3.8$.

To maximise axial turbulence reduction, a finer screen ($D_W = 0.4\text{mm}; \psi = 1.29\text{mm}; \beta = 57\%$) was located 200mm or 80 mesh widths, $M$ (where $M = \psi + D_W$), downstream of the coarse screen, since this was in excess of the minimum of $25M$ decay length recommended by Groth and Johansson (1988), for the small scale turbulence generated by a screen operating in the supercritical regime to decay below incoming levels. A conservative settling length of 118M (200mm) was allowed downstream of the second screen prior to the contraction.

The sides of the settling chamber were designed as two fully welded parallel flange channel (PFC) H-frames, to which the transverse roof and floor members were bolted (Figure 2-1). For installation, the opposing sides of each screen were sandwiched between 2.5mm thick stainless steel hook strips and then TIG welded along the edge (Figure 2-2). Care was taken to prevent warping. The hook strips were hung on PFC tensioning bars located on the sides of the H-Frame, and jacking bolts used to push the bars out and tension the screens.
Figure 2-1: Fully welded settling chamber A-Frame with honeycomb partially installed and the second turbulence screen visible downstream.

Figure 2-2: Screen sandwiched between hooks strips ready to be TIG welded along the edge.
2.3 CONTRACTION

A contraction is used to improve flow uniformity and reduce the turbulence intensity in the working section. Although this may also be achieved through utilising a sufficient number of screens, a contraction also significantly reduces the dynamic loads and total pressure losses on screens and honeycombs placed in the settling chamber.

A review of literature concerning contraction design provided by Su (1991), shows that most of the work is concerned with two dimensional or axisymmetric contractions, whereas most practical contractions are three-dimensional and of rectangular cross section. The reviewer states that:

“…with little regard for design criteria and real conditions in wind tunnel contractions, these solutions are tools of flow analysis rather than methods of contraction design...”

The designer of a three-dimensional wind tunnel contraction of rectangular cross section, has to address the following issues:

- Selection of the contraction ratio (CR).
- Definition of corner fillet growth.
- Determination of the contraction length ($L_C$).
- Decision on aspect ratio change between inlet and exit.
- Definition of the contraction contour.

Since the contraction is so critical to the flow quality in the test section, and because of the level of uncertainty surrounding contraction design, models of a new contraction are often made to check the design (Barlow, 1999). Due to resource limitations, this was not possible for the tunnel presented in this Thesis.
2.3.1 CONTRACTION RATIO

The selection of the contraction ratio is largely driven by the required working section turbulence intensity, $T_u$, defined as:

$$ T_u = \frac{\sqrt{u'^2}}{\bar{u}} $$

Where, $u'$, is the fluctuating component of the local axial velocity and $\bar{u}$, is the time averaged local axial velocity.

Since the RMS velocity $\sqrt{u'^2}$ does not change appreciably during contraction (Pankhurst and Holder, 1968), the turbulence intensity is thereby reduced by the value of $CR$.

The contraction ratio of 7.3 used in this design was the largest possible when coupled to a working section area of 2.489m$^2$, whilst also allowing adequate space for the other tunnel modules.

2.3.2 VELOCITY UNDER-SHOOT AND OVER-SHOOT

Although the bulk flow velocity increases along the length of the contraction, the near-wall velocity in a finite length contraction does not increase monotonically. Since the flow is subsonic the wall curvature at inlet to the contraction has an upstream effect and causes the streamlines in the parallel duct downstream of the last screen to curve (Morel, 1975).

For this reason, the end of the 200mm settling length (118M) downstream of the last turbulence screen was placed 516mm (equating to $0.11D_C$), upstream of the start of curvature of the contraction, where $D_C$ is the hydraulic diameter at inlet to the contraction.

This streamline curvature produces a non-uniform velocity profile within this 516mm parallel section, and the near-wall velocity reaches a local minimum (termed velocity undershoot by Morel) slightly downstream of the inlet of the contraction. In contrast, downstream of the inflection point of the contraction contour, stronger acceleration of the near wall flow (relative to the general acceleration of the bulk flow field), results in a local near-wall velocity maximum (velocity overshoot) near the exit of the contraction. It is this that accounts for the slightly concave velocity profile generally seen in wind tunnel working sections (see Figure 3-2). This is seldom captured by CFD evaluations of contraction design.
2.3.3 CORNER FILLETS

45-degree corner fillets were used to reduce secondary flows caused by interaction between the wall and roof boundary layers and also to house lights. The fillets grow from nothing at inlet to the contraction, to their final value of 150mm x 150mm at exit, according to the following cubic relation put forward by Tinkler and Fritz (1986).

\[
[\text{Local Fillet Area}] = \left(3\left(\frac{x}{L_C}\right)^2 - 2\left(\frac{x}{L_C}\right)^3\right) [\text{Fillet Area at Exit}]
\]

Where \( x \) is the local axial distance measured from the inlet of the contraction, and \( L_C \) is the contraction length.

2.3.4 LENGTH

Due to a greater risk of separation at the corners, a rectangular contraction needs to be longer than the equivalent axisymmetric one, since increasing the length of the contraction alleviates the regions of adverse pressure gradient. A long contraction, however, results in an excessively thick boundary layer at exit and less available space for other sections of the wind tunnel.

In light of the near-wall velocity overshoot at the exit of the contraction (described in section 2.3.2), a length of parallel duct equivalent to one hydraulic duct radius (R), was allowed between the end of curvature of the contraction and the inlet of the working section (as advised by Mehta and Bradshaw 1979).

The numerical investigation conducted by Su (1991) into the effect of the relative length \( L_c/D_c \) of a three-dimensional contraction of rectangular cross section, on exit flow uniformity, velocity overshoot and corner velocity distribution, showed that an \( L_c/D_c \approx 1 \), was a good design compromise, which was free from separation for a CR of 9. An \( L_c/D_c = 1.03 \) was used in this design giving \( L_c = 4980 \text{mm} \) as shown in Figure 2-3.
Station A on Figure 2-3 is located 154mm upstream of the centre of the working section (and the balance point of resolution). Station A, is the plane in which boundary layer, velocity uniformity and turbulence measurements were performed (see relevant sections in Chapter 3).

2.3.5 ASPECT RATIO

Su (1991) also showed that the rule-of-thumb of maintaining aspect ratio similarity along the length of the contraction in order to prevent the flow from distorting was not necessary. Su proposed that a square cross section was a good choice for the settling chamber no matter what the geometry of the test section.

An inlet of internal dimensions 4270mm x 4270mm was therefore used in this design, since restrictions in the available width for the wind tunnel meant that having a square settling chamber allowed a larger diffuser to be used downstream of the fan (see Figure 1-6).
2.3.6 CONTRACTION CONTOUR

Su (1991) presented the following equations for generating contours of matched curves:

\[
(J - J_2)(J_1 - J_2) = 1 - \left[ \left( \frac{x/L_C}{X} \right)^n \right]^{n-1} \quad 0 \leq x/L_C \leq X
\]

\[
(J - J_1)(J_2 - J_1) = 1 - \left[ \left( \frac{1 - (x/L_C)}{1 - X} \right)^n \right]^{n-1} \quad X \leq x/L_C \leq 1
\]

Where, \( J \), is the local height or width of contraction measured from the centreline; \( X \), is the matchpoint between the two equations; and \( n \), is the power used to produce the required radius of curvature.

Downie et al (1984) advocated the use of a large radius of curvature towards the inlet and exit of a contraction to alleviate adverse pressure gradients. A consequence of this being that the central portion of the contraction then has a relatively more aggressive inflection. Graphical illustrations of the growth of the boundary layer along such a contour are available from Chmielewski (1974).

As discussed in section 2.3.2, streamline curvature at the contraction inlet causes a thickening of the boundary layer, which reaches its maximum thickness at some point just downstream of the inlet. More aggressive wall curvature then causes the near wall flow to accelerate rapidly up to the point of inflection (or matchpoint, \( X \) shown in Figure 2-3), at which point the boundary layer attains its minimum thickness, and in some cases relaminarises. This acceleration produces a near-wall velocity overshoot (relative to the bulk velocity) at the exit of the contraction.

Since the risk of boundary layer separation is greatest at the inlet, and exit uniformity is critical to models under test, these may be set as the two most important criteria by which to evaluate potential contours. Although the philosophy of having a short, steep, central section with large radii of curvature at each end holds true, results from a numerical investigation by Su (1991), show that combining a slightly lower power contour (smaller radius of curvature) upstream of the matchpoint with a higher power contour downstream, reduces the velocity undershoot at inlet whilst providing a parallel duct at the end of the contraction. The latter allowing the velocity overshoot to decay. An upstream power, \( n \), of 4 and a downstream power of 6 were therefore used to generate contours for this design. The resultant parallel duct was equal to one working section hydraulic radius (as advised by Mehta and Bradshaw, 1979).
Matching the two different contours resulted in a discontinuity at the match point, which was eliminated by calculating the local gradients of both contours at either side of the matchpoint and making cuts at the two points where the gradients matched. A cubic spline was then fitted between these two points to produce a smooth transition.

Su and Downie et al (1984) report no advantage in having different matchpoint locations for the vertical and horizontal contours, and so a matchpoint of $X = 0.41L_C$ was selected since Su had shown this to produce good exit flow uniformity. Despite the transition from square to rectangular geometry, the same values of $n$, were used for the roof/floor and sidewall contours.

The contraction was manufactured in four sections (Figure 2-4), which were then bolted together on site. Due to height restrictions in the laboratory, the contraction was assembled with its downstream end placed on the floor. Once bolted together, a lifting frame was constructed and the contraction tipped into place as shown in Figure 2-5. Each section comprised 18mm thick plywood formers overlaid with a layer of 3mm thick plywood to form the internal skin. This method cost approximately 1/3 that of a more traditional fibreglass construction, whilst making it much easier to achieve a smooth profile. The external ribbing provided useful storage!

Figure 2-4: One side of the contraction
2.3.7 CONTRACTION LOSS COEFFICIENT

Barlow et al (1999) suggests that losses in the contraction may be considered to be from friction only, and presents the following equation from Wattendorf (1938), the contraction loss coefficient:

\[
K_l = 0.32 \left( \frac{f_1 + f_2}{2} \right) \left( \frac{L_C}{D_{WS}} \right) \left( \frac{q_{WS}}{q_1} \right)
\]

Where \( f \) is the friction factor and WS denotes working section data. For smooth pipes at high Reynolds number, Shames (1992) and Barlow give an iterative solution algorithm for the Prandtl universal law of friction relating the Reynolds number and the friction factor:

\[
f = \left[ 2 \log_{10} \left( R_e \sqrt{f} \right) - 0.8 \right]^2
\]

A starting value of \( f = 1 \) will lead to convergence to four significant figures or better within four to six iterations. Since the loss in the contraction represents a small proportion of the overall circuit loss, errors in estimating contraction loss coefficient are much less important than errors in estimating losses in the high velocity section of the circuit.
2.4 WORKING SECTION

The need to accommodate a 25% scale model of an automotive saloon at 5% blockage based on model frontal area was specified in the design brief. A full-scale car with a frontal area of around \(2 \text{m}^2\), gave a model frontal area of around \(0.125 \text{m}^2\) and consequently a working section area of around \(2.5 \text{m}^2\).

Figure 2-6 shows a 25% scale automotive model in the working section. The model is mounted to an underfloor balance through the four pairs of off-centric discs in the balance turntable, which allow adjustment for wheelbase and track. Care was taken to ensure the absence of forward steps between the turntable and the tunnel floor.

During operation the working section is subjected to a static pressure that is lower than atmospheric and so air will leak into the working section through any gaps that exist. Care was therefore taken to ensure adequate sealing between the off-centric discs and the turntable and around the interface between the turntable and the tunnel floor.

The fully-welded steel working section carcass was bolted to the 9-inch thick concrete laboratory floor and was isolated from the wind tunnel by means of flexible couplings at either end to minimise the transmission of structure borne vibration.
2.4.1 ASPECT RATIO

Originally, it was envisaged that aeronautical models would span the width of the tunnel. Data from Barlow et al (1999) had shown that for a small wing, the wall correction factor was a minimum for a width to height ratio of about 1.5 (which is why many tunnels have been built in the 7x10ft, 8x12ft and 9x13ft size range).

Such an aspect ratio also matched that of a typical saloon and a value of 1.46:1 was ultimately chosen for convenience of geometry, giving inlet internal dimensions of 1920mm (width) x 1320mm (height).

Due to the expense of procuring an elevation frame for the underfloor balance shown in Figure 2-7, it was decided to test aeronautical models in the half model configuration as shown in Figure 2-8. The 1320mm height enabled three-dimensional tests to be performed on an aerofoil with a span of 1000mm.
Figure 2-7: Underfloor, 6-component, virtual-centre balance

Figure 2-8: Aeronautical half model mounted in the working section
2.4.2 HORIZONTAL BUOYANCY

Boundary layer growth along the walls of the working section produces an increase in aerodynamic blockage due to the increase in displacement thickness. The cross-section was therefore increased from 1920mm (width) x 1320mm (height) at inlet, to 1940mm (width) x 1320mm (height) at exit to account for the growth in displacement thickness. An axial static pressure gradient of ~0.002Cp/m is widely deemed to be an acceptable limit.

The aim being to help prevent a longitudinal drop in static pressure which would otherwise have the effect of sucking the model downstream, producing an artificial increase in drag. The easiest means of implementing this area increase was to put the taper in the sidewalls rather than the roof and floor (Barlow, 2000). 150mm x 150mm fillets were used in each corner to minimise secondary flows and are also employed to house working section lighting.

2.4.3 LENGTH

Bradshaw (1968), Barlow et al (1999), and Pankhurst and Holder (1968), recommend that the length of the working section be between 2-3 times its hydraulic diameter (D), to allow model wakes to mix out prior to the first diffuser. Using the lower limit of this range (due to space constraints) and a cross sectional area of 2.489m², gave a working section length in the region of 3.6m.

Garry et al (1994) carried out an experimental investigation into the effect on aerodynamic drag of the longitudinal position of an automotive model in a wind tunnel working section using three different wind tunnels. These investigators showed that large changes in drag occurred when the model was closer than twice the square root of its base area from the end of the working section. They attribute this effect to base pressure changes in the proximity of the downstream diffuser.

Applying the findings of Garry et al to a typical automotive model with an overall length of 1m and width of 0.5m, gives a minimum separation distance of around 1.4m. Due to convenience of geometry the balance point of resolution was placed 1.695m downstream of the working section inlet (or 105mm upstream of the working section centreline). Therefore, for a 3.6m long working section shown in Figure 2-3 and a 1m long model, a separation of 1.405m was achieved between the end of the model and the start of the first diffuser (~0.8D).


2.4.4 LOSS COEFFICIENT

As described by Barlow et al (1999), flow losses in constant area pipes are commonly expressed as:

\[ K_L = f \frac{L}{D} \]

The friction factor, \( f \), may be calculated as indicated in section 2.3.7.

