Suppression of junction flow effects in half model wind tunnel testing

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

Additional Information:

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

Metadata Record: https://dspace.lboro.ac.uk/2134/11568

Publisher: © Abdullah Malik

Please cite the published version.
This item was submitted to Loughborough University as a PhD thesis by the author and is made available in the Institutional Repository (https://dspace.lboro.ac.uk/) under the following Creative Commons Licence conditions.

![Creative Commons Licence](https://creativecommons.org/licenses/by-nc-nd/2.5/)

You are free:
- to copy, distribute, display, and perform the work

Under the following conditions:

**Attribution.** You must attribute the work in the manner specified by the author or licensor.

**Noncommercial.** You may not use this work for commercial purposes.

**No Derivative Works.** You may not alter, transform, or build upon this work.

- For any reuse or distribution, you must make clear to others the license terms of this work.
- Any of these conditions can be waived if you get permission from the copyright holder.

Your fair use and other rights are in no way affected by the above.

This is a human-readable summary of the [Legal Code (the full license)](http://creativecommons.org/licenses/by-nc-nd/2.5/)

For the full text of this licence, please go to:
http://creativecommons.org/licenses/by-nc-nd/2.5/
Loughborough University

Abdullah Malik

Suppression of Junction Flow Effects in Half Model Wind Tunnel Testing

Supervisor: Dr. P.M. Render

AERONAUTICAL AND AUTOMOTIVE ENGINEERING

Submitted as partial fulfilment for the award of the Doctor of Philosophy
August 2012
Loughborough University

AERONAUTICAL AND AUTOMOTIVE ENGINEERING

Submitted as partial fulfilment of the award of the Doctor of Philosophy

Abdullah Malik

Suppression of Junction Flow Effects in Half Model Wind Tunnel Testing

Supervisor: Dr. P.M. Render

August 2012

Submitted as partial fulfilment of the award of the Doctor of Philosophy
© Loughborough University, 2012. All rights reserved. No part of this publication may be reproduced without the written permission of the copyright holder.
Half model testing is considered a valuable wind tunnel technique that offers many benefits over conventional full span testing. The technique suffers from aerodynamic losses due to flow separations on the model surfaces near the model/floor junction. Computational Fluid Dynamics, employing the Spalart-Allmaras turbulence model, and experimental investigations were carried out to evaluate the losses and to investigate the effect of localised suction on the junction flows. The wind tunnel model used was a rectangular and untwisted wing having a NASA LS(1)-0413 cross section and with a physical aspect ratio of 3. Tests were conducted at 10.0° incidence at a Reynolds number of 0.44 x 10^6. Aerodynamic performance of the wind tunnel half model was obtained by surface flow visualisation and pressure measurements on the wing surface in the junction region. CFD predictions showed significantly large losses compared to the experimental findings and therefore CFD predicted significant influence and benefits of suction. These were seen as elimination of the model surface separation and also recovery of the wing surface pressure distributions. In contrast to this, experiments showed much smaller separation than CFD without suction and applying suction in experiments, showed only a marginal effect on the flow separations, which also further deteriorated the pressure distributions. Future CFD studies on junction flows should be conducted using more advanced turbulence models such as Large Eddy Simulations (LES). In addition, to validate these CFD studies, velocity and turbulence measurements in the wing/floor junction region are also needed.
Acknowledgement

After having spent years studying for my PhD in Loughborough University I can say it has been a wonderful although sometimes a trying learning experience working in the Department of Aeronautical and Automotive Engineering at the Loughborough University. I owe an appreciation and gratitude to the following people for their contribution towards this project.

First & foremost I would like to express my gratitude to Dr. P.M. Render my Supervisor, for his guidance, supervision and sharing his vast experience of aircraft design and aerodynamics. I owe him great credit for his encouragement that has kept me focused throughout.

I would also like to thank & acknowledge Prof. Jim McGuirk who has been my Director of Research who provided guidance in difficult times. Professor Jim McGuirk was always there to provide advice whenever needed and was never too much trouble for him. I would like to extend a huge hand of gratitude towards him.

Dr. B.P. Vale, Assistant Registrar and her team (Alison Howells, Tracy Bergstrom and Sally Alldritt to name some of them) at research office, were always on hand to help me during tough & testing times of my course. Their support & encouragement whenever needed by myself never went unnoticed and always helped to boost my morale.

I would also like to thank Prof. Richard Stobart & Rui Chen for helping me when they could. They were not at the forefront of my course but were always on hand for advice. Many thanks.

Acknowledgments also to Dr. Martin Passmore, for always being available for any advice needed but also was there to lend me his equipment whenever needed and never was this too much trouble for him.

My gratitude also goes out to Rob Hunter, Stacey Prentice, Peter Stinchcombe and Keith Coulthard for their huge efforts in model manufacturing and setting up
Acknowledgement

the models in the wind tunnel for me, this was always a huge help to me. Thank you.

Thank you also to Harshad Purohit, Martin Cramp and Pardeep Karia in the electronic workshop. A lot of what we do in the wind tunnel would not be as easy if not for you guys.