2.4.5 SAFETY SCREEN

A safety screen of \( D_W = 2.5 \text{mm} \), and \( \psi = 23 \text{mm} \), was located at the end of the working section. Due to the high porosity of the mesh \( \beta = 81.4\% \), the pressure loss coefficient was calculated by treating each wire as an infinitely long cylinder with a drag coefficient \( (C_D) \) of 1, and neglecting interference effects between wires.

It was recognized that locating the safety screen at the downstream end of the working section would result in a significant pressure drop, but it was done to protect the remainder of the tunnel circuit.

The value of doing so was proved during fan commissioning when the difference in static pressure between the working section and the surrounding laboratory resulted in the temporary working section floor (fitted for the inaugural run) being sucked up. Had the safety mesh not been there, significant damage would have resulted to the first corner turning vanes.
2.5 DIFFUSERS

Since local power losses around the wind tunnel circuit are proportional to the cube of the local velocity, diffusers are used to convert the kinetic energy of the airstream into pressure energy as efficiently as possible (Bradshaw ad Pankhurst, 1964). As explained by Barlow et al (1999), diffusers are sensitive to the quality of their near wall flow. Intermittent and steady flow separations within the diffuser must be avoided as they can cause vibrations, oscillating fan loading and increased losses in downstream tunnel components. Since the flow is subsonic, diffuser separations can have upstream effects and appear as oscillations in test section velocities.

The flow through a wind tunnel diffuser depends on its geometric parameters shown in Figure 2-9, such as its length, \( L \), area ratio, \( A_r \), equivalent conical included angle, \( \theta \), cross-sectional shape, wall contour and throat curvature; and aerodynamic factors such as inlet Reynolds number, inlet blockage, inlet non-uniformity and distortion, and inlet swirl.

![Figure 2-9: Conical diffuser geometry](image)

The equivalent conical included angle, \( \theta \), is defined as:

\[
\theta = \arctan \left( \frac{r_2 - r_1}{L} \right).
\]

Although the equivalent conical included angle, \( \theta \), is commonly used as a geometric parameter for wind tunnel diffusers, most wind tunnels employ low aspect ratio, plane-walled diffusers rather than conical designs, for reasons of cost, ease of construction and simplicity of mating to the end of plane-
walled working sections. However, plane-walled diffusers used in wind tunnels often feature 45° corner fillets to minimise secondary flows associated with interactions between the roof and wall boundary layers. These fillets grow along the length of the diffuser to change the internal geometry close to that of an octagon at inlet to the constant area shape change upstream of the fan. Wind tunnel diffusers are therefore essentially a hybrid between plane-walled and conical designs.

In their tests comparing square-sectioned and conical diffusers, Gibson (1911) and Dolan and Runstadler (1973), showed that the performance of the two types were similar up to $2\theta \approx 8^\circ$. Their results and the use of corner fillets would suggest that treating wind tunnel diffusers as equivalent conical diffusers (up to $2\theta \approx 8^\circ$) is justified. This enables empirical performance charts developed for conical diffusers to be used in the design process. Engineering Sciences Data Units (ESDU) have collated experimental data available in the wider literature to generate conical diffuser performance charts presented in datasheets 76027 and 73024. Of these charts, those relevant to the diffusers used in this wind tunnel are reproduced in Figures 2-10 and 2-11.

The parameters used to quantify diffuser performance are the total pressure loss coefficient, $K_L$, and the static pressure recovery coefficient, $C_{Pr}$, defined as:

$$C_{Pr} = \frac{p_2 - p_1}{\frac{1}{2} \rho U^2}$$

Where $p$ is the static pressure. As described by Massey (1994), the loss of head that occurs in a diffuser depends on its total included angle and area ratio. One contribution to the loss is made by pipe friction, which decreases as $2\theta$ increases, since for a given area ratio, a larger divergence angle gives a smaller length. For all but the smallest angles, however, energy is also dissipated by eddies caused by the separation of the flow from the walls. This loss increases with $2\theta$. Massey states that for a conical diffuser with a smooth surface, the total included angle for which the sum of the two types of loss is a minimum is approximately $2\theta \approx 6^\circ$. This is supported by empirical data summarised in ESDU datasheet 76027, which also shows that maximum $C_{Pr}$ for conical diffusers is typically achieved at $2\theta \approx 6^\circ$, for $1.5 < A_r < 6$. The bulk of experimental data concerning conical diffusers suggests that $C_{Pr}$ is substantially independent of inlet Reynolds number (based on inlet hydraulic diameter), provided $Re > 5 \times 10^4$.

Copp (1951) found that the static pressure measured at the wall immediately either side of the junction between the diffuser and upstream duct is below the mean to an extent that depends on the throat shape and wall angle. Copp considered junctions featuring a sharp break and also a wide range
of small to large curvature transition regions for a $2\theta = 10^\circ$ diffuser, and found no influence on $C_p_r$.

Consequently sharp junctions were employed in all diffusers used in this wind tunnel, as they were much less costly to construct than rounded throats. ESDU datasheet 73024 suggests that for $2\theta > 14^\circ$, a rounded throat is desirable to avoid premature separation.

Figure 2-10: Flow regime boundaries for conical diffusers. Source ESDU 73024
(n.b For explanation of symbols refer to Fig 2-9 where $\phi = \theta$ and $R = r$)
Figure 2-11: Pressure recovery coefficient contours for conical diffusers. Source ESDU 73024

(n.b For explanation of symbols refer to Fig 2-9 where $\phi = \theta$ and $R = r$)
2.5.1 FIRST DIFFUSER

The first diffuser was designed with an area ratio, $A_r$, of 1.584, an $L/r_1$ of 5.0, and a $2\theta$ of 6° to ensure it operated within the attached flow boundary indicated on Figure 2-10. Although Figure 2-10 might suggest that $2\theta \approx 7^\circ$ could be employed whilst maintaining attached flow, it ought to be noted that these performance charts were derived for diffusers operating with undistorted inlet flow. It is common practice to be conservative in the choice of wall angles for wind tunnel diffusers since diffusers are known to be sensitive to the quality of their near-wall flow and automotive models placed on the floor of the working section will produce near-wall, flow distortions at inlet to the diffuser. Unfortunately, there is an absence of published data by which the performance of conical diffusers with such near-wall, asymmetric inlet flow distortions may be quantified. A roof/floor $2\theta$ of 5.9° and a sidewall $2\theta$ of 5.2° was employed, with greater divergence in the vertical plane to reduce the aspect ratio of the internal circuit and thereby prevent the local wall angles in the constant-area shape-change module from being excessive. At a working section velocity of 40m/s, the inlet Reynolds number based on hydraulic diameter is $4.9\times10^6$. The static pressure recovery coefficient, $C_p$, was derived from Figure 2-11 as 0.483. The total pressure loss coefficient, $K_L$, for diffusers followed by a downstream duct is defined in ESDU datasheet 73024 as:

$$K_L = \alpha_i - \left(\frac{\alpha_2}{A_r^2}\right) - C_p$$

Where $\alpha_i$ is the kinetic energy coefficient, which is a measure of the degree of flow non-uniformity. The latter is defined as:

$$\alpha = \frac{1}{AU_3^3} \int_A u^3 dA$$

Given the wide variety of models that are likely to be tested, a very conservative value of $\alpha_i = 1.2$ was used for the first diffuser when calculating its contribution to the overall stagnation pressure rise required from the fan, particularly as there was no risk to maximum tunnel speed associated in doing so. The equation for diffuser pressure loss coefficient might suggest that a high value of $\alpha_2$ gives a low total pressure loss, but in practice diffusers producing highly distorted flows also give low static pressure recovery. A high value of $\alpha_2$ is also undesirable because it implies a very non-uniform exit flow, which is likely to have an adverse effect on the performance of the next module in the wind tunnel.
2.5.2 SECOND DIFFUSER

The second diffuser was designed with an area ratio of 1.384, an \( L/r_1 \) of 2.9, and a \( 2\theta \) of 7.0° to ensure it operated within the attached flow boundary indicated on Figure 2-10. It features a roof/floor \( 2\theta \) of 7.0° and a sidewall \( 2\theta \) of 5.1 degrees. At a working section velocity of 40m/s, inlet Reynolds number based on hydraulic diameter is 3.9x10^6. Its \( C_{pr} \) was derived from Figure 2-11 as 0.400. Although the flow may still be non-uniform, an \( \alpha_1 = 1.1 \) was used when calculating its contribution to the overall stagnation pressure rise required from the fan since the wake produced by a model on the floor of the working section would have had time and space to mix out.

2.5.3 FINAL DIFFUSER

A conical rather than plane-walled diffuser is located downstream of the fan since it is better able to take advantage of the 5° of swirl generated by the latter and also because it eliminated the need for a second shape change module. McDonald et al (1971) and Senoo et al (1978) both report that inlet swirl improves the \( C_{pr} \) of conical diffusers by energising the boundary layer and reducing boundary layer displacement thickness. Their experimental results suggest that swirl is optimum when the mean swirl angle, \( \phi \), is equal to the total included angle, \( 2\theta \), and that in these conditions, an increase in \( C_{pr} \) of around 15% may be achieved. Senoo, however, warns that generating yet more swirl produces a low axial velocity core near the centre of the diffuser (termed a Rankin Vortex). The fan was designed with straightener vanes that produced 5° of exit swirl.

The diffuser was designed with an area ratio of 1.744 and a \( 2\theta \) of 6.0° to ensure the diffuser operated just within the attached flow boundary shown on Figure 2-10. At a working section velocity of 40m/s, inlet Reynolds number based on hydraulic diameter is 3.3x10^6. Its \( C_{pr} \), was derived from Figure 2-11 as 0.538 and an \( \alpha_1=1.1 \) was adopted when calculating its contribution to the overall stagnation pressure rise required from the fan. This high value of \( \alpha_1 \) was chosen to account for non-uniformity downstream of the fan.

As the kinetic energy contained within the jet at exit of the final diffuser is not used in a downstream duct, the total pressure loss coefficient, \( K_L \), is defined in ESDU datasheet 76027 as:

\[
K_L = \alpha_1 - C_{pr}
\]

An exit mesh identical to the working section safety screen was placed at the end of the final diffuser to prevent foreign object ingress. Its loss coefficient may be calculated as described in section 2.4.5.
2.6 TURNING VANES

The tunnel includes two sets of constant area, 90° corners, to guide the flow around the 180° bend between the working section and the fan. As reported by Idelchik and Fried (1989) and Krober (1932), the pressure loss coefficient, $K_L$, across such a corner would be greater than unity if no turning vanes were employed.

Previous test rig data from Klein et al (1930), Collar (1936), Salter (1946) and Winter (1947) have shown that thin ¼ circle turning vanes with a space to chord ratio ($s/c$) of between 0.20-0.25, produced $K_L$ of between 0.12 - 0.20. However, these sources focused on determining $K_L$ rather than quantifying flow quality downstream of the vanes. Although $K_L$ is important in terms of achieving a high tunnel energy ratio, it is perhaps secondary to downstream flow quality since the stream exiting the corners of a typical wind tunnel either immediately enters a diffuser, where performance is directly related to the quality of its near wall flow; the fan, whose manufacturer will probably only guarantee pressure rise provided the fan ingests a uniform airstream; or the settling chamber, that has to condition the flow for the working section. Since the flow is subsonic, perturbations caused by poor vane design may be seen upstream in the working section.

The use of modern aerofoil sections both in constant area and expanding bends have been proposed by Sahlin and Johansson (1991) and Lindgren et al (1998) respectively, to further improve efficiency over traditional ¼ circle vanes. Although these vanes have been shown (in test rigs) to reduce $K_L$ to as low as 0.04, they have yet to be proved sufficiently insensitive to inlet flow angularity, which might be produced by wakes from models placed in the wind tunnel working section. As far as the author is aware, such vanes have also yet to be tested in wind tunnels, and based on cost estimates obtained for the wind tunnel described in this Thesis, were an order of magnitude more expensive to manufacture than ¼ circle vanes. Although these modern aerofoils may offer lower $K_L$ and improvements in downstream uniformity, the additional cost was felt to be unjustified since the two corners only contribute a total of around 11% (see Table 2-3) to the overall loss of the wind tunnel.

Although test rig $K_L$ data exists for ¼ circle vanes and such vanes have been used widely in wind tunnels, there is no information on how well test rig data compares to in situ wind tunnel measurements in terms of full corner loss coefficient and flow uniformity downstream of the cascade. These issues will be addressed in Chapter 4. However given the lack of published information on flow uniformity downstream of a vane cascade, an experimental investigation was performed in a constant area vane test module to obtain pressure loss and velocity profile data for vanes that were ultimately used in the wind tunnel presented in this Thesis.
2.6.1 VANE TEST MODULE

The span of the test vanes was limited to 450mm, since the test module had to mate to the end of the working section of a blower wind tunnel measuring 450mm x 450mm. Tests were performed at the same chord Reynolds number ($Re_c$) as would be seen in the second corner of the wind tunnel presented in this Thesis. This corner was chosen as the authors wished to evaluate the likely flow quality upstream of the fan, since the fan manufacturers would only guarantee an overall pressure rise provided the fan ingested a reasonably uniform airstream.

As shown in Figure 2-12, the vanes were designed with an angle of attack of $4^\circ$. Since a radial force must be present to turn the flow around the corner this implies a higher pressure on the outer bend than on the inner. This pressure gradient results in streamline curvature upstream of the vanes. The $4^\circ$ angle of attack aligns the leading edge of the vanes with the incoming streamlines. This is supported by an investigation by Idelchik and Fried (1989) into the effect of leading edge angle of attack on the pressure loss coefficient across the cascade, which shows that $K_L$ is a minimum if the vanes are set at around $4^\circ$.

Figure 2-12: Turning vanes used in the test module and wind tunnel.
¼ circle vanes have been shown to achieve $K_L$ of 0.12 (Salter, 1946), and 0.14 (Patterson, 1936) when tested in rigs at chord Reynolds numbers of $2 \times 10^5$, and $4 \times 10^4$ respectively. Since $K_L$ reduces with increasing chord Reynolds number, it was decided to employ vanes of large chord to generate $Re_c$ in excess of $10^5$ at the velocities seen in the second corner of the wind tunnel. With the wind tunnel running at a working section velocity of 40m/s, the velocity in the second corner is 18m/s.

The test vanes were constructed from 1.5mm thick rolled aluminium (3mm for the eventual wind tunnel vanes) with an inside radius, $r_i$, of 245mm. To encourage the flow to leave the vanes axially, a trailing edge extension ($TE$) of 165mm (approximately 1/3 of the chord of the vanes) was employed. This geometry resulted in a chord, $c$, of 468.5mm, which gave an $Re_c$ of $5.74 \times 10^5$ at 18m/s. Aside from the aerodynamic benefit, vanes of large chord are more rigid and therefore would not require additional stiffeners when installed vertically in the wind tunnel. Increasing the chord also reduces the number of vanes and hence cost, for the same $s/c$.

The vane test module is shown in Figures 2-13 and 2-14. With 3 vanes installed an $s/c$ of 0.237 was achieved, which was as close as it was possible to get (due to the requirement for an integer number of vanes) to the $s/c$ of 0.25 suggested by most sources for minimum $K_L$. However, since Salter (1946) had proposed that greater stability of the near wall stream may be obtained by employing an $s/c$ of 0.20, a fourth vane was added to the test module (producing an $s/c$ of 0.190) to enable this hypothesis to be further investigated.