My acknowledgments also go to Grenville Cunningham and Norman Randall for their advice whenever I approached them also for doing the jobs on late Fridays afternoons many times.

May I also thank Robert Flint, AAE I.T. Systems Manager, for the many times he was required help me to fix my system and help solve my I.T. issues who never found it too much trouble.

Sue Rolland, Gloria Brown, Gill Youngs and Pat Griffin were always a huge help when it came to my everyday administrative issues. Thank you ladies.

Great gratitude is also extended to Martin Matthews and Andy Deane from Student Support for their moral support & just generally being there for me outside of their contractual duties. Their encouragement was a huge boost for me.

Lydia Jowsey, Robert Littlewood, Joshua Fuller and Daniel Wood, my colleagues from the Applied Aerodynamics research group I would like to thank these guys for their undivided support with the experiments when needed with the wind tunnel testing.

I would also like to thank my colleagues Philipp Romelt and Professor Joseph Loughlan and for helping me in stressing calculations.

My gratitude also to Dr. Perviz Behrouzi for his invaluable advice on wind tunnel testing and encouragement.
# Table of Contents

Abstract.......................................................................................................................... i  
Acknowledgement............................................................................................................ ii  
1 Introduction ...................................................................................................................... 1  
   1.1 Half Model Wind Tunnel Testing................................................................. 2  
   1.2 Review of Half Model Wind Tunnel Testing from Literature ............... 3  
      1.2.1 Barberis et al. Experiment^6 ................................................................. 3  
      1.2.2 Philips et al. Experiment^7 ................................................................. 5  
      1.2.3 Mendelsohn & Polhamus Experiment^8 ........................................ 6  
      1.2.4 Bernstein & Hamid Experiment^4 .................................................... 8  
      1.2.5 Wood & Westphal Experiment^9 ...................................................... 11  
      1.2.6 Applin Experiment^11 ................................................................. 12  
      1.2.7 Formation of the Junction Horseshoe Vortex ......................... 14  
   1.3 Techniques Used for Half Wing Testing ..................................................... 21  
      1.3.1 Attempts to reduce the onset Boundary Layer Thickness ...... 21  
      1.3.2 Attempts to Remove the Horseshoe Vortex ............................... 28  
      1.3.3 Discussion of Suction Experiments of Philips et al. ^7 and Barberis et al. ^6 31  
   1.4 Aim and Objectives of the Project ................................................................. 32  
2 CFD Methodology Selection and Validation ......................................................... 34  
   2.1 CFD Computational Details-2D Aerofoil Flow ...................................... 34  
      2.1.1 Grid Independence Study ................................................................. 36  
      2.1.2 Grid sensitivity study ...................................................................... 39  
      2.1.3 Final Grid Selected ......................................................................... 42  
   2.2 Analysis of Predicted Results-2D Aerofoil Flow .................................. 42  
      2.2.1 Summary-2D Aerofoil flow ............................................................... 56  
   2.3 Evaluation of Spalart-Allmaras Model for Junction Flow Predictions . 56  
   2.4 Computational Details-3D Junction Flow ............................................. 58  
   2.5 Analysis of predicted Results-3D Junction Flow .................................. 60  
   2.6 Summary-3D Junction Flow .................................................................. 70  
3 CFD Predictions of Lifting Half Wing Characteristics................................. 71  
   3.1 3D Computational Details ................................................................. 71  
      3.1.1 Half Wing and Ideal Half Wing predictions ................................. 74  
      3.1.2 CFD Results and Analysis ............................................................... 75  
   3.2 Summary-3D Junction Flow Predictions for a Lifting Half Wing ...... 90
# Table of Contents

4  CFD Predictions of Junction Flow Modification using Localised Suction ................................................................. 91
  4.1 Half Wing CFD Predictions with Suction ................................................. 91
  4.2 Flow Conditions and Grid Description .................................................... 93
    4.2.1 Analysis of the Results ........................................................................ 95
  4.3 Summary- CFD Predicted Effect of Suction on Junction Flow for a Lifting Half Wing ......................................................... 104

5  Test Facilities and Experimental Setup ............................................. 105
  5.1 Half Wing Design and Geometry ............................................................. 106
  5.2 Half Wing Measurements ......................................................................... 108
  5.3 Suction System ......................................................................................... 111
    5.3.1 Calibration of the Suction System .......................................................... 111
    5.3.2 Installation of Suction System in Wind Tunnel ...................................... 114
  5.4 Flow visualisation ..................................................................................... 117

6  Wind Tunnel Results and Comparison with CFD ................................ 119
  6.1 Half Wing Flow Patterns Without Suction, Re=0.89 x 10^6 .......... 119
  6.2 Half wing flow features, Re=0.44 x 10^6 .............................................. 123
  6.3 CFD Predictions at lower Re=0.44 x 10^6 .............................................. 125
  6.4 Comparison of Experiment and CFD, No-Suction, Re=0.44 x 10^6 .... 126
  6.5 CFD Predictions with suction at Re=0.44 x 10^6 .................................. 129
  6.6 The Effects of Suction on Half Wing Junction Flow .............................. 131
  6.7 Effect of Suction Hole Location ............................................................... 138
  6.8 Comparison of Experiment and CFD with suction, Re=0.44 x 10^6 ... 140
  6.9 Summary .................................................................................................. 142