Figure 2-13: Vane test module with 4 vanes installed ($s/c = 0.190$).
The pressure loss coefficient, $K_L$, was determined by measuring the static pressure loss across the cascade and dividing this by the dynamic pressure in the working section of the blower wind tunnel to which the test module was mated. Static pressure loss across the cascade was measured by means of the averaged reading from a ring of static tappings located at positions A and B on Figure 2-13. A and B are respectively located 25mm upstream and 25mm downstream of the inner bend of the cascade. An additional ring of tappings was provided at C.

Each ring comprised 4 tappings located at the midpoint on each side of the module. The tappings were constructed from brass tubing with an outside diameter of 1.6mm and an inside diameter of 0.85mm. As recommended by Shaw (1960), to improve data quality the tappings were squared off, de-burred and mounted flush with the wetted surface. Epoxy resin was applied around the non-wetted junction between the tapping and the module surface to prevent leakage.

Velocity profiles were measured perpendicular to the test module walls at locations A, B and C using a pitot probe referenced to the averaged reading from the relevant ring of local wall static tappings. The flow was ejected to atmosphere 600mm downstream of location C. The head of the pitot was placed in the same plane as the tappings. Location C was positioned 875mm downstream of the cascade, since this was approximately halfway between the second corner and the fan in the wind tunnel. The authors anticipated that velocity profiles determined at location C, would therefore provide valuable information on the quality of the near wall flow as it developed far downstream of the cascade and approached the fan.

The pitot probe was constructed from tubing with an outside diameter of 3.3mm and an inside diameter of 2.5mm. The probe was traversed across the mid span of the vanes in 10mm increments.
as the flow was anticipated to be two-dimensional in this region and enable correlations to be made with future mid span traverses in the actual wind tunnel.

For both pressure loss and velocity profile measurements, the differential pressure transducers used were accurate to 0.25% of reading and had an output of +/- 10V. A 12-bit data acquisition card was used in conjunction with software written in National Instruments LabVIEW to acquire data. Data was sampled at 1kHz for 10 seconds and averaged over 10 repeats to promote confidence in the mean. A 2-minute settling time was allowed between readings since the tubing used was necessarily long.

2.6.1.1 VANE TEST MODULE RESULTS

The velocity profiles presented in Figure 2-15 and 2-16, are non-dimensionalised by dividing the local axial velocity, $u$, measured by the traversed pitot, with the bulk velocity, $U$, measured by a pitot static in the working section of the blower wind tunnel.

Figure 2-15: Non-dimensionalised velocity profiles at mid span, measured 25mm upstream, and 25mm downstream of the cascade.
Figure 2-15 shows that 25mm upstream of the cascade, the bulk flow exhibits a flat velocity profile with inner and outer wall boundary layers that are free from reversed flow. Boundary layer thickness, \( \delta \), displacement thickness, \( \delta^* \) and momentum thickness, \( \theta \), were calculated using numerical integration (trapezoidal rule), and are presented in Table 2-1. Since \( \frac{u}{U} > 1.0 \) as a result of vane wake blockage, the thicknesses presented in Table 2-1 were calculated by integrating to the local velocity maxima in a region \( \sim Y100 \)mm adjacent to each wall. Although the inner bend boundary layer is thicker than the outer, \( \delta^* \) and \( \theta \) for both bends are commensurate with that expected for a turbulent boundary layer. No downstream data is provided due to the difficulty in confidently determining the true edge of the boundary layer.

Table 2-1: Boundary layer characteristics 25mm upstream of the cascade.

<table>
<thead>
<tr>
<th></th>
<th>Outer Bend</th>
<th>Inner Bend</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \delta ) (mm)</td>
<td>40</td>
<td>60</td>
</tr>
<tr>
<td>( \delta^* ) (mm)</td>
<td>4.5</td>
<td>6.5</td>
</tr>
<tr>
<td>( \theta ) (mm)</td>
<td>3.2</td>
<td>4.9</td>
</tr>
</tbody>
</table>

25mm downstream of the cascade, the observed velocity minima in Figure 2-15, are in line with the trailing edges of the vanes. The vane wakes for both s/c’s tested, show a greater deficit on the suction side of the vane. This is due to a thicker boundary layer. The additional work performed by the 3 vane cascade compared to the 4 vane is evidenced by the larger wakes and lower velocity minima produced by the former. The bulk flow in each case exhibits a flat velocity profile, which is desirable. \( K_L \) data at an \( Re_c \) of \( 5.74 \times 10^5 \) for the two s/c’s investigated, are shown in Table 2-2. It is clear from this data that there is very little difference in loss coefficient between the two configurations.

Table 2-2: \( K_L \) values for each test configuration at \( Re_c \) of \( 5.74 \times 10^5 \)

<table>
<thead>
<tr>
<th>s/c</th>
<th>( K_L )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.237</td>
<td>0.132</td>
</tr>
<tr>
<td>0.190</td>
<td>0.128</td>
</tr>
</tbody>
</table>

The value of \( K_L \) measured at an s/c of 0.237 may be compared to previous data from Salter (1946), which showed that ¼ circle vanes set at an s/c ratio of 0.25, produced a \( K_L \) of 0.120 when tested at an \( Re_c \) of \( 2 \times 10^5 \). It is not possible to judge whether the difference in \( K_L \) between the two investigations is due to differences in downstream flow quality, since Salter did not publish any detailed velocity profile data. Non-dimensionalised velocity profiles measured 25mm upstream, and 875mm downstream of the cascade are presented in Figure 2-16.
875mm downstream of the cascade, the 3-vane configuration shows a significant momentum deficit at the inner bend compared to the 4 vane set-up. This is surprising given how closely matched the inner wall profiles of the 3 and 4 vanes were, 25mm downstream of the cascade. This deficit may be the result of some near wall instability which originates downstream of the inner bend and worsens as the near wall flow looses energy as it propagates further downstream.

As stated earlier, turning vane data currently available in the literature focuses almost entirely on determining pressure loss coefficient, with test rig data from Klein et al (1930), Collar (1936), Salter (1946) and Winter (1947) showing that thin ¼ circle turning vanes with a space to chord ratio ($s/c$), produced $K_L$ of between 0.12-0.20.

If $K_L$ had been the sole metric of consideration for the wind tunnel presented in this Thesis, then following the vane test module results, an $s/c$ of 0.237 would have been chosen over an $s/c$ of 0.190, given that pressure loss coefficients were similar and fewer vanes would have been required. However, the far-field uniformity data suggested that an $s/c$ of 0.190 (more vanes) gave better control of the inner-bend near-wall flow field, and this was therefore selected for use in the wind tunnel.
2.6.2 WIND TUNNEL VANES

Vanes used in both corners of the wind tunnel were identical to those employed in the vane test module except that the thickness was increased to 3mm ($r_i$ was maintained at 245mm) to enhance rigidity and thereby ensure the vanes maintained their profile across the entire span when under load.

It also enabled more roundness to be applied to the leading edge to make the vanes less sensitive to inlet flow angularity. This additional thickness was expected to have a negligible effect when making comparisons between the wind tunnel and vane test module results.

Employing a space to chord ratio of 0.190 resulted in there being 25 vanes in the first corner and 28 in the second as shown in Figure 1-4 and Figure 2-17. When the wind tunnel was run at its target working section velocity of 40m/s, the vanes in the first corner operated at an $Re_c$ of $7.97 \times 10^5$ and those in the second corner at an $Re_c$ of $5.74 \times 10^5$. The latter is the same $Re_c$ used in the vane test module described earlier.

Figure 2-17: Turning vanes installed in the first corner
The modules upstream and downstream of both corners feature 45° corner fillets to reduce secondary flows caused by interaction between the wall boundary layers. Careful attention was therefore paid to how each corner fillet would grow around the 90° corner. 25mm thick plywood spacers were used to ensure accurate setting of the vanes outside of the 45° corner fillet region.

In both corners, three vanes intersected each 45° degree fillet region as shown in Figure 2-18. Between each of these vanes a spacer was used to achieve most of the required depth and an appropriately carved piece of jelutong was fastened to the spacer to produce a smooth corner fillet.

Figure 2-18: Corner fillet treatment at inner bend of second corner
2.7 SHAPE CHANGE

A constant area shape change module shown in Figure 2-19, was placed between the fan and the second corner to effect a change of geometry from octagonal to circular.

![Figure 2-19: Constant area shape change](image)

Although velocity profile data from Figure 2-16, showed that wakes from the turning vanes had mixed out 875mm downstream of the cascade, the shape change was designed with a length of 1835mm to ensure that the local wall angles were not excessive. Consequently, the length of the shape change corresponded to approximately 0.7\(D_1\).

The pressure loss coefficient across the shape change was calculated in the same manner as for the working section.
2.8 THE FAN

The 2.65m-diameter, variable-speed, fixed-pitch, fan designed by Voith Howden is shown in Figure 2-19 and Figure 2-20. It has a hub to tip ratio of 0.42 and 9 free vortex designed rotor blades. The DC motor is housed in the nacelle and integrally cooled through 8 hollow straightener vanes. The latter are designed to produce 5° of swirl to aid the pressure recovery of the downstream diffuser. Rather than tapering to a sharp point, the end of the tailcone is abruptly cut-off to ensure a consistent point of separation. The fan housing is connected to the wind tunnel by means of a flexible coupling to minimise the transmission of structure borne vibration. The fan is bolted to a reinforced concrete plinth to minimise the transmission of vibration into the laboratory floor and hence the balance.

Figure 2-19: Fan schematic courtesy of Voith Howden GmbH

![Fan schematic](image)

Figure 2-20: Final positioning of the fan

![Final positioning of the fan](image)
The overall fan stagnation pressure rise $\Delta H$, was calculated by summing the stagnation pressure losses across each module used in the wind tunnel. These losses were determined using the methods described in previous sections of this Chapter:

$$\Delta H = \sum K_L q_L$$

The line power $P$, drawn by the fan was then computed using the equation below:

$$P = \frac{\Delta H A_{WS} U_{WS} (RF)}{\eta_{total}}$$

$$\eta_{total} = \eta_{fan} \times \eta_{motor} \times \eta_{controller}$$

Where $RF$ is a reserve factor of 1.1 commonly used to allow for additional losses through leaks and joints (Barlow 1999), and $\eta$ is efficiency. To minimise leakage losses flexible sealant was used between tunnel modules. Typical efficiencies advised by the fan manufacturer were 86%, 97% and 90% for the fan, controller and motor respectively.

The tunnel energy ratio, $ER$, is often used as a measure of the energy efficiency of a wind tunnel, although as stated by Barlow et al (1999), it is by no means a measure of the value of the tunnel for research and development. The definition of the energy ratio excludes the energy losses associated with the fan and the motor in order to focus on the aerodynamic aspects of the energy budget. The energy ratio is commonly defined as:

$$ER = \frac{1}{\sum K_L \left( \frac{q_L}{q_{WS}} \right)}$$

$$P = \frac{\left( \frac{1}{ER} \right) A_{WS} U_{WS} (RF)}{\eta_{total}}$$

Barlow states that the energy ratio for closed return and open circuit tunnels other than free-jet facilities is nearly always greater than unity. It is typically in the range 3-7 for closed throat tunnels. The energy ratio for a free-jet facility is always less than 1, which is why there are no large-sized facilities using the free-jet configuration.
2.9 BREAKDOWN OF PREDICTED PERFORMANCE AT INAUGURAL RUN

During fan commissioning, the wind tunnel was run without a model in the working section and without louvres and exit guide vanes.

To represent a clean working section in the theoretical prediction, $\alpha_1 = \alpha_2 = 1.04$ was used to determine the loss associated with the first and second diffusers, as recommended by ESDU datasheet 73024 for diffusers with upstream duct lengths less than $9D_f$. An $\alpha_1$ of 1.1 was used at inlet to the final diffuser given likely non-uniformity downstream of the fan.

The tunnel was commissioned in this configuration on a day with an ambient pressure of 102,500Pa and ambient temperature of 9°C. The tunnel achieved a top speed of 46.2m/s compared to a theoretical prediction of 43.8m/s as shown in Table 2-3 below.

Possible reasons for the discrepancy between predicted and measured tunnel speed will be discussed in Chapter 4.
Table 2-3: Breakdown of anticipated circuit losses at the predicted maximum working section velocity of 43.8 m/s in the inaugural run configuration.

- Inaugural run configuration: (no model / no inlet louvres/ no exit guide vanes)
- Ambient conditions at inaugural run: (Pressure = 102,500 Pa, Temperature = 9 C).
- Predicted maximum test section velocity = 43.8 m/s
- Actual maximum test section velocity = 46.2 m/s

<table>
<thead>
<tr>
<th>Component</th>
<th>Predicted $K_L$</th>
<th>$q_L$ (Pa)</th>
<th>$K_{LWS}$</th>
<th>$H_L$ (Pa)</th>
<th>$\alpha_1$</th>
<th>$\alpha_2$</th>
<th>$C_p$</th>
<th>Loss (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Filter Mesh</td>
<td>3.252</td>
<td>22.6</td>
<td>0.0606</td>
<td>73.5</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>7.7</td>
</tr>
<tr>
<td>Honeycomb</td>
<td>0.350</td>
<td>22.6</td>
<td>0.0065</td>
<td>7.9</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>0.8</td>
</tr>
<tr>
<td>Screen 1</td>
<td>0.895</td>
<td>22.6</td>
<td>0.0167</td>
<td>20.2</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>2.1</td>
</tr>
<tr>
<td>Screen 2</td>
<td>0.979</td>
<td>22.6</td>
<td>0.0183</td>
<td>22.1</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>2.3</td>
</tr>
<tr>
<td>Contraction</td>
<td>0.461</td>
<td>22.6</td>
<td>0.0086</td>
<td>10.4</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>1.1</td>
</tr>
<tr>
<td>Working Section</td>
<td>0.018</td>
<td>1212.9</td>
<td>0.0179</td>
<td>21.8</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>2.3</td>
</tr>
<tr>
<td>Safety Screen</td>
<td>0.199</td>
<td>1187.6</td>
<td>0.1944</td>
<td>235.8</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>24.7</td>
</tr>
<tr>
<td>First Diffuser</td>
<td>0.142</td>
<td>1187.6</td>
<td>0.1395</td>
<td>169.2</td>
<td>1.04</td>
<td>1.04</td>
<td>0.483</td>
<td>17.7</td>
</tr>
<tr>
<td>First Corner</td>
<td>0.150</td>
<td>473.4</td>
<td>0.0585</td>
<td>71.0</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>7.4</td>
</tr>
<tr>
<td>Second Diffuser</td>
<td>0.098</td>
<td>473.4</td>
<td>0.0381</td>
<td>46.2</td>
<td>1.04</td>
<td>1.04</td>
<td>0.400</td>
<td>4.8</td>
</tr>
<tr>
<td>Second Corner</td>
<td>0.150</td>
<td>246.9</td>
<td>0.0305</td>
<td>37.0</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>3.9</td>
</tr>
<tr>
<td>Shape Change</td>
<td>0.002</td>
<td>246.9</td>
<td>0.0004</td>
<td>0.5</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>0.0</td>
</tr>
<tr>
<td>Third Diffuser</td>
<td>0.562</td>
<td>244.1</td>
<td>0.1131</td>
<td>137.2</td>
<td>1.10</td>
<td>-</td>
<td>0.538</td>
<td>14.4</td>
</tr>
<tr>
<td>Exit Guard Mesh</td>
<td>0.199</td>
<td>80.3</td>
<td>0.0131</td>
<td>15.9</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>1.7</td>
</tr>
<tr>
<td><strong>TOTAL</strong></td>
<td><strong>0.7163</strong></td>
<td><strong>868.8</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td><strong>100.0</strong></td>
</tr>
</tbody>
</table>

- Working Section Velocity: 43.8 m/s
- Allowance for leaks and joints: 10%
- Fan Stagnation Pressure Rise: 955.7 Pa
- Fan Volume Flow Rate: 109.0 m$^3$/s
- Total Efficiency: $75.0\% \quad (\eta_{\text{fan}}(\eta_{\text{controller}})(\eta_{\text{motor}}) = (0.86)(0.97)(0.90))$
- Fan Line Power (inc 1.8kW cooling fan): 140.6 kW
- Energy Ratio: 1.40
Although the energy ratio of 1.40 at first appears quite low (when compared to the range of 3-7 proposed by Barlow et al (1999) for closed circuit tunnels), it should be noted that the wind tunnel is an indraft design and features a very fine and consequently high-loss filter mesh that is not usually employed in traditional open circuit wind tunnels that are housed within laboratories.