7  Summary, Conclusions and Suggestions for Future Work ............ 143
  7.1 Summary and Conclusions ....................................................................... 143
  7.2 Suggestions for Future Work .................................................................... 144

8  References .................................................................................................. 146

9  Appendix A - Modified LS(1)-0413 Properties .................................... 154

10 Appendix B - Engineering Drawings ...................................................... 155
List of Figures

Fig. 1.1: Illustration of Method of Images\textsuperscript{5} ................................................................. 3
Fig. 1.2: Top view of Barberis et al. arrangement\textsuperscript{6} ............................................................. 4
Fig. 1.3: Vortex core developed in symmetry plane, Barberis et al. experiment\textsuperscript{6,4} .................. 4
Fig. 1.4: Top view of Philips et al. arrangement\textsuperscript{7} ................................................................. 5
Fig. 1.5: Velocity vectors in axial plane at 0.24c, Philips et al. experiment\textsuperscript{7}....................... 6
Fig. 1.6: Cp distribution on NACA 651-012 half wing, 3.0\textdegree incidence\textsuperscript{8} ............... 7
Fig. 1.7: Cp distribution at 3.0\textdegree incidence, Zoomed-in view\textsuperscript{8} ........................................ 7
Fig. 1.8: Cp distribution on NACA 651-012 half wing, 12.0\textdegree incidence\textsuperscript{8} .............. 8
Fig. 1.9: Cp distribution at 12.0\textdegree incidence, Zoomed-in view\textsuperscript{8} ............................. 8
Fig. 1.10: Cp distribution on NACA 0015 half wing, 3.0\textdegree incidence\textsuperscript{4} ................... 9
Fig. 1.11: Cp distribution on NACA 0015 half wing, 12.0\textdegree incidence\textsuperscript{4} ............. 10
Fig. 1.12: NACA 0015 half wing sectional characteristics\textsuperscript{4} ............................................... 11
Fig. 1.13: Cp distribution on NACA 0012 half wing, 4.0\textdegree incidence\textsuperscript{11} ................. 13
Fig. 1.14: Cp distribution on NACA 0012 half wing, 10.0\textdegree incidence\textsuperscript{11} ............... 13
Fig. 1.15: Cp distribution on NACA 0012 half wing, 18.0\textdegree incidence\textsuperscript{11} ................. 14
Fig. 1.16: Sketch of flow patterns in impingement plane ............................................................ 15
Fig. 1.17: Sketch of flow patterns on tunnel floor............................................................................ 16
Fig. 1.18: Flow patterns sketch in symmetry neighbouring plane ................................................. 16
Fig. 1.19: Junction flow patterns, Olcmen & Simpson Experiment\textsuperscript{18} .................................. 17
Fig. 1.20: Different cross-section shapes investigated by Mehta\textsuperscript{16} .................................... 19
Fig. 1.21: Flow vectors in axial plane at 175 mm from leading edge\textsuperscript{16} ............................. 20
Fig. 1.22: Reduction of boundary layer thickness with splitter plate\textsuperscript{20} ............................... 22
Fig. 1.23: Flow patterns on NACA 4415 wing, 21.0\textdegree incidence\textsuperscript{21} ............................. 23
Fig. 1.24: Separation size for different splitter plate length, 21.0\textdegree incidence\textsuperscript{21} ........ 23
Fig. 1.25: Flow patterns for half wing with splitter plate, 21.0\textdegree incidence\textsuperscript{21} .......... 24
Fig. 1.26: Cp distribution at height of 71 mm (3.8 % of span), 21.0\textdegree incidence\textsuperscript{21} ....... 24
.......................................................................................................................................................... 25
Fig. 1.27: Sketch of Distributed Suction Setup, Bippes & Turk Experiment\textsuperscript{22} .................. 25
Fig. 1.28: Cp distribution at height of 147 mm (7.9 % of span), 21.0\textdegree incidence\textsuperscript{22} ...... 26
.......................................................................................................................................................... 26
Fig. 1.29: Floor boundary layer reduction by upstream suction\textsuperscript{20} ..................................... 27
Fig. 1.30: Suction effect on NACA 0012 half wing, Height= 97.4 mm\textsuperscript{11} ....................... 27
Fig. 1.31: Suction effect on NACA 0012 half wing, Height= 1391 mm\textsuperscript{11} ..................... 28
Fig. 1.32: Top view of Philips et al. setup with suction\textsuperscript{7} .................................................... 28
List of Figures

Fig. 1.33: Effect of suction on junction velocity vectors, Philips et al. Experiment ................................................................. 29
Fig. 1.34: Top view of Barberis et al.’ arrangement with suction .......................... 30
Fig. 1.35: Effect of suction on vortex core in symmetry plane .......................... 31
Fig. 2.1: Solution domain for 2D aerofoil and mesh analysis ............................ 36
Fig. 2.2: Grids with varying far-field boundaries ........................................... 37
Fig. 2.3: Grids with varying traverse grid points .......................................... 38
Fig. 2.4: Cp distribution for Modified LS(1)-0413 predicted by XFOIL code ........... 39
Fig. 2.5: Various grids for grid independence study ....................................... 40
Fig. 2.6: G0-Grid selected for final CFD Studies .......................................... 42
Fig. 2.7: Wall y+ plot, Re=2 x 10^6 ............................................................ 43
Fig. 2.8: Flow separation predicted behind trailing edge, Re=2 x 10^6 ............... 43
Fig. 2.9: Drag Coefficient, Re=2 x 10^6 .................................................... 44
Fig. 2.10: Lift coefficient, Re=2 x 10^6 ...................................................... 45
Fig. 2.11: Pitching moment coefficient, Re=2 x 10^6 .................................... 45
Fig. 2.12: XFOIL prediction of transition location, Re=2 x 10^6 ...................... 46
Fig. 2.13: Setup for modelling transition in RANS CFD (0° incidence) ............... 47
Fig. 2.14: 10.0° incidence-CFD (Natural transition), Re=2 x 10^6 .................... 48
Fig. 2.15: Predicted Drag Coefficient, Re=2 x 10^6 .................................... 49
Fig. 2.16: Predicted Lift Coefficient, Re=2 x 10^6 ....................................... 50
Fig. 2.17: Predicted Pitching Moment Coefficient, Re=2 x 10^6 ..................... 50
Fig. 2.18: 10.0° incidence, CFD (Natural Transition) predictions, Re=2 x 10^6 ................................. 51
Fig. 2.19: 12.0° incidence, CFD (Natural transition), Re=2 x 10^6 ................. 52
Fig. 2.20: Predicted Drag coefficient, Re=1 x 10^6 ...................................... 53
Fig. 2.21: Predicted Lift coefficient, Re=1 x 10^6 ........................................ 53
Fig. 2.22: Predicted Pitching moment coefficient, Re=1 x 10^6 ....................... 54
Fig. 2.23: Experimental Lift and Drag Coefficient ................................. 31 54
Fig. 2.24: Predicted Lift and Drag Coefficient, Spalart-Allmaras ................... 55
Fig. 2.25: Experimental Pitching Moment Coefficient .............................. 31 55
Fig. 2.26: Predicted Pitching Moment Coefficient, Spalart-Allmaras ............... 56
Fig. 2.27: Rood Aerofoil Section ......................................................... 58
Fig. 2.28: Sketch of experimental and computational set-up ........................... 59
Fig. 2.29: Grid around the Rood aerofoil ............................................... 60
Fig. 2.30: Three dimensional flow domain ............................................... 60
Fig. 2.31: Boundary layer development in symmetry plane ......................... 61
Fig. 2.32: Boundary layer development in symmetry plane (Zoomed-in view) . 61
Fig. 2.33: Predicted patterns on tunnel floor for Rood half wing ................... 62
Fig. 2.34: Illustration of Nodal point of attachment ........................................ 63
Fig. 2.35: Secondary vortex in Rood junction .................................................. 63
Fig. 2.36: Predicted velocity vectors in the symmetry plane ............................ 64
Fig. 2.37: Junction horseshoe vortex in Rood junction .................................. 64
Fig. 2.38: Measured velocity vectors the in symmetry plane .......................... 65
Fig. 2.39: Axial velocity contours in symmetry plane .................................... 65
Fig. 2.40: Axial planes of interest ................................................................. 66
Fig. 2.41: Axial velocity contours in x = 0.179c plane .................................. 67
Fig. 2.42: Axial velocity contours in x = 0.64c plane ................................... 67
Fig. 2.43: Axial velocity contours in x = 1.05c plane ................................... 68