Additionally, the safety screen at the end of the working section contributed nearly a quarter of the overall circuit loss. Although there had been debate during the design phase on whether this screen ought to be located upstream of the first corner turning vanes or in front of the fan, the value of placing it at the end of the working section was witnessed during fan commissioning when the difference in static pressure between the working section and the surrounding laboratory resulted in the temporary working section floor (fitted for the inaugural run) being sucked up. Had the safety mesh not been there, or even been placed at the end of the first diffuser, significant damage would have resulted to the turning vanes and the first diffuser itself.

The three main constituents of the wind tunnel, namely the flow conditioning devices, diffusers and turning vanes, represent 12.9%, 36.9% and 11.3% of the overall circuit loss respectively. The relatively large contraction ratio helps the honeycomb and turbulence reduction screens to contribute a combined total of only 5.2% to the overall loss of the tunnel, and also ensures that maximum tunnel speed will be reasonably unaffected should the filter mesh get clogged with dirt. Whilst the three diffusers represent 36.9% of the total loss, it is the first and final diffusers that account for the majority of this total. The loss associated with the first diffuser is large due to its high inlet dynamic pressure, and the final diffuser loss is substantial because the kinetic energy contained within its exit flow is dumped to atmosphere rather than being utilised in a downstream duct. Whilst the corner cascades were predicted to represent 11.3% of the overall loss of the wind tunnel, this was derived by using the same loss coefficient in each corner, since the lack of published information made it impossible to be any more refined in apportioning corner loss coefficients.

This Chapter has established the methodologies used to conduct the pragmatic design of each component in the wind tunnel circuit. A prediction for the maximum working section velocity and a detailed breakdown of losses around the circuit has also been presented. The following Chapter will provide information on the calibration and commissioning of the working section to enable future designers to evaluate the suitability of such a design for their own applications. Chapter 4 will then compare the predicted performance data listed in Table 2-3 with in situ measurements taken from the wind tunnel to promote an understanding of how modules perform when part of a complete wind tunnel system. By feeding this in situ data back into the software developed during the design phase it will be possible to quantify the extent to which future performance predictions may be improved.
3 CALIBRATION AND COMMISSIONING

As discussed in Chapter 1, published data on the performance of indraft wind tunnels of similar size and layout, as the one presented in this Thesis does not currently exist. Therefore one of the primary aims of this Thesis is to provide such data in order to enable future designers to determine whether such a tunnel would suit their requirements.

This Chapter provides experimental data to quantify how well the tunnel met the design brief set out in section 1.2. Specifically, this relates to the measurement of the boundary layer; velocity uniformity and turbulence intensity at the working section mid plane; the distribution of roof Cp with a Davis model in the working section and the repeatability of aerodynamic coefficients obtained from such a model.

Since the inlet and exit of the tunnel are exposed to atmosphere, these data are presented both for days with still air and those with ambient winds.

3.1 BOUNDARY LAYER

Knowledge of the boundary layer thickness in the working section is needed to enable the ground plane simulation and boundary layer control system to be designed. A rapid and robust method for predicting the working section boundary layer thickness at the conceptual design stage is therefore required. Although CFD lends itself to such a task, it is computationally expensive and has tended not to fully capture the velocity overshoot and undershoot described in section 2.3.2 (Su, 1991 and Sykes 2004). This may be due to the boundary conditions for most models employing a simple, uniform inlet velocity profile.

In their tests on the new boundary layer control system installed in NRC’s 9m x 9m tunnel, Larose et al (2001), proposed that for modern contractions designed along the guidelines put forward by Downie et al (1984) (discussed in section 2.3.6), where large radii of curvature are employed at inlet and exit of the contraction to alleviate adverse pressure gradients, and a short steep central section is used to thin the boundary layer to a minimum at the point of inflection, the boundary layer thickness, $\delta$, in a wind tunnel working section may be evaluated analytically by treating it’s growth as that of a turbulent boundary layer growing along a flat plate of length, $x$, originating at the point of inflection of the contraction (location, $X$, in Figure 2-3). One common equation for the growth of a turbulent boundary layer in a zero pressure gradient, is provided by Street (1996) as:
\[
\frac{\delta}{x} = \frac{0.38}{\text{Re}^{0.2}}
\]

As described in section 2.3.6, Downie’s guidelines were adhered to during the design of the contraction for this wind tunnel.

From Figure 2-3 it may be seen that the distance between the point of inflection, \(X\), and the working section mid plane (measurement location A), is 4477mm, giving a predicted boundary layer thickness of 65mm at 40m/s. The calibration of the working section provided an opportunity to test the applicability of Larose’s method to a smaller contraction.

Figure 3-1 compares an experimentally determined boundary layer profile with a turbulent boundary layer (of the equivalent experimentally determined \(\delta\)), following a \(1/7^{\text{th}}\) power law. The profile was measured in the middle of the floor at Station A on Figure 2-3.

The experimental profile was measured using a pitot probe referenced to tunnel static. The probe was traversed in 5mm increments. The differential pressure transducers used to measure dynamic pressure had ranges of +/- 150mmH\(_2\)O for +/-10V output and were accurate to 0.25% of reading down to 25mmH\(_2\)O. A 12-bit data acquisition card was used in conjunction with software written in National Instruments LabVIEW to acquire data. Data was sampled at 1kHz and averaged over 40,000 samples to ensure confidence in the mean. After moving the probe to the next traverse position, a settling time of 30 seconds was allowed prior to acquiring data.

Boundary layer displacement and momentum thicknesses are presented in Table 3-1 and were calculated using the equations below, which employ numerical integration by the trapezoidal rule, of the deficit in the boundary layer profile:

\[
\delta^* = \int_0^\infty \left(1 - \frac{\tilde{u}}{U_\infty}\right) dy \approx \sum_{i=1}^{N} \left\{ \frac{\left(1 - \frac{\tilde{u}}{U_\infty}\right)_{i} + \left(1 - \frac{\tilde{u}}{U_\infty}\right)_{i+1}}{2(y_{i+1} - y_i)} \right\}
\]

\[
\theta = \int_0^\infty \frac{\tilde{u}}{U_\infty} \left(1 - \frac{\tilde{u}}{U_\infty}\right) dy \approx \sum_{i=1}^{N} \left\{ \frac{\left(\frac{\tilde{u}}{U_\infty}\right)_{i} \left(1 - \frac{\tilde{u}}{U_\infty}\right)_{i} + \left(\frac{\tilde{u}}{U_\infty}\right)_{i+1} \left(1 - \frac{\tilde{u}}{U_\infty}\right)_{i+1}}{2(y_{i+1} - y_i)} \right\}
\]
Figure 3-1: Boundary Layer Profiles at Station A, Figure 2-3 at 40m/s

![Floor Boundary Layer Profile](image)

Table 3-1: Comparison between Theoretical and Experimental Boundary Layer Data, at Station A, Figure 2-3

<table>
<thead>
<tr>
<th></th>
<th>Turbulent</th>
<th>Experimental</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\delta) (mm)</td>
<td>60.0</td>
<td>60.0</td>
</tr>
<tr>
<td>(\delta^*) (mm)</td>
<td>8.9</td>
<td>9.4</td>
</tr>
<tr>
<td>(\theta) (mm)</td>
<td>4.6</td>
<td>5.5</td>
</tr>
</tbody>
</table>

The experimental data compares well with Larose’s method but perhaps suggests that in this case, the boundary layer originates a little downstream of the point of inflection. A boundary layer thickness of 60mm would be produced over a development length of around 4100mm, which places the point of origin approximately 377mm or 0.075\(L_C\) downstream of the point of inflection.

These findings and those of Larose (the latter for a larger scale contraction), suggest that using the contraction inflection point as the origin of the boundary layer provides a sensible approximation of
the boundary layer thickness in the working section. However, since most modern contractions are
designed using the same philosophy of large inlet and outlet radii and a short steep central section,
this method provides a useful and rapid first approximation, which is both reliable and robust and
without an appreciable scale effect.

The close agreement between the experimental and theoretical boundary layer thickness \( \delta \),
displacement thickness \( \delta^* \), and momentum thickness \( \theta \), shown in Table 3-1 also suggests that the
boundary layer develops along the surface of the contraction without separation.

3.2 VELOCITY UNIFORMITY

Barlow et al, 1999, proposed that for a general purpose wind tunnel such as the one described in this
Thesis, the velocity variation in the working area of the jet ought to deviate from the mean by less
than 0.3\%. The following sections describe how the velocity uniformity was experimentally
determined both on days with still air and in the presence of ambient winds. The level of
repeatability of the data is also discussed.

3.2.1 METHOD:

Having established (in section 3.1) a boundary layer thickness of 60mm at 40m/s (at station, A in
Figure 2-3), a traverse outside the boundary layer was performed in order to establish the core flow
velocity uniformity. A pitot probe was traversed in an 80mm grid, 100mm in from the roof and floor
of the working section, and dynamic pressure was obtained by referencing total pressure measured by
the pitot, to tunnel static. At each traverse position, dynamic pressure from the traversed probe and
the tunnel pitot static were recorded simultaneously.

The tunnel pitot static was a Prandtl design with a tube outer diameter of 8mm. The static holes were
located 8 diameters downstream of the tip (to minimise the influence of local acceleration around the
tip) and 8 diameters upstream of the stem (away from the pressure signature of the stem). The head
of the tunnel pitot static was located at the start of the 3600mm long working section shown in Figure
2-3, and was positioned 200mm below the tunnel roof and 500mm to the left of the tunnel centreline
(looking in the flow direction).

The differential pressure transducers used to measure dynamic pressure had ranges of +/- 150mmH\(_2\)O
for +/-10V output and were accurate to 0.25% of reading down to 25mmH\(_2\)O. A 12-bit data
acquisition card was used in conjunction with software written in National Instruments LabVIEW to
acquire data. A settling period of 30 seconds was allowed after the probe had moved to a new
position, and data was sampled at 1 kHz and averaged over 40,000 samples to promote confidence in the mean.

In order to quantify the effect of ambient winds on working section uniformity, the Department had access to data recorded by an Oregon Scientific WMR112 Professional Weather Station anemometer located in line with the face of the boiler house facing the wind tunnel inlet; 7m above the ground and 22m from the centreline of the contraction, as shown in Figure 1-6. By sampling anemometer data during a uniformity traverse, it was possible to determine the magnitude and direction of the strongest ambient gust.

Ideally, a series of anemometers in the locality of the inlet and exit of the wind tunnel would have provided a more complete picture of the flow characteristics in the region. Nevertheless, the anemometer used was adequately positioned to allow it to record the magnitude and direction of the ambient wind.

Being 7m above the ground it was at least 2.2m higher than the nearest building (which was the boiler house).

### 3.2.2 STILL AIR BASELINE

Figure 3-2 depicts the percentage variation from the mean axial velocity, $U$, at station A in Figure 2-3. This map was produced from measurements taken on days with no ambient winds. It may be seen that the velocity variation in the area normally occupied by floor mounted automotive models ($Y +/\sim 250\text{mm}, Z < 350\text{mm}$) deviates by around 0.1% from the average.

Some detail of the quality of the flow closer to the ground is given in Figure 3-1 and more detailed assessments of the flow quality in this region will be made during a future installation of a boundary layer control system.

In the area normally occupied by aeronautical half models ($Y +/\sim 100\text{mm}, Z < 1000\text{mm}$), velocity uniformity deviates by around 0.3% from the average. In three localised areas outside of the model test area, velocity uniformity reaches $-0.4\%$. The typical concave velocity distribution produced at exit of a contraction (discussed earlier in section 2.3.2) is shown by the velocity deficit in the core of the jet compared to the velocity overshoot around the perimeter.
Figure 3-2: Percentage Variation from Mean Velocity at Station A, Figure 2-3 at 40m/s (looking in flow direction).
3.2.3 REPEATABILITY OF STILL AIR BASELINE

To determine the level of repeatability of the still air baseline condition, uniformity traverses were performed on three separate days, during which the weather station described in section 3.2.1 indicated the absence of ambient winds.

The traverses were performed at a nominal velocity of 40m/s, along the vertical centreline of the working section (Y = 0mm), at station A on Figure 2-3. The pitot probe was traversed in vertical increments of 80mm starting 100mm above the floor (Z = 100mm) and finishing 100mm below the roof (Z = 1220mm). The methodology used for data capture was as described in section 3.2.1. Since it was not possible to predict the days on which there would be no ambient winds, the traverse rig had to be installed in the working section prior to each traverse.

The results from the three traverses are presented in Figure 3-3. Each traverse again shows the typical velocity deficit in the core of the jet compared to a velocity overshoot around the perimeter (as previously discussed in section 3.2.2).

It is evident from the graph that the repeatability of the uniformity measurement is of the order of 0.1%. In each traverse, the level of uniformity does not exceed the limit of 0.3% proposed by Barlow (1999).

Having established that the velocity uniformity in the working section is satisfactorily repeatable in the absence of ambient winds, the next step was to quantify the effect of ambient winds. This is the subject of the next section.
Figure 3-3: Percentage Variation from Mean Velocity Along Working Section Vertical Centreline at Station A, Figure 2-3 at 40m/s. No Ambient Winds.
3.2.4 EFFECT OF AMBIENT WINDS

To determine the effect of ambient winds on the velocity uniformity in the working section, traverses were performed on three separate days. As a control, the first day was the still air baseline used in Figure 3-2 and 3-3. The remaining two traverses were performed when the weather station described in section 3.2.1, indicated that ambient winds were blowing from a direction of approximately 10°, with a magnitude of approximately 2.1 m/s (5.3% of the test velocity) and 4.6 m/s (11.5% of the test velocity). Although data from days with stronger winds was desired (Ross et al. 1986, had measured acceptable uniformity for the 80ft x 120ft indraft tunnel at Ames in the presence of ambient winds up to 15% of the test velocity), it was not possible to get this data within the timescales of the test programme due to the unpredictability of the weather.