Fig. 2.44: Skin friction lines on wing surface ................................................ 69
Fig. 2.45: Cp distribution at various spanwise stations .................................. 69
Fig. 2.46: Overview of junction flow prediction .............................................. 70
Fig. 3.1: Grid distribution in x-z plane .......................................................... 72
Fig. 3.2: Grid distribution in x-y plane .......................................................... 73
Fig. 3.3: Domain size for half wing computations ........................................... 74
Fig. 3.4: Boundary layer profile in an empty tunnel, Re=0.86 x 10^6 ............ 75
Fig. 3.5: Predicted flow patterns for ideal half wing, 10.0^0 incidence ............ 75
Fig. 3.6: Flow patterns on wing surface of ideal half wing, 10.0^0 incidence .... 76
Fig. 3.7: Predicted flow patterns on floor for half wing, at 0^0 incidence ......... 77
Fig. 3.8: Junction horseshoe vortex at 0^0 incidence ..................................... 78
Fig. 3.9: 3D view of oncoming flow deflected from leading edge, 0^0 incidence 78
Fig. 3.10: Secondary vortex, 0.0^0 incidence ............................................... 79
Fig. 3.11: Zoomed-in top view of two vortices, 0^0 incidence ......................... 79
Fig. 3.12: Predicted flow patterns on half wing surface at 0.0^0 incidence ....... 80
Fig. 3.13: Zoomed-in view of leading edge, 0^0 incidence ............................. 81
Fig. 3.14: Zoomed-in view of trailing edge, 0^0 incidence ............................. 81
Fig. 3.15: Flow pattern in wake, 0^0 incidence ............................................. 82
Fig. 3.16: Cp distribution at height=75 mm (7.9 % Span), 0^0 incidence ........ 83
Fig. 3.17: Cp distribution at height=10 mm (1.1 % Span), 0^0 incidence .......... 83
Fig. 3.18: Cp distribution at height=5 mm (0.5 % Span), 0^0 incidence .......... 83
Fig. 3.19: Junction horseshoe vortex, 10.0^0 incidence ............................... 84
Fig. 3.20: Secondary vortex, 10.0^0 incidence ............................................ 85
Fig. 3.21: Predicted flow patterns on floor for half wing, 10.0^0 incidence ...... 85
Fig. 3.22: Predicted flow patterns above the floor, 10.0^0 incidence ............... 86
Fig. 3.23: Flow patterns on half wing suction surface, 10.0^0 incidence ........... 87
Fig. 3.24: Flow patterns on the half wing pressure surface, 10.0^0 incidence ... 88
<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.25</td>
<td>Cp distribution at height=5 mm (0.5% span), 10.0° incidence</td>
</tr>
<tr>
<td>3.26</td>
<td>Cp distribution at height=75 mm (7.9% span), 10.0° incidence</td>
</tr>
<tr>
<td>3.27</td>
<td>Cp distribution at height=475 mm (50% span), 10.0° incidence</td>
</tr>
<tr>
<td>4.1</td>
<td>Illustration of suction and boundary layer mass flow</td>
</tr>
<tr>
<td>4.2</td>
<td>Grid around the suction hole</td>
</tr>
<tr>
<td>4.3</td>
<td>Grid in horizontal plane at midspan (z=0.65 m)</td>
</tr>
<tr>
<td>4.4</td>
<td>Flow patterns on the floor with suction</td>
</tr>
<tr>
<td>4.5</td>
<td>Flow patterns in the stagnation plane</td>
</tr>
<tr>
<td>4.6</td>
<td>Flow patterns in the stagnation plane (Zoomed-in view)</td>
</tr>
<tr>
<td>4.7</td>
<td>Vortices formed in the junction around wing, 10.0° incidence</td>
</tr>
<tr>
<td>4.8</td>
<td>Flow patterns in the chordwise plane, 10.0° incidence</td>
</tr>
<tr>
<td>4.9</td>
<td>Flow patterns on wing surface, 10.0° incidence</td>
</tr>
<tr>
<td>4.10</td>
<td>Close-up of flow patterns on suction surface, 10.0° incidence</td>
</tr>
<tr>
<td>4.11</td>
<td>Close-up of flow patterns on pressure surface, 10.0° incidence</td>
</tr>
<tr>
<td>4.12</td>
<td>Turbulent viscosity contours at x/c=1</td>
</tr>
<tr>
<td>4.13</td>
<td>Cp distribution comparison at height=5 mm (0.5% span)</td>
</tr>
<tr>
<td>4.14</td>
<td>Cp distribution comparison at height=75 mm (7.9% span)</td>
</tr>
<tr>
<td>4.15</td>
<td>Cp distribution comparison at height=475 mm (50% span)</td>
</tr>
<tr>
<td>5.1</td>
<td>Model of Loughborough University indraft wind tunnel</td>
</tr>
<tr>
<td>5.2</td>
<td>Test Section Reference Axis System</td>
</tr>
<tr>
<td>5.3</td>
<td>View of test section looking into the downstream</td>
</tr>
<tr>