With the working section operating at 40 m/s, the velocities at inlet to the bellmouth and settling chamber were 4.3 m/s and 5.5 m/s respectively. It was therefore judged that performing traverses in ambient winds of around 2.1 m/s (5.3% of the test velocity) and 4.6 m/s (11.5% of the test velocity) would provide useful information on the effect of ambient winds on uniformity in the working section.

Since the wind tunnel was itself angled at 35° from North (Figure 1-6), an impingement angle of 10° (measured from North) was chosen to ensure the ambient wind was blowing at an angle of 45° to the face of the bellmouth. This angle was judged to be potentially the most detrimental to working section flow quality, as it would in all likelihood induce a separation off the corner of the boiler house. Since it was not possible to predict the days on which these specific ambient winds would occur, the traverse rig had to be installed in the working section prior to each traverse.

The methodology used for data capture was as described in section 3.2.1, and the results are presented in Figure 3-4. Each traverse shows a velocity deficit in the core of the jet compared to a velocity overshoot around the perimeter. As mentioned in section 3.2.2, this is typical of the concave velocity distribution produced at the exit of a contraction.

It is evident from Figure 3-4 that the repeatability of the uniformity measurement is of the order of 0.1%. Whilst in each traverse, the level of uniformity does not exceed the limit of 0.3% proposed by Barlow (1999), both the traverses on days with ambient winds exhibit uniformity very close to 0.3% up to a height of 200 mm above the floor. This compares to a uniformity of close to 0.2%, in the same region, observed on three days with no ambient winds (Figure 3-3). However, since the level of repeatability of the still air condition was also around 0.1%, it is hard to say conclusively whether this observed difference between the ‘ambient wind’ and ‘no ambient wind’ condition is significant.
Since this is the area occupied by automotive models, data from days with stronger winds would be a useful addition to Figure 3-4.

It is reasonable to argue that the boiler house in Figure 1-6 helps shield the inlet and exit of the tunnel from the ambient winds blowing directly into them. Since space is often a premium, new indraft tunnels are likely to have buildings around their locality, which can be used directly or perhaps modified (with false wall extensions) to help shield them from the effects of ambient winds. In the event that the designer is fortunate enough to have access to a clear site around the tunnel, trees (or even a false wall) can be constructed approximately one hydraulic diameter opposite the inlet and exit. Such a barrier would also reduce the impact of noise on local residents and possibly improve aesthetics.
Figure 3-4: Percentage Variation from Mean Velocity Along Working Section Vertical Centreline at Station A, Figure 2-3 at 40m/s. Varying Ambient Winds.
3.3 TURBULENCE INTENSITY

The design specification in section 1.1 stated that it was desirable to have a working section axial turbulence intensity of around 0.1% at a velocity of 40m/s. A single-wire miniature hot-wire was used with a Constant Temperature Anemometer (CTA) and associated wire calibration and post processing software from Dantec Dynamics, to determine the turbulence intensity in the working section. The wire was approximately 5 μm in diameter and 1.2 mm long, and is commonly used for airflow applications with turbulence intensities up to around 5-10% (Jorgensen 2002). These types of wires have a high frequency response and can be repaired if (or rather when!) broken by foreign object impact. The sensor is mounted perpendicular (and with the prongs parallel) to the flow, to prevent separation from the prongs influencing the results.

The CTA anemometer works on the basis of convective heat transfer from a heated sensor to the surrounding fluid, with the heat transfer being primarily related to the fluid velocity (Jorgensen 1996 and Bruun 1996). The wire is heated to approximately 250K above the temperature of the surrounding fluid. A Wheatstone bridge circuit with feedback, allows correction of the feeding current to compensate for changes in heat loss due to convection changes around the wire in the presence of a moving fluid. As a result the temperature of the wire is held constant (hence the term CTA). The wire voltage time history can be converted to velocity data through application of a transfer function derived from the velocity calibration of the wire.

Dantec’s StreamWare professional application software was used to configure the hot wire probe, perform the hardware and signal conditioner set-ups, perform the wire velocity calibration, sample the wire voltage time history and post process the acquired data. The methodology used to perform a hot wire test is outlined below.

3.3.1 METHOD

1) An automated traverse mounted in the working section was used to move the wire to the centre-point of the working section for the velocity calibration of the wire.

2) Ambient gust magnitude and direction were recorded from the weather station described in section 3.2.1.

3) The tunnel was run at 40m/s for a few minutes to equalise the working section and ambient air temperatures. This was necessary to ensure that there would be no change in working section air temperature between wire calibration and capture of the wire voltage time history. As a result it was not necessary to apply temperature compensation to the dataset.
4) Care was taken to ensure that the BNC’s for the temperature and wire probes did not make electrical contact with each other or any metal objects.

5) A shorting probe was used in place of the wire to determine the resistances for the cables and supports. The wire was then reconnected.

6) The tunnel was run up to the lower end of the 37-43m/s velocity calibration range (for a test at 40m/s). In order to maximise the resolution of the 12-bit A/D board, the signal conditioner was used to subtract a DC offset voltage at 37m/s and a suitable gain was then applied to ensure that as much of the 0-10V range of the A/D converter was utilised at 43m/s. As it might be sensibly argued that most of the turbulent energy in a clean wind tunnel working section would be associated with large scale eddies, and that the major dimension of these eddies would be constrained by the 1.92m width of the working section, this would result in a large eddy frequency of approximately 20Hz at a convection velocity of 40m/s. A low-pass anti-aliasing filter was therefore set at a Nyquist frequency of 200Hz (10 times the large eddy frequency).

7) A velocity calibration was performed over 13 points between 37 and 43m/s. At each calibration point, data was acquired at 1kHz and averaged over 8192 samples to promote confidence in the mean.

8) The CTA application software then automatically applied a curve fit (either polynomial or King’s Law) to the velocity calibration data to minimise curve fit errors.

9) The traverse was used to position the wire at the required location in the working section and after a settling period of 20 seconds, the wire voltage was acquired at a 2kHz-sampling rate (10 times the Nyquist frequency), with a time history length of 327680 samples to enable the data to be ensemble averaged over 40 blocks with a frequency resolution of 0.244Hz. The acquired data was then post processed within the Dantec StreamWare application software using the relevant velocity calibration transfer function and signal conditioner settings (removing the gain and DC offset previously applied) to produce a wire velocity time history.

10) Dantec StreamWare then calculated statistically averaged values of mean and RMS velocity, using the equations shown below.

11) Steps ‘9 –10’ were repeated for other locations within the working section

12) At the end of the test, ambient gust magnitude and direction were recorded from the weather station described in section 3.2.1.

\[
U = \bar{u} = \frac{1}{N} \sum_{i=1}^{N} u \quad \text{(mean velocity)}
\]
\[
\sqrt{u'^2} = \sqrt{\frac{1}{N} \sum_{i=1}^{N} u_i^2} - \bar{u}^2 \quad \text{(RMS velocity)}
\]

\[
T_u = \frac{\sqrt{u'^2}}{u} \quad \text{(turbulence intensity)}
\]

### 3.3.2 RESULTS AND DISCUSSION

Figure 3-5 shows a plot of turbulence intensity along the vertical centreline of the working section at Station A on Figure 2-3 at 40m/s. A new hot wire set-up and calibration was performed for each traverse. Set-up and calibration of the hot wire and the acquisition and post processing of the velocity time histories are outlined in section 3.3.1. Four traverses were performed between 300 < Z < 1020 in 60mm increments, and one between 60 < Z < 275 in 10mm increments up to 100mm and thereafter in increments of 25mm.

The ‘No Ambient Wind’ baseline traverse has an average turbulence intensity (averaged along the height of the traverse) of 0.14%, with data scatter of +/- 0.03% about this mean largely due to temporal variations in ambient winds during the traverse. A fresh set-up and calibration of the hot wire was used for the repeat traverse, resulting in an average turbulence intensity of 0.16%, with data scatter of +/- 0.02% about the mean.

A subsequent traverse in the presence of an ambient wind with a magnitude of 1.8m/s at 268° (Figure 1-4) also resulted in an average turbulence intensity of 0.16%, with data scatter of +/- 0.02% about the mean. Therefore, in still air (and very close to still air) conditions, the turbulence intensity outside of the boundary layer and along the vertical centreline of the working section may sensibly be said to be in the region of 0.15%, which is higher than the target of 0.1% set out in the design brief (section 1.1). If desired by AAE, this turbulence intensity target may be achieved through the addition of a third screen without reducing top speed below the target of 40m/s.

Another traverse was performed in the presence of a stronger ambient wind with a magnitude of 6.4m/s (16% of the test velocity) at 230°. In this case, not only did the average turbulence intensity increase from 0.15% to 0.35%, but the data scatter also widened to a variation of +/- 0.11% about the mean. This indicates that the average and time dependent working section turbulence intensity is dependent upon the magnitude of the ambient wind. This is in broad agreement with data taken by Ross et al (1986) who showed that axial turbulence intensity for the Ames 80ft x 120ft indraft tunnel increased from 0.4% to 0.6% when ambient wind increased from 13% to 23% of the test velocity.
It is argued that since devices within the circuit (screens/contraction) acting to suppress axial turbulence are a constant, the global axial turbulence intensity in the working section is dependent on the magnitude of the fluctuating velocity component at inlet to the bellmouth. The magnitude of this fluctuating velocity component is likely to increase as a consequence of eddies formed by ambient gusts. Although tests under a wider range of ambient winds are required to build a map of acceptable operating regimes (beyond the timescale of this Thesis), this data is useful both to potential indraft tunnel designers and users of the existing tunnel. For the former, it presents data previously unavailable for a tunnel of this size and configuration, and better enables a decision to be made on the number of screens to be used in the settling chamber, and also on the degree of shielding necessary (perhaps by trees) around the wind tunnel inlet to protect it from impinging gusts. For the latter it highlights the need to monitor ambient wind magnitude and direction and aids the decision to reschedule tests where control of working section turbulence below 0.35% is necessary.

The final traverse presented in Figure 3-5 depicts the ‘No Ambient Wind’ variation in turbulence intensity between 60 < Z < 275 (the region occupied by automotive models). For this traverse, a wire velocity calibration between 0m/s and 45m/s was required and at the edge of the boundary layer (z = 60mm as shown by Figure 3-1), a turbulence intensity of 3% was measured. The turbulence intensity outside the influence of the wall (i.e in the core), may be expected to be low and increase as the wall is approached. This is exemplified by the gradient in turbulence intensity visible in Figure 3-5.

Figure 3-5: Effect of Ambient Winds on Turbulence Intensity Along the Working Section Vertical Centreline at Station A, Figure 2-3 at 40m/s.
3.4 DISTRIBUTION OF ROOF STATIC PRESSURE COEFFICIENT

As discussed in section 2.4.2, the cross-section of the working section was increased from 1920mm (width) x 1320mm (height) at inlet, to 1940mm (width) x 1320mm (height) at exit, to account for the growth in boundary layer displacement along the working section. This was intended to help minimise the longitudinal drop in static pressure (horizontal buoyancy), which would otherwise have the effect of sucking the model downstream, thereby producing an artificial increase in drag.

The static pressure coefficient distribution ($C_p$), along the centreline of the roof of the tunnel was therefore determined both in clean conditions and with a Davis model in the working section at 40m/s, in order to determine the following:

- Quantify the horizontal buoyancy along the working section.
- Whether the tunnel pitot static was positioned a sufficient distance upstream of a 25% scale automotive model.
- Whether the working section was sufficiently long to allow $C_p$ to recover prior to the first diffuser.

$C_p$ is defined as:

$$C_p = \frac{p_L - p_\infty}{\frac{1}{2} \rho U_\infty^2}$$

A Davis model was chosen since this type of model had been specifically developed for a study into road vehicle wakes and to exhibit the overall flow field characteristics of a real vehicle (Davis, 1982). A picture of the Davis model in the working section is provided in Figure 3-6.
The Department had access to several Davis models and the one closest to representing a 25% scale car with 5% blockage based on frontal area was chosen for the tests. Characteristics for the chosen model are listed in Table 3-2, with the model scale based on a full size car having a frontal area of $2\text{m}^2$. The model was centered about the balance point of resolution (section 2.4.3) with a ride height of 60mm (the height of the boundary layer).

<table>
<thead>
<tr>
<th>Table 3-2: Davis Model Characteristics</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length (mm)</td>
</tr>
<tr>
<td>Width (mm)</td>
</tr>
<tr>
<td>Height (mm)</td>
</tr>
<tr>
<td>Wheelbase</td>
</tr>
<tr>
<td>Front/rear track</td>
</tr>
<tr>
<td>Front/rear ride height</td>
</tr>
<tr>
<td>Frontal Area (m$^2$)</td>
</tr>
<tr>
<td>Blockage (%)</td>
</tr>
<tr>
<td>Scale (%)</td>
</tr>
</tbody>
</table>

Static pressure tappings were inserted along the centreline of the roof of the tunnel. To enable these tappings to be used in future for Pressure Signature Correction, the spacing between tappings was chosen as 5% of the width of the working section, as proposed by Hackett et al (1979); the originators of this correction method.
The tappings were constructed from brass tubing with an outside diameter of 1.6mm and an inside diameter of 0.85mm. As recommended by Shaw (1960), to improve data quality, the tappings were squared off, de-burred and mounted flush with the wetted surface. Epoxy resin was applied around the non-wetted junction between the tapping and the module surface to prevent leakage.

The differential pressure transducers used were accurate to 0.25% of reading and had an output of +/- 10V. A 12-bit data acquisition card was used in conjunction with software written in National Instruments LabVIEW to acquire data. The tappings were connected to a scanivalve, and data was sampled at 1kHz and averaged over 30,000 samples to promote confidence in the mean. A 1-minute settling time was allowed between cycles of the scanivalve to a new tapping, since the tubing used was necessarily long.

A baseline \( Cp \) distribution and two repeats were performed in a clean working section at a nominal velocity of 40m/s. The Davis model was then installed, with its length and width equally spaced about the balance point of resolution, and with a front and rear ride height of 60mm. The \( Cp \) distribution was again measured at a nominal velocity of 40m/s.

All tests were performed on the same day; over the course of which, the peak ambient wind measured by the anemometer described in section 3.2.1, had a magnitude of 3.7m/s at a direction of 22°.

All four sets of data are plotted in Figure 3-7. The repeatability of the \( Cp \) data for the three clean configuration runs was within the \( Cp \) +/- 0.002 range proposed by Hackett (1979).

The data scatter along the length of the working section is probably due to local roughness effects and minor protrusions around the tappings. These are understandable, since the roof of the tunnel was constructed from plywood and then painted. Using a machined aluminium surface would have improved the quality of the data but at significant cost.