<td>5.4</td>
<td>Optimised wing cross section geometry</td>
</tr>
<tr>
<td>5.5</td>
<td>Sketch of model setup for basic half wing tests</td>
</tr>
<tr>
<td>5.6</td>
<td>Sketch of half wing setup (without suction)</td>
</tr>
<tr>
<td>5.7</td>
<td>Half wing in test section</td>
</tr>
<tr>
<td>5.8</td>
<td>Embedded tubes routed out through interface block</td>
</tr>
<tr>
<td>5.9</td>
<td>Routed out pressure tubes connected to ‘scanner disconnect’</td>
</tr>
<tr>
<td>5.10</td>
<td>Suction system calibration setup</td>
</tr>
<tr>
<td>5.11</td>
<td>Velocity profiles measured at Station ‘A’</td>
</tr>
<tr>
<td>5.12</td>
<td>Calibration measurements outside and in the wind tunnel</td>
</tr>
<tr>
<td>5.13</td>
<td>Suction hole arrangement in Loughborough University wind tunnel</td>
</tr>
<tr>
<td>5.14</td>
<td>Sketch of suction system installation</td>
</tr>
<tr>
<td>5.15</td>
<td>Suction system installed in the wind tunnel</td>
</tr>
<tr>
<td>6.1</td>
<td>Overall view of junction flow features, Re=0.89x10^6</td>
</tr>
<tr>
<td>6.2</td>
<td>Wing suction surface, Re=0.89x10^6</td>
</tr>
<tr>
<td>6.3</td>
<td>Wing suction surface (zoomed-in views), Re=0.89x10^6</td>
</tr>
<tr>
<td>6.4</td>
<td>Wing pressure surface, Re=0.89x10^6</td>
</tr>
</tbody>
</table>
Fig. 6.5: Wing pressure surface (zoomed-in views), Re=0.89 x 10^6 .......... 122
Fig. 6.6: Zoomed-in view of saddle and nodal point, Re=0.89 x 10^6 .......... 122
Fig. 6.7: Overall view of junction flow features, Re=0.44 x 10^6 ................. 123
Fig. 6.8: Wing suction surface (zoomed-in views), Re=0.44 x 10^6 .......... 124
Fig. 6.9: Measured Cp distribution, Re=0.44 x 10^6 ................................ 125
Fig. 6.10: Predicted flow patterns on floor, Re=0.44 x 10^6 .................. 125
Fig. 6.11: Predicted flow patterns on wing surface, Re=0.44 x 10^6 .......... 126
Fig. 6.12: Separation line location on floor, Re=0.44 x 10^6 .................. 126
Fig. 6.13: Wing suction surface, Re=0.44 x 10^6 .................................. 127
Fig. 6.14: Wing pressure surface, Re=0.44 x 10^6 ................................ 127
Fig. 6.15: Cp distribution at 5 mm, Re=0.44 x 10^6 ..................... 128
Fig. 6.16: Cp distribution at 75 mm, Re=0.44 x 10^6 .................. 128
Fig. 6.17: Cp distribution at 5 mm, 22.0^0 incidence, Re=0.22 x 10^6 .......... 129
Fig. 6.18: Predicted flow patterns on floor with suction, Re=0.44 x 10^6 ...... 130
Fig. 6.19: Predicted flow patterns on wing surface with suction, Re=0.44 x 10^6 ................................................................. 130
Fig. 6.20: Predicted Cp distribution at 5 mm, Re=0.44 x 10^6 .................. 131
Fig. 6.21: Predicted Cp distribution at 75 mm, Re=0.44 x 10^6 ................ 131
Fig. 6.22: Overall view of junction with suction, Re=0.44 x 10^6 ............. 132
Fig. 6.23: Flow around suction hole, Re=0.44 x 10^6 .......................... 133
Fig. 6.24: Wing suction surface with suction, Re=0.44 x 10^6 ............. 133
Fig. 6.25: Wing suction surface with suction (zoomed-in views), Re=0.44 x 10^6 ................................................................. 134
Fig. 6.26: Wing suction surface (zoomed-in views), Re=0.44 x 10^6 .......... 134
Fig. 6.27: Wing pressure surface with suction, Re=0.44 x 10^6 ............. 135
Fig. 6.28: Wing pressure surface (zoomed-in views) with suction, Re=0.44 x 10^6 ................................................................. 135
Fig. 6.29: Wing pressure surface (zoomed-in views), no-suction case, Re=0.44 x 10^6 ................................................................. 136
Fig. 6.30: Wing suction surface, Re=0.44 x 10^6 .................................. 136
Fig. 6.31: Cp distribution at 75 mm, Re=0.44 x 10^6 .......................... 137
Fig. 6.32: Cp distribution at 5 mm, Re=0.44 x 10^6 .......................... 137
Fig. 6.33: Cp distribution at 5 mm (22.0^0 incidence), Re=0.44 x 10^6 ........ 138
Fig. 6.34: Cp distribution at 5 mm, Hole relocated, Re=0.44 x 10^6 .......... 139
Fig. 6.35: Comparison of the two suction hole locations, Re=0.44 x 10^6 ...... 139
Fig. 6.36: Cp distribution at 5 mm with suction at 3.3Cq, Re=0.44 x 10^6 .... 140
Fig. 6.37: Cp distribution at 75 mm with suction at 3.3Cq, Re=0.44 x 10^6 ...... 141
List of Figures

Fig. 6.38: Predicted locations of separation lines, Re=0.44 x 10^6 .................. 141
Fig. 6.39: Predicted saddle point location with suction, Re=0.44 x 10^6 .......... 142
List of Tables

Table 1.1: Effect of cross section shape on separation distance\textsuperscript{15} ....................... 18
Table 1.2: Effect of Bluntness factor on separation distance................................. 21
Table 2.1: Variation of predicted aerodynamic coefficients with far-field distance
........................................................................................................................................... 37
Table 2.2: Variation of predicted coefficients with total number of grid points .. 38
Table 2.3: Predicted variation of aerodynamic coefficients with grid points...... 40
Table 2.4: Predicted coefficients with varying first cell height.............................. 41
Table 3.1: Force and moment coefficient predictions............................................. 76
Table 3.2: Drag breakdown predictions for half wing and ideal half wing ....... 76
Table 4.1: Effect of suction on half wing aerodynamic coefficients, $10.0^\circ$
incidence......................................................................................................................... 93
Table 4.2: Effect of grid modification on aerodynamic coefficients............... 95
Nomenclature

Symbols

A  Wing axial force, Area
BF Bluntness factor
c Chord
Cd Aerofoil drag coefficient
CD Wing drag coefficient
Cl Aerofoil lift coefficient
C\textsubscript{\textalpha} Aerofoil lift curve slope
CL Wing lift coefficient
CL\textsubscript{\textalpha} Wing lift curve slope
Cm Aerofoil pitching moment coefficient
CM Wing pitching moment coefficient
CP Surface static pressure coefficient
Cq Suction coefficient
d Splitter plate length upstream of wing leading edge, diameter
I Turbulence intensity
k-\varepsilon k-epsilon
k-\omega k-omega
l Turbulent length scale
L Characteristics length
m Mass flow
Re Chord Reynolds number
r Radius
S\textsubscript{(t)\textsubscript{max}} Distance along aerofoil surface from leading edge to \textsubscript{t\textsubscript{max}}
t Aerofoil thickness
u Local flow speed in x-direction
V Velocity
x Axial distance, Flow Direction in test section
y Direction along the model span
z Direction normal to the model
Greek Symbols

\(\alpha\) Incidence angle
\(\delta\) Boundary layer thickness
\(\delta^*\) Boundary layer displacement thickness
\(\nu_t\) Turbulent viscosity
\(\mu\) Coefficient of viscosity
\(\rho\) Density

Subscripts

\(\infty\) Free-stream value
a.c. Aerodynamic centre
avg Average
bl Boundary layer
c Chord
le Leading edge
max Maximum
s Separation

Abbreviations

2D Two-dimensional
3D Three-dimensional
CFD Computational fluid dynamics
LDV Laser Doppler Velocimetry
LES Large Eddy Simulations
NACA National Advisory Committee for Aeronautics
NASA National Aeronautics and Space Administration
RANS Reynolds-averaged Navier-Stokes
1 Introduction

Wind tunnels are commonly used both for aeronautical and non-aeronautical research, even in this age of greatly developed Computational Fluid Dynamics (CFD) tools. The significance of wind tunnels has not been overshadowed with the advent of CFD, rather the development of numerical capabilities during the last five decades has benefited greatly from experimental data. All new computational models require validation by experiment.

The focus of this thesis is on the use of a technique known as ‘half model testing’, which offers a number of advantages over testing a complete model. Some of these benefits include:

I. An increased Reynolds number is achieved\(^1,2,3\). The scale of the increase depends on the geometry of the wind tunnel working section and the orientation of the half model. Mounting a half model in the same orientation as a full model will increase the working Reynolds number by a factor of two.

II. Improved data quality due to:

a. The absence of stings or struts required to mount full span models\(^1,2\). Their absence removes the need to determine their contribution to measured forces and moments. Possible interference effects between the struts (or sting) are also removed.

b. Half models are larger and produce greater forces, and moments for a given free-stream wind speed. This increases the measurement accuracy.

c. In larger models the manufacturing tolerances are less significant than in smaller models which are thus machined with greater accuracy. This also facilitates more accurate placement of various model components for example flaps, slats etc.
III. Manufacturing costs reduce, particularly for complex models\textsuperscript{1,2,3}. In complex models, involving control surfaces and propulsion systems, just one set would be required for testing.

However, these gains do not come at zero cost; well established discrepancies in a half model as compared to full model testing are:

I. It is not possible to test half models for different yaw angles as this would require deflecting the model with respect to the reflection plane, which is one of the wind tunnel walls. Hence testing is limited to incidence variations only\textsuperscript{3}.

II. Larger forces and moments generated in half models may limit the operating incidence range to avoid overloading the balance.

III. Half models exhibit extra aerodynamic losses due to flow separation in the junction\textsuperscript{1,4}:
   a. Half models exhibit lower lift coefficient values. This loss of lift increases with incidence, resulting in a reduced lift curve slope for half models.
   b. There is an increment in drag as the junction is approached.

This thesis investigates whether the scale of the aerodynamic losses due to junction flow can be reduced or avoided. A review of half model testing is provided in the following section.

1.1 Half Model Wind Tunnel Testing

Almost all flying vehicles have a plane of symmetry along the fuselage. This is exploited by the half model technique as only half of the model is tested, with the wind tunnel side wall or floor acting as a plane of symmetry. The technique works on the principle of the ‘method of images’\textsuperscript{5}. The principle may be illustrated by considering two aerofoils in inviscid flow and in close proximity to each other so that their flows are interacting. The aerofoils are placed in a fashion such that they are each other’s mirror image with respect to the symmetric plane AA’, as shown in Fig. 1.1.
Since the aerofoils are the same, the flow field around them will also be the same. Hence, the flow around aerofoil B can be analyzed by replacing aerofoil B' with a solid, shear free boundary at AA'. Replacing aerofoil B' with such a boundary at AA' is analogous to splitting a model through its plane of symmetry and fixing it on a tunnel floor or wall which then acts as the symmetric plane.