The monotonic decline of static pressure by \(-0.02Cp\) in the region 500mm upstream of the start of the working section is expected as this forms part of the contraction. However, a further static pressure drop of \(-0.02Cp\) along the first \(-1000mm\) of the working section suggests that pressure gradients are still slightly decaying aft of the contraction. It may be argued that the parallel duct at the trailing edge of the contraction ought to be increased from 1 hydraulic radius to 1 hydraulic diameter. Despite the sidewalls diverging from the start to the end of the test section (0<X<3600mm), \( Cp \) can be seen to decay along the first half of the test section prior to flat-lining. It
is proposed that the decay may be due to cross flows between the wall and roof boundary layers taking longer than anticipated to attenuate.

The tip of the pitot static is located at the start of the working section (X = 0), and its static ports are positioned 64mm (8 tube outer diameters) downstream of the tip. The static ports are therefore positioned 0.55 hydraulic working section diameters upstream of the front of the model. It may be seen from Figure 3-7, that this is a satisfactory location for the pitot static, as it is not influenced by the static pressure signature of the model.

As discussed in section 2.4.3, the working section length of 3600mm was chosen to be twice its hydraulic diameter, since this was the lower limit of the range of 2-3 hydraulic diameters proposed by Bradshaw (1968), Barlow et al (1999), and Pankhurst and Holder (1968). However, extrapolating the model $C_p$ data using the trendline shown in Figure 3-7, suggests that working section length ought to be increased by 1 hydraulic radius to allow $C_p$ to recover to the ‘no-model’ condition at the end of the working section.

This suggests that for a wind tunnel designed to accommodate a single model of a typical automotive saloon, the working section length ought to be equivalent to at least 2.5 times the hydraulic diameter.

Figure 3-7: Roof $C_p$ distribution
3.5 REPEATABILITY OF AERODYNAMIC COEFFICIENTS

A fundamental requirement of any useful wind tunnel is to provide the user with repeatable model coefficient data. The wind tunnel was equipped with an Aerotech underfloor, virtual centre, 6-component balance, shown in Figure 3-8. Force and moment ranges, and accuracies determined from the onsite calibration programme are listed in Table 3-3. The balance featured analogue to digital conversion at the load cell to minimise signal degradation, and an automated yaw mechanism with a positional accuracy of 0.1°.

The balance was enclosed in a room to shield it from draughts occasionally present in the surrounding laboratory. Off-centric discs in the turntable (Figure 2-6) enable many combinations of wheelbase and track to be attained.

Figure 3-8: Underfloor, virtual centre, 6-component balance and turntable (Aerotech, UK)
Table 3-3: Balance Force and Moment Ranges and Accuracies

<table>
<thead>
<tr>
<th>Component</th>
<th>Force and Moment Ranges</th>
<th>Accuracy determined from onsite calibration (% of full scale)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Automotive and Aeronautical Drag ( C_D )</td>
<td>+/- 120N</td>
<td>0.010</td>
</tr>
<tr>
<td>Automotive Side Force ( C_Y )</td>
<td>+/- 420N</td>
<td>0.005</td>
</tr>
<tr>
<td>Aeronautical Lift ( C_L )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Automotive Lift ( C_L )</td>
<td>+/- 500N</td>
<td>0.010</td>
</tr>
<tr>
<td>Aeronautical Side Force ( C_Y )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Automotive and Aeronautical Rolling Moment ( C_{MX} )</td>
<td>+/- 150Nm (large due to aeronautical half model testing)</td>
<td>0.010</td>
</tr>
<tr>
<td>Automotive Pitching Moment ( C_{MY} )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Aeronautical Yawing Moment ( C_{MZ} )</td>
<td>+/- 60Nm</td>
<td>0.010</td>
</tr>
<tr>
<td>Automotive Yawing Moment ( C_{MZ} )</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Aeronautical Pitching Moment ( C_{MY} )</td>
<td>+/- 45Nm</td>
<td>0.015</td>
</tr>
</tbody>
</table>

Since the wind tunnel was designed to accommodate a 25% scale model of a typical automotive saloon, the same Davis model as described in section 3.4, was used in these sets of tests.

The Davis model was centred about the balance point of resolution (section 2.4.3) and supported 60mm above the tunnel floor by 4 rigid struts, which were each connected to an electronically switchable permanent magnet mounted to the live balance platform. Each strut was connected to the model via an 8mm threaded bar with a lock nut at either side of the model’s floor.

The model was installed and the balance zeroed in the wind-off condition. The tunnel was started and after a settling time of 20 seconds data was sampled at 20Hz for 10 seconds for each point in the yaw map. After collecting data on day 1, the tunnel was shut down and the balance locked. To remove the influence of model installation accuracy, the model was left in the working section overnight. On day 2 the balance was unlocked and re-zeroed in the wind off condition. The magnitude and direction of the peak gust during each test was determined using the weather station and method described in section 3.2.1. Data for both days are presented in Table 3-4 and Figure 3-9.
Table 3-4: Repeatability of Davis model coefficient data

<table>
<thead>
<tr>
<th>Day 1</th>
<th>wind 5.3m/s @ 3 deg</th>
<th>Working Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>Run/Cfg</td>
<td>Velocity (m/s)</td>
<td>Yaw (deg)</td>
</tr>
<tr>
<td>1/0</td>
<td>39.96</td>
<td>25</td>
</tr>
<tr>
<td>2/0</td>
<td>40.08</td>
<td>20</td>
</tr>
<tr>
<td>3/0</td>
<td>40.10</td>
<td>15</td>
</tr>
<tr>
<td>4/0</td>
<td>40.08</td>
<td>10</td>
</tr>
<tr>
<td>5/0</td>
<td>40.14</td>
<td>5</td>
</tr>
<tr>
<td>6/0</td>
<td>40.12</td>
<td>0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Day 2</th>
<th>wind 2.6m/s @ 15 deg</th>
<th>Working Section</th>
</tr>
</thead>
<tbody>
<tr>
<td>Run/Cfg</td>
<td>Velocity (m/s)</td>
<td>Yaw (deg)</td>
</tr>
<tr>
<td>1/0</td>
<td>40.00</td>
<td>25</td>
</tr>
<tr>
<td>2/0</td>
<td>40.19</td>
<td>20</td>
</tr>
<tr>
<td>3/0</td>
<td>40.15</td>
<td>15</td>
</tr>
<tr>
<td>4/0</td>
<td>40.11</td>
<td>10</td>
</tr>
<tr>
<td>5/0</td>
<td>40.15</td>
<td>5</td>
</tr>
<tr>
<td>6/0</td>
<td>40.11</td>
<td>0</td>
</tr>
</tbody>
</table>
Having removed the effect of model installation inaccuracy (by leaving the model in the working section overnight), the data in Table 3-4 and Figure 3-9 indicate a satisfactory level of repeatability for the 6 force and moment coefficients between two days, where the peak ambient gust was less than 5.3m/s (13% of the test velocity). Table 3-4 shows that at its worst, $C_D$ repeats to 2 drag counts, which will enable the tunnel to be used for vehicle optimisation studies.

Although it would have been desirable to perform these sets of tests in a range of ambient wind conditions, the unpredictability of the weather and the demands of other users for wind tunnel time did not make this possible. However, the data does correlate with that presented in section 3.2.4, which indicated that velocity uniformity was largely insensitive to ambient winds of less than 4.6m/s.

As tunnel usage builds in the future an understanding will develop of how model coefficient repeatability is affected by the presence of stronger ambient winds, and it will be useful to publish this data to aid indraft tunnel designers.
Chapter 4 – Comparison of Measured and Predicted Losses

4 COMPARISON OF MEASURED AND PREDICTED LOSSES

“...In theory there is no difference between theory and practice – but in practice there is...”

(Anon)

As acknowledged by Rae et al (1984) and Knowles et al (1998), wind tunnels represent a large capital investment to management and the pressure to begin testing is therefore irresistible. Consequently there is seldom time to conduct a detailed assessment of how the performance of each component of the tunnel compares with theoretical predictions.

Understanding how each component of the tunnel contributes to loss when part of a complete wind tunnel system (rather than in isolation) enables a better assessment of spatial, energy (and consequently budgetary) requirements to be made. It also allows the designer to tackle unconventional circuit layouts with greater confidence.

This Chapter aims to answer the following questions:

1. How does the measured fan running line and maximum working section velocity compare with theoretical predictions detailed in Table 2-3? What improvement in accuracy is obtained by substituting measured pressure loss coefficients \(K_L\) and diffuser pressure recovery coefficients \(C_{pr}\) into the model used to predict overall tunnel performance?

2. How well does the screen pressure loss \(K_L\) empirically derived from experiments in pipe rigs by Loehrke and Nagib (1972) and Schieman (1981) compare with data measured from the wind tunnel?

3. How well do diffuser pressure recovery coefficients \(C_{pr}\) presented in ESDU data sheets compare with data measured from the wind tunnel?

4. How well do pressure losses \(K_L\) and velocity profile data across ¼ circle turning vane cascades derived from experiments in model scale test rigs by Salter (1946), Winter (1947) and Johl et al (2007), compare with measured data from the wind tunnel?
4.1 COMPARISON OF OVERALL TUNNEL PERFORMANCE

As summarised by Table 2-3, it was apparent during the inaugural run, that the maximum achievable working section velocity (46.2m/s) was in excess of the predicted maximum (43.8m/s). In order to better understand the reasons for this error, the fan running line and losses across individual components were measured.

To determine the fan running line, the static pressure rise across the fan was measured by means of the averaged reading from two rings of static tappings located 80mm upstream of the fan inlet and 80mm downstream of the end of the tailcone (Figure 1-4).

Each ring comprised 4 tappings positioned at 45°, 135°, 225, and 315° around the circumference (the vertical plane being 0°). The tappings were constructed from brass tubing with an outside diameter of 1.6mm and an inside diameter of 0.85mm. As recommended by Shaw (1960), to improve data quality the tappings were squared off, de-burred and mounted flush with the wetted surface. Epoxy resin was applied around the non-wetted junction between the tapping and the tunnel outer surface to prevent leakage.

The differential pressure transducers used to measure the static pressure difference across the fan and corresponding working section dynamic pressure had ranges of +/- 150mmH₂O for +/-10V output and were accurate to 0.25% of reading down to 25mmH₂O. A 12-bit data acquisition card was used in conjunction with software written in National Instruments LabVIEW to acquire data.

From rest, the tunnel was run in increments of approximately 5m/s up to its maximum velocity of close to 45m/s. At each velocity, working section dynamic pressure and the static pressure rise across the fan were sampled at 1kHz for 10 seconds and averaged over 10 repeats to ensure confidence in the mean. A settling time of 2 minutes was allowed before sampling data at each speed since the tubing used was necessarily long.
Chapter 4 – Comparison of Measured and Predicted Losses

Figure 4-1: Comparison of measured and predicted fan running lines

Figure 4-1, indicates a 19% over-prediction (reasonably consistent through the speed range) of the fan pressure rise required to achieve a given volume flow rate.

Note that substituting measured loss and pressure recovery coefficients into the model used to predict overall tunnel performance generated the ‘feedback’ line.

Note that the volume flow rate, $Q$, is the product of the cross sectional area and velocity at any local cross section of the wind tunnel circuit (see relevant equations in section 1.4). For convenience working section data was used.
A summary of measured and predicted pressure loss coefficients ($K_L$) for the screens and corners and diffuser pressure recovery coefficients ($C_{pr}$) at 40m/s is presented in Table 4-1.

Table 4-1: Measured and predicted $K_L$ and $C_{pr}$ at 40m/s for the screens, corners and diffusers.

<table>
<thead>
<tr>
<th>Tunnel Section</th>
<th>$K_L$ (MEASURED)</th>
<th>$K_L$ (PREDICTED)</th>
<th>$C_{pr}$ (MEASURED)</th>
<th>$C_{pr}$ (PREDICTED)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet filter mesh</td>
<td>4.800</td>
<td>3.293</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Screen 1 + Honeycomb</td>
<td>1.445</td>
<td>1.266</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Screen 2</td>
<td>1.158</td>
<td>1.010</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Safety screen</td>
<td>0.251</td>
<td>0.199</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>First diffuser</td>
<td>-</td>
<td>-</td>
<td>0.563</td>
<td>0.483</td>
</tr>
<tr>
<td>Corner 1</td>
<td>0.120</td>
<td>0.150</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Second diffuser</td>
<td>-</td>
<td>-</td>
<td>0.429</td>
<td>0.400</td>
</tr>
<tr>
<td>Corner 2</td>
<td>0.171</td>
<td>0.150</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Third diffuser</td>
<td>-</td>
<td>-</td>
<td>0.643</td>
<td>0.538</td>
</tr>
<tr>
<td>Exit guard mesh</td>
<td>0.241</td>
<td>0.199</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

Loss and pressure recovery coefficients quoted in Table 4-1 were measured using a ring of static tappings located immediately upstream and downstream of each component. Each ring consisted of 4 tappings equally spaced around the perimeter. In the case of rectangular cross sections, the tappings were located in the centre of each flat section. The experimental set-up, data collection and data reduction techniques are as described at the start of section 4-1.

Feeding measured pressure loss coefficients ($K_L$) and diffuser pressure recovery coefficients ($C_{pr}$) quoted in Table 4-1 back into the model used to predict overall tunnel performance, results in the percentage over-prediction error reducing from ~19% to ~7.5%. The over-prediction error of ~7.5% is reasonably consistent through the speed range and the resultant running line is shown in Figure 4-1.

It is likely that the main contributor to the 7.5% over-prediction error is the 10% safety factor allowed for leaks and joints. It should be noted that this 10% allowance was only a rule of thumb proposed by Barlow (1999), and great care was taken in this design to adequately seal between tunnel modules (particularly as the test section static pressure was below atmospheric). The difference between measured and predicted circuit performance may suggest that 2.5% is a more realistic leakage figure. This being calculated as: The 10% allowed for leaks and joints minus the 7.5% over-prediction error following feedback of measured circuit losses.
It is evident from Table 4-1 that the screens generally contribute more losses than predicted whilst the diffusers perform better than predicted. Of the two turning vanes cascades, one performs slightly better and one slightly worse than anticipated. These issues will be discussed in more detail in upcoming sections of this Chapter. A summary of measured losses at 40m/s (with 2.5% allowed for leakage) is presented in Table 4-2.

Table 4-2: Breakdown of measured circuit losses at 40m/s in the inaugural run configuration.

- Inaugural run configuration: (no model / no inlet louvres/ no exit guide vanes)
- Ambient conditions at inaugural run: (Pressure = 102,500Pa, Temperature = 9 C)

<table>
<thead>
<tr>
<th>Tunnel Section</th>
<th>KL</th>
<th>qL (Pa)</th>
<th>KLWS</th>
<th>HL (Pa)</th>
<th>KEFC1</th>
<th>KEFC2</th>
<th>Cpr</th>
<th>Loss (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet filter mesh</td>
<td>4.800</td>
<td>18.9</td>
<td>0.0895</td>
<td>90.5</td>
<td>-</td>
<td>-</td>
<td></td>
<td>13.1</td>
</tr>
<tr>
<td>Screen 1 + HC</td>
<td>1.445</td>
<td>18.9</td>
<td>0.0269</td>
<td>27.2</td>
<td>-</td>
<td>-</td>
<td></td>
<td>3.9</td>
</tr>
<tr>
<td>Screen 2</td>
<td>1.158</td>
<td>18.9</td>
<td>0.0216</td>
<td>21.8</td>
<td>-</td>
<td>-</td>
<td></td>
<td>3.2</td>
</tr>
<tr>
<td>Contraction</td>
<td>0.468</td>
<td>18.9</td>
<td>0.0087</td>
<td>8.8</td>
<td>-</td>
<td>-</td>
<td></td>
<td>1.3</td>
</tr>
<tr>
<td>Working section</td>
<td>0.018</td>
<td>1011.6</td>
<td>0.0182</td>
<td>18.4</td>
<td>-</td>
<td>-</td>
<td></td>
<td>2.7</td>
</tr>
<tr>
<td>Safety screen</td>
<td>0.251</td>
<td>990.5</td>
<td>0.2458</td>
<td>248.6</td>
<td>-</td>
<td>-</td>
<td></td>
<td>28.4</td>
</tr>
<tr>
<td>First diffuser</td>
<td>0.062</td>
<td>990.5</td>
<td>0.0611</td>
<td>61.8</td>
<td>1.04</td>
<td>1.04</td>
<td>0.563</td>
<td>8.9</td>
</tr>
<tr>
<td>First corner</td>
<td>0.120</td>
<td>394.8</td>
<td>0.0468</td>
<td>47.4</td>
<td>-</td>
<td>-</td>
<td></td>
<td>6.8</td>
</tr>
<tr>
<td>Second diffuser</td>
<td>0.069</td>
<td>394.8</td>
<td>0.0268</td>
<td>27.1</td>
<td>1.04</td>
<td>1.04</td>
<td>0.429</td>
<td>3.9</td>
</tr>
<tr>
<td>Second corner</td>
<td>0.171</td>
<td>205.9</td>
<td>0.0348</td>
<td>35.2</td>
<td>-</td>
<td>-</td>
<td></td>
<td>5.1</td>
</tr>
<tr>
<td>Shape change</td>
<td>0.154</td>
<td>205.9</td>
<td>0.0313</td>
<td>31.7</td>
<td>-</td>
<td>-</td>
<td></td>
<td>4.6</td>
</tr>
<tr>
<td>Third diffuser</td>
<td>0.457</td>
<td>203.6</td>
<td>0.0920</td>
<td>93.0</td>
<td>1.10</td>
<td>-</td>
<td>0.643</td>
<td>13.4</td>
</tr>
<tr>
<td>Exit guard mesh</td>
<td>0.241</td>
<td>66.9</td>
<td>0.0159</td>
<td>16.1</td>
<td>-</td>
<td>-</td>
<td></td>
<td>2.3</td>
</tr>
<tr>
<td>TOTAL</td>
<td>0.7195</td>
<td>727.8</td>
<td></td>
<td>100.0</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Working Section Velocity 40.0 m/s
Allowance for leaks and joints 2.5 %
Fan Stagnation Pressure Rise 746.0 Pa
Fan Volume Flow Rate 99.6 m³/s
Fan Line Power (inc 1.8kW cooling fan) 93.7 kW
Energy Ratio 1.39

Using measured pressure loss and pressure recovery coefficients and a leakage allowance of 2.5%, results in a predicted maximum test section velocity of 45.8m/s compared to a measured maximum of 46.2m/s.
4.2 SCREEN LOSSES

As discussed in section 2.2, the wind tunnel is fitted with a fine filter mesh at inlet to the settling chamber to damp out ambient winds and prevent ingestion of small particles; a coarse turbulence reduction screen immediately downstream of a set of honeycomb; and a finer turbulence reduction screen upstream of the contraction inlet. A weld-mesh safety screen is fitted at the end of the working section to prevent damage to the first diffuser and first corner turning vanes. The same type of screen is also fitted at the exit of the final diffuser to prevent objects from entering the tunnel.

Loss coefficient ($K_L$) data presented in Table 4.1, suggest that empirically derived equations (from pipe rig tests by Schieman, 1981), presented in section 2.2.3, to predict screen loss coefficient, result in an under-estimation of $K_L$.

The three screens in the vicinity of the settling chamber were predicted to contribute 12.9% (Table 2-3) of the overall loss of the wind tunnel and were measured as contributing 20.2% (Table 4-2). The main reason for this error was the significant under-prediction of the loss associated with the fine filter mesh. Further discussion relating to each type of screen is contained in upcoming sections.

4.2.1 INLET FILTER MESH

$K_L$ for the filter mesh was predicted as 3.293 and measured as 4.800, an under-estimation of 46%. Consequently this filter mesh contributes 13.1% of the overall loss of the tunnel compared with a predicted value of 7.7%. Groth and Johansson (1988), have stated that wake coalescence and consequent flow unsteadiness might occur for screens with open areas < 57% so this may contribute to the error in $K_L$ witnessed for the fine filter mesh (open area of 39.3%). Unfortunately Schieman, 1981 does not quote an open area boundary outside of which his equations are no longer applicable.

Given the limited data available, future designers wishing to calculate $K_L$ data for fine meshes with open areas << 57% are advised to perform pipe rig tests of their own. The publication of such data, combined with that presented in this Thesis will eventually lead to a suitable dataset from which an appropriate set of empirical equations can be derived.
4.2.2 SCREEN 2

Note $K_L$ for Screen 1 (the coarser screen immediately downstream of the honeycomb) cannot be measured because the contribution from the honeycomb cannot be isolated.

$K_L$ for Screen 1 (the finer screen upstream of the contraction inlet) was predicted as 1.010 and measured as 1.158. This under-estimation of 15% is in broad agreement with errors quoted by Loehrke and Nagib (1972), Schieman (1981), and Groth and Johansson (1988).

4.2.3 SAFETY SCREEN / EXIT GUARD MESH

The safety screen at inlet to the first diffuser and also at exit of the final diffuser was predicted to contribute 24.7% and 1.7% (Table 2-3) of the overall loss of the wind tunnel respectively.

Measured contributions were 28.4% and 2.3% (Table 4-2). $K_L$ for the screen was predicted as 0.199 and measured as 0.251 for the safety screen and 0.241 for the exit guard mesh.

Given that the open area, aperture and wire diameter of the weld mesh were large (being 81.4%, 23mm and 2.5mm respectively) $K_L$ was calculated by treating each wire as an infinitely long cylinder with an assumed drag coefficient ($C_D$) of 1, and neglecting interference effects between wires. This method is believed valid since application of Schieman’s equations for mesh screens presented in section 2.2.3 would have resulted in $K_L$ of only 0.094 for the weld mesh due to excessively high wire Reynolds number and large open area.

Working back from the known loss coefficients results in a wire $C_D$ of 1.26 for the safety mesh and 1.21 for the exit mesh.
4.2.4 REYNOLDS NUMBER SCANS

To aid future designers, Reynolds number scans of each screen (based on wire diameter) are presented in the figures below. Results confirm the trend evident from Schieman’s equations (1981) presented in section 2.2.3, with $K_L$ increasing with reduced wire Reynolds number.

Figure 4-2: Reynolds number sweep for inlet filter mesh

$(D_w = 0.31\text{mm}, \psi = 0.52\text{mm}, \beta = 39\%)$

![Variation of Filter Mesh $K_L$ with Wire Reynolds Number](image)

Figure 4-3: Reynolds number sweep for screen upstream of contraction (Screen 2)

$(D_w = 0.4\text{mm}, \psi = 1.29\text{mm}, \beta = 57\%)$

![Variation of Screen 2 $K_L$ with Wire Reynolds Number](image)
It is of note that the two screens (inlet filter mesh and Screen 2) with low wire Reynolds numbers (<200) exhibit a clear trend of reduced loss with increasing Reynolds number. This is consistent with wire $C_D$ tending to reduce with increasing Reynolds number. However, the dependency of safety screen $K_L$ on Reynolds number is more benign since this screen and the exit guard mesh are already operating at comparatively high Reynolds numbers of ~7000 and ~2000 respectively.
4.3 TURNING VANE LOSSES

As summarised in table 2-3, the two corners of the wind tunnel were predicted to contribute 11.3% of the loss of the overall circuit. Of this the first corner contributed 7.4% and the second corner 3.9%. Measured loss data in table 4-2 shows that the combined contribution from both corners to overall circuit loss is 11.9%, with the first corner contributing 6.8% and the second corner 5.1%.

The merit of using a vane test module to evaluate full-chord (but reduced span) versions of the vanes ultimately used in the wind tunnel has been discussed in section 2.6.

Turning vane data currently available in the literature focuses almost entirely on determining pressure loss coefficient, with test rig data from Klein et al (1930), Collar (1936), Salter (1946) and Winter (1947) showing that thin ¼ circle turning vanes with a space to chord ratio (s/c), produced $K_L$ of between 0.12-0.20.

If $K_L$ had been the sole metric of consideration for the wind tunnel presented in this Thesis, then following the vane test module results, an s/c of 0.237 would have been chosen over an s/c of 0.190, given that pressure loss coefficients were similar and fewer vanes would have been required. However, far-field uniformity data suggested that an s/c of 0.190 (more vanes) gave better control of the inner-bend near-wall flow field and this was therefore selected for use in the wind tunnel.

Vanes used in both corners of the wind tunnel were identical to those employed in the vane test module except that the thickness was increased to 3mm (inside radius, $r$, Figure 2-12, maintained at 245mm) to enhance rigidity and thereby ensure the vanes maintained their profile across the entire span when under load. It also enabled more roundness to be applied to the leading edge to make the vanes less sensitive to inlet flow angularity. This additional thickness was expected to have a negligible effect when making comparisons between the wind tunnel and vane test module results.

As discussed in section 2.6.2, employing an s/c of 0.190 resulted in 25 vanes for the first corner and 28 for the second. When the wind tunnel was run at its target working section velocity of 40m/s, the vanes in the first corner operated at an $Re_c$ of $7.97 \times 10^5$ and those in the second corner at an $Re_c$ of $5.74 \times 10^5$. The latter is the same $Re_c$ as used in the vane test module.

As shown in Figure 2-18, 45° corner fillets were used in both corners to reduce secondary flows caused by interaction between the wall boundary layers.
Figure 4-5 presents velocity profiles measured perpendicular to the tunnel walls at mid span, on the inner and outer bends, 220mm upstream and 80mm downstream of each corner. These locations were as close as it was possible to get to the inlet and exit of each corner due to construction of the wind tunnel and the pitot probe traverse mechanism. The local widths, Y, at the traverse planes shown in Figure 4-5, differ from those quoted in Table 1-1, on account of the diffusers located upstream and downstream of the corners. Positive and negative values of Y relate to the inner and outer bends respectively.

Velocity profiles were determined using a pitot probe referenced to the averaged reading from a ring of 4 wall static tappings. One tapping was located at the mid point of each wall of the wind tunnel in the same plane as the head of the pitot. To enable comparison with the vane test module, the probe was traversed across the vanes in 10mm increments. The construction of the probe and tappings, and the data acquisition and reduction technique employed, were the same as that described in section 2.6.1 for the vane test module.

Figure 4-5: Velocity profiles 220mm upstream and 80mm downstream of the first and second corner

**Corner 1**
To quantify the near wall flow quality entering each corner, boundary layer thickness (δ), displacement thickness (δ*), and momentum thickness (θ) 220mm upstream of the corner were calculated using numerical integration (trapezoidal rule), and are presented in Table 4-3. No downstream data is provided due to the difficulty in confidently determining the true edge of the boundary layer.

Table 4-3: Boundary layer characteristics at inlet to the first and second corners.

<table>
<thead>
<tr>
<th></th>
<th>First Corner Inlet</th>
<th>Second Corner Inlet</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Outer Bend</td>
<td>Inner Bend</td>
</tr>
<tr>
<td>δ (mm)</td>
<td>170</td>
<td>200</td>
</tr>
<tr>
<td>δ* (mm)</td>
<td>39.6</td>
<td>48.5</td>
</tr>
<tr>
<td>θ (mm)</td>
<td>23</td>
<td>29.7</td>
</tr>
</tbody>
</table>

It is clear from Figure 4-5, that despite individual differences, the boundary layers at inlet and exit of both corners are free from reversed flow. As may be expected for a turbulent boundary layer developing under an adverse pressure gradient, the displacement and momentum thicknesses shown in Table 4-3, are significantly higher than for a typical turbulent profile growing in a zero pressure gradient.
Downstream of the first corner, it can be seen that the wakes from the outer bend have mixed out whilst those from the inner bend have not. This is a consequence of the flow having had to travel further along the outer wall than the inner. Variations can be seen in the wake profiles downstream of the inner bend. This may partly be due to greater mixing towards the outer bend and perhaps also due to local inlet non-uniformity resulting in variations in the amount of work done by each vane.

The greater distance traversed by the outer bend flow in passing through the first corner results in the large differential seen in Table 4-3 between the outer and inner bend boundary layer thicknesses at inlet to the second corner. However, although Table 4-3 shows a marked difference between the inner and outer bend boundary layer thickness in the second corner compared to the first, the relative differences in displacement and momentum thicknesses between the inner and outer bend at each corner are not as considerable. The latter is encouraging as it suggests that near wall energy levels between the two bends are reasonably well balanced in each corner. Remnants of the wakes from the first corner can still be seen in the upstream inner bend of the second corner. The different Y-location of these wakes is due to the flow having passed through a diffuser.

To determine $K_L$, the static pressure loss across each corner was measured by means of the averaged reading from a ring of 4 static tappings located 80mm upstream and downstream of each cascade. For each ring, one tapping was located at the mid point of each wall of the wind tunnel. The construction of the tappings and the data acquisition and reduction technique employed, were the same as that described in section 2.6.1 for the vane test module.

The first corner was found to have a pressure loss coefficient, $K_L$, of 0.120 at an $Re_c$ of $7.97 \times 10^5$ and the second corner a $K_L$ of 0.171 at an $Re_c$ of $5.74 \times 10^5$. A Reynolds number sweep of the first corner was performed to determine whether the higher $K_L$ observed in the second corner was Reynolds dependent or due to differences in the boundary layer characteristics shown in Table 4-3.

It can be seen from Figure 4-6, that when the first corner was run at the equivalent second corner $Re_c$ of $5.74 \times 10^5$, its pressure loss coefficient was only 0.116. This suggests that $K_L$ may predominantly be a function of total pressure losses within the boundary layer rather than chord Reynolds number. This is supported by Table 4-3, which shows greater boundary layer, displacement and momentum thicknesses upstream of corner 2 compared to corner 1. It is also apparent from Figure 4-5 that whilst the velocity profile for corner 1 is flatter than for corner 2, the wake profiles within the core of each corner are similar.
Data for the second corner suggests that $K_L$ remains reasonably constant at $\sim 0.170$ down to $4\times 10^5$. 
4.3.1.1  COMPARISON OF VANE TEST MODULE AND WIND TUNNEL RESULTS

The most appropriate wind tunnel corner to use for comparison to the vane test module results presented in section 2.6.1.1 is the first corner, since this has the thinnest boundary layer.