1.2 Review of Half Model Wind Tunnel Testing from Literature

However in real life, unlike in the above illustration, a boundary layer forms along the tunnel wall and hence the reflection plane is not perfect. In half model wind tunnel testing, interaction of the floor boundary layer with the model affects its aerodynamic characteristics\(^2^3\). The current study will focus on the performance of wing-only half models as opposed to half aircraft models which incorporate a fuselage. Use of the wing alone model allows investigation into the effect of junction flows without introducing the complexities of fuselage effects. Consequences of the interaction of the floor boundary layer and the wing are described in the following sections.

1.2.1 Barberis et al. Experiment\(^6\)

Barberis et al.\(^6\) conducted Laser Doppler Velocimetry (LDV) measurements to study the flow patterns in the junction region symmetry plane just upstream of a wing leading edge (Fig. 1.2). Tests were conducted using a wing with a symmetric cross section, at 0\(^0\) incidence and at a chord Reynolds number (Re) of 3.9 x 10\(^6\). Experiments were carried out in the ONERA F2 subsonic wind tunnel. The wind tunnel had height and width of 1800 mm and 1400 mm,
respectively. The wing was mounted on a false floor, spanning the wind tunnel length and width, at a height of 530 mm from the tunnel floor. The height of the turbulent boundary layer at the wing leading edge was not specified but may be approximated as 45 mm using flat plate theory for the given conditions. The wing had a 1090 mm chord with a span of 940 mm and maximum thickness of 360 mm.

![Fig. 1.2: Top view of Barberis et al. arrangement](image)

The measured flow patterns are shown in Fig. 1.3 where the oncoming flow, after interaction with the wing spirals into a vortex core, which is the origin of a complex 3D flow structure. This is the so-called ‘horseshoe vortex’, which is described in more detail later in this chapter. The vortex core had a height of approximately 13 mm and was located about 86 mm from the wing surface. The presence of this vortex as it wraps around the wing was not explored in this study but this is clearly an important structure in wing lift loss and drag.

![Fig. 1.3: Vortex core developed in symmetry plane, Barberis et al. experiment](image)
1.2.2 Philips et al. Experiment

Philips et al.\textsuperscript{7} conducted a similar experiment at Pennsylvania State University in a subsonic wind tunnel. The test section had a height and width of 310 mm and 970 mm, respectively. Measurements were done for a half wing with a symmetric cross section (Fig. 1.4) at a $\text{Re} = 0.42 \times 10^6$. The wing had a maximum thickness of 140 mm, a chord of 900 mm and a span of 203 mm. The tunnel wall boundary layer was turbulent with a thickness of 30 mm at the wing leading edge.

![Fig. 1.4: Top view of Philips et al. arrangement\textsuperscript{7}](image)

Measurements were done using a five hole probe. Intrusive effects of the probe may be present but were assumed to be negligible. The measurements were made in a plane perpendicular to the flow located 215 mm (24 % of chord) downstream of the leading edge (Fig. 1.4); velocity vectors in the plane are shown in Fig. 1.5. Velocity vectors in the junction region seem to indicate four vortices as shown by the arrows in Fig. 1.5. Given its location and sense of rotation, the vortex labelled ‘V1’ is identified as the junction horseshoe vortex mentioned above. The vortex is located in the corner between the wing and floor and has the right direction for the horseshoe ‘leg’ of the leading edge vortex earlier shown in Fig. 1.3. The origin of the other vortices is not known but it is pertinent to mention that the small size of the test section may contribute to the formation of these vortices; for the plane shown in Fig. 1.5, the distance between the wing surface and tunnel wall was 423 mm. Also, it was realised that the junction flow may not be independent of the wing span for such a small aspect ratio (0.23) but no further information was provided in this respect.
1.2.3 Mendelsohn & Polhamus Experiment

Mendelsohn & Polhamus conducted experiments on a 2D wing with NACA 651-012 cross section. The experiments were carried out in the NASA Langley Stability Tunnel having a test section height and width of 1829 mm and 762 mm, respectively. The wing spanned the tunnel height and had a chord of 610 mm. The height of the boundary layer 150 mm upstream of the wing leading edge was 55 mm. The pressure measurements along the span at $3.0^0$ and $12.0^0$ incidence at a $Re = 2.32 \times 10^6$ were carried out to study the wall boundary layer effect (No flow visualisation is available). Pressure distributions at $3.0^0$ incidence are shown in Fig. 1.6 with a zoomed-in view of the leading edge suction peak shown in Fig. 1.7. Moving away from the floor, leading edge pressure coefficients increase upto a height of 122 mm but at a height of 914 mm (significantly away from the junction) the suction peak drops. This is probably due to a leading edge laminar separation. In the region of the wing affected by the junction flow this separation is prevented due to the increased turbulent mixing making it less prone to separation. Downstream of 2.5 % of chord location, the pressure values increase with height from the tunnel floor.
Pressure distributions at 12.0° incidence are shown in Fig. 1.8 with a zoomed-in view of the leading edge suction