For a common s/c of 0.190, the measured pressure loss coefficient for the vane test module and the first corner of the wind tunnel compare well - being 0.128 and 0.120 respectively. The smaller loss coefficient observed for the wind tunnel cascade might be due to a significantly wider core (25 vanes for the wind tunnel compared to only 4 for the test module) acting to offset the thicker boundary layer observed at inlet to the wind tunnel corner. Using surface roughness to thicken the boundary layer within the test module would not have been possible given the boundary layer thickness at inlet to the first corner is ~150mm at each wall and hence 2/3 the width of the test module.

Comparison of velocity profiles immediately downstream of both the wind tunnel (Figure 4-5) and test module vanes (Figures 2-15 and 2-16), suggests that core flow velocity distribution is similar, with the perturbation between velocity maxima and minima being approximately 0.2U. This is to be expected since (for a given vane geometry) the core flow wake region is primarily a function of the amount of work done by each vane, which is governed by the space to chord ratio.
4.4 DIFFUSER LOSSES

As summarised in table 2-3, the three diffusers of the wind tunnel were predicted to contribute 36.9% of the loss of the overall circuit. Of this, the first diffuser contributes 17.7%, the second diffuser 4.8% and the final diffuser 14.4%.

The loss associated with the first diffuser is large due to its high inlet dynamic head, and the final diffuser loss is substantial because the kinetic energy contained within its exit flow is dumped to atmosphere rather than being utilised in a downstream duct.

Measured loss data in table 4-2 shows that the combined contribution from the three diffusers to overall circuit loss is 26.2%, with the first diffuser contributing 8.9%, the second diffuser 3.9%, and the final diffuser 13.4%. Possible reasons for the discrepancy between predicted and measured losses will be discussed below.

A comparison of measured and predicted $C_p$ values for each diffuser are quoted in table 4-1.

4.4.1 FIRST DIFFUSER

The predicted pressure recovery coefficient for the first diffuser, (defined in section 2.5 and taken from figure 2-11) suggests a $C_p$ of 0.483. Measured $C_p$ was 0.563. ESDU data presented in Figure 2-11 is for diffusers with similar inlet aerodynamic blockage (~1.2%) to that at the first diffuser. Inlet aerodynamic blockage ($B$) is defined as the ratio between boundary layer displacement thickness ($\delta^*$) and hydraulic radius ($R$):

$$B = \frac{\delta^*}{R}$$

In his review of available literature, Klein (1981), concluded that $C_p$ is a maximum at zero inlet blockage, and drops by around 12% to reach a minimum as inlet blockage increases to approximately 14% for fully developed flow. Consequently, variances in inlet blockage cannot account for the substantial increase in measured $C_p$ for the first diffuser compared with that predicted by ESDU.

Japiske (1985) reports that due to the significant data scatter from a variety of experimental investigations:
“... any interpretation of diffuser flow fields using only inlet aerodynamic blockage is subject to substantial error...”

He suggests that Reynolds number, turbulence intensity and boundary layer quality should also be considered.

A Reynolds number scan (based on inlet hydraulic diameter), summarised in figure 4-8, suggests that \( C_p \) is relatively insensitive to inlet Reynolds number.

Figure 4-8: First diffuser Reynolds number sweep

Japiske suggests that turbulence may play a key role in affecting diffuser \( C_p \):

“...for no macroscopic change in velocity profile increasing turbulence generally improves diffuser performance...turbulence may be produced in two ways. By model wakes or by grids in the flow. One is beneficial and one not, because the former is accompanied by a non-uniform velocity profile and inlet distortion and the latter is not...”

The safety screen (discussed in section 2.4.5) is, of course, located at the inlet of the first diffuser. With a wire diameter of 2.5mm and an aperture of 23mm, this screen is likely to generate significant turbulent mixing without the wake coalescence (discussed in section 2.2.1) observed with high solidity screens. This turbulent mixing may contribute to the improvement in first diffuser \( C_p \).

Whilst an on-off test with the safety screen was not performed, an estimation of the likely improvement in \( C_p \) due to the safety screen may be derived with help from measured data for the
second diffuser (discussed in the following section). Table 4-1 shows that measured $Cp_r$ for the second diffuser is 7% higher than quoted by ESDU in figure 2-11. This compares with measured $Cp_r$ for the first diffuser being 17% higher than quoted by ESDU. A crude analysis may therefore be to ignore the effects of inlet blockage and assume that the safety screen improves first diffuser $Cp_r$ by ~10% (equating to a 6.8% reduction in overall circuit loss). Ignoring the effects of inlet blockage may be reasonable since blockage will be slightly higher for the second diffuser than for the first and it is known from Klein (1981) that $Cp_r$ reduces with increased blockage.

This being the case, the downside (in terms of loss) of placing a safety screen at the end of the working section may not be as great as it first appears. Whilst measured data in table 4-2, shows that the safety screen contributes 28.4% of the overall circuit loss, this does not account for any improvement to the performance of the first diffuser. It may also be argued that the safety screen will assist in mixing out model wakes and hence desensitise the first diffuser to the presence of models in the working section. Further work in this area may be of merit since results would help future designers choose the most appropriate location for the high loss safety mesh.
4.4.2 SECOND DIFFUSER

Of the three wind tunnel diffusers, the second diffuser correlates most closely with $C_p$, data proposed by ESDU (figure 2-11 and table 4-1), with measured $C_p$ being 7% higher than predicted.

This is perhaps unsurprising, given that unlike the first diffuser, the second diffuser does not have a screen positioned at its inlet and hence its $C_p$ does not benefit from the resultant turbulent mixing. Additionally, unlike the third diffuser (discussed in the section below), $C_p$ for the second diffuser does not benefit from inlet swirl. Figure 4-9, suggests that the discrepancy in $C_p$ is not due to a Reynolds number dependency.

In producing its diffuser performance charts, ESDU have collated diffuser design data available in the literature, and Japiske (1985) reports that this suffers from significant data scatter due to the variety of experimental investigations.

Wallis’ perspective (1983) on diffuser design charts is that they are:

“...largely empirical and therefore dependant on whether someone else has tested your configuration with your expected inlet flow profiles...”
4.4.3 THIRD DIFFUSER

As described in section 2.5.3, the third or final wind tunnel diffuser is a conical rather than plane-walled design. This diffuser was predicted to contribute 14.4% (table 2-3) of the overall loss of the wind tunnel and was measured as contributing 13.4% (table 4-2). This error of 1% is in spite of the large difference between predicted and measured $C_p$, discussed below, and is due to the relatively low inlet dynamic head.

Being located immediately downstream of the fan this diffuser suffers from any non-uniformity generated by the fan but benefits from the 5° of swirl generated by its exit guide vanes. McDonald et al (1971) and Senoo et al (1978) both report that inlet swirl improves the $C_p$ of conical diffusers by energising the boundary layer and reducing boundary layer displacement thickness.

Their experimental results suggest that swirl is optimum when the mean swirl angle, $\phi$, is equal to the total included angle, $2\theta$, and that in these conditions, an increase in $C_p$ of around 15% may be achieved. The diffuser was designed with a $2\theta$ of 6.0° and to account for non-uniformity at inlet a relatively high value of $\alpha_1$ of 1.1 was adopted since no data was available beforehand.

A counter argument may be that a swirl velocity component will result in development of a cross flow boundary layer, which, when added to the axial boundary layer may result in a boundary layer of greater net thickness than the no-swirl condition.

Predicted $C_p$ for the third diffuser was derived from Figure 2-11 as 0.538. Measured $C_p$ was 0.643 and hence 20% higher than anticipated. From the overview above, it may be concluded that inlet swirl accounts for ~15% of the increase in measured $C_p$. The remaining 5% was thought to be attributable to Reynolds effects, however, whilst a Reynolds number scan (figure 4-10) showed more data scatter than for the other two diffusers, $C_p$ was relatively consistent down to $2\times10^6$. The data scatter may be due to non-uniformity downstream of the fan and gusts blowing at the exit. Locating the static tappings on a curved surface and making them flush was also more difficult than on the plane-walled diffusers.
Figure 4-10: Third diffuser Reynolds number sweep

![Graph showing the variation of measured Diffuser 3 Cpr with Reynolds Number based on Inlet Hydraulic Diameter.](image)
A comparison of the measured and predicted variation of total pressure along the tunnel circuit (Figure 4-11) provides a visual summary of the following key points:

- Of the screens in the settling chamber, the inlet filter mesh suffers from the worst under-estimation of loss (section 4.2.1).
- Whilst loss from the safety screen was significantly under-predicted, so were benefits to the first diffuser arising from turbulent mixing aft of the safety screen (section 4.4.1).
- The pressure rise required by the fan was less than predicted and most likely due to smaller than expected leakage losses (section 4.1).

### 4.5 GRAPHICAL SUMMARY OF CIRCUIT TOTAL PRESSURE VARIATION

Figure 4-11: Measured vs predicted variation of total pressure along the tunnel circuit

*Note $\Delta H$ does not return to zero since losses that were not measured (contraction etc) have not been plotted.*
5 CONCLUSIONS AND IMPLICATIONS

- The performance of the wind tunnel presented in this Thesis has endorsed the selection of a novel indraft configuration to meet the design brief whilst overcoming severe spatial and financial constraints. The implication of this is that future designers should not shy away from such a configuration by default.

- Compared to a traditional straight-through (Eiffel) open circuit design, the idea of bending an Eiffel tunnel into a horseshoe and connecting the inlet and exit to atmosphere, successfully prevented pressure loading of the building within which the tunnel was housed and also eliminated the problem of exhausting a fast flowing jet within the confines of a building.

- Compared to trying to ‘squeeze’ a traditional closed return (Gottingen) design into the available footprint, the horseshoe indraft tunnel presented in this Thesis effectively removed corners 3 and 4, thereby allowing the remaining the tunnel modules to more closely adhere to the design guidelines discussed in Chapter 2. Specifically, a larger contraction ratio was possible and costs associated with a heat exchanger were eliminated. It is however acknowledged that with test section static pressure being below atmospheric, the future inclusion (and sealing) of a moving ground would be difficult (but by no means impossible).

- Following a literature survey, it was concluded that whilst valuable design guidelines and rules of thumb on general low-speed wind tunnel design did exist, the dataset was by no means exhaustive. There appeared, therefore, to be room alongside these established texts for a Thesis, which acted as a complete reference point on information required for the pragmatic design of each component of a low speed wind tunnel - irrespective of its configuration.

- Despite no published data being available on comparable indraft wind tunnels, this Thesis has shown that it was possible to apply concepts used for the much larger and differently configured NASA Ames 80ft x 120ft indraft tunnel - specifically, the use of a high loss inlet filter mesh to damp ambient winds. Calibration and commissioning data has shown that the indraft tunnel presented in this Thesis possesses sufficient flow quality and repeatability to be used for aeronautical and automotive teaching in the presence of ambient winds up to 13% of the test section velocity. Tests in the presence of stronger ambient winds would compliment the current dataset. In the working area of the jet, at 40m/s, deviation from the mean axial
velocity was within 0.3% and averaged axial turbulence intensity was ~0.15%. $C_D$ for a 25% scale Davis model repeated to 2 drag counts. Consequently, no plans were made to include further flow conditioning devices.

- Once measured loss coefficient ($K_L$) and diffuser pressure recovery coefficient ($C_{p_r}$) data were fed-back into the design spreadsheet, it became apparent that a 2.5% leakage allowance was more realistic for a well-sealed tunnel than the guideline of 10% proposed by the major texts. Future designers should be mindful of this since the additional power budget may make other aspects of their design (e.g inclusion of further flow conditioning devices) more feasible.

- Roof $C_p$ data with a 25% Davis model (5.9% blockage) indicated that the static ports of the tunnel pitot static ought to be a minimum of 0.55 hydraulic working section diameters upstream of the leading edge of the model to not be influenced by the model’s static pressure signature.

- Roof $C_p$ data suggested the parallel duct at the trailing edge of the contraction be increased from 1 hydraulic radius to 1 hydraulic diameter to better allow pressure gradients to decay prior to the model.

- The minimum working section L/D of 2, proposed by established texts appears to be too low. The author proposes 2.5, since although an L/D of 2 is sufficient to allow the trailing edge of the model to be located no closer to the end of the working section than twice the square root of its base area, roof $C_p$ data suggests that a further hydraulic radius be provided to allow for wake recovery.

- The severe under prediction (46%) of $K_L$ for the high-loss inlet filter mesh (likely due to wake coalescence experienced by a high solidity screen) and the significant contribution this screen makes to overall circuit loss implies that future designers ought to conduct pipe rig tests on screens with open areas <<57%.

- $K_L$ for the low solidity safety screen was calculated by treating each wire as a cylinder with a $C_D$ of 1, and neglecting interference effects between wires. However, measured data suggests a wire $C_D$ of ~ 1.26 to be more appropriate.
• This Thesis has shown that $K_L$ should not be the sole metric by which to select the space-to-chord ratio ($s/c$) of corner cascades. The importance of measuring uniformity far downstream of a test cascade (rather than just immediately downstream) has been demonstrated - with far field flow quality improving with more closely spaced cascades ($s/c$ of 0.190 rather than 0.237). Since far-field flow quality aft of corner 1 effectively becomes the inlet condition for corner 2, it is evident that improvement to near-wall, far-field uniformity for the upstream corner ultimately leads to reductions in $K_L$ for the downstream corner.

• Whilst the safety screen would ideally have to be removed to prove the hypothesis it is suggested that turbulent mixing aft of the safety screen (located at the end of the working section) appears to offer a $\sim$10% $C_p$ improvement to the first diffuser.
REFERENCES

“…stealing from one is plagiarism - stealing from many is research…”

(Anon)


ESDU 76027. Introduction to Design and Performance Data for Diffusers.

ESDU 80037. Pressure Recovery of Axisymmetric Intakes at Subsonic Speeds.


References


Salter, C; Experiments on Thin Turning Vanes; Reports and Memoranda No 2469 (Aerodynamics Division N.P.L); 25th October 1946.


Winter, K. G; Comparative Tests of Thick and Thin Turning Vanes in the Royal Aircraft Establishment 4 x 3-ft Wind Tunnel; Reports and Memoranda No 2589; August 1947.


Internet Resources

http://history.nasa.gov/SP-440/ch7-2.htm (history of wind tunnels)

www.aerolab.com (suppliers of small open circuit wind tunnels)

www.nrc-cnrc.gc.ca (Propulsion and Icing Wind Tunnel - National Research Council Canada)
Flow chart 1 (Key decisions)

START - Flow 1
(Key Decisions)

DEFINE KEY TEST SECTION DATA 
(SEC 2.4)

Aspect Ratio 
(minimum of yaw range)

Cross Sectional 
Area (minimum of 
blockage)

Maximum Velocity 
(minimal of adequate 
Reynolds number)

ASSESS POWER 
AND BUDGET FEASIBILITY

Fan Power 
(calculated from sec 2.4)

Balance Budget 
(matching range and 
accuracy data in Table 3.3)

Budget for rest of tunnel 
(= equivalent to 
fan=balance budgets)

NO

SUFFICIENT 
PPOWER AND 
BUDGET

YES

END - go to Flow 2 
(Outline Design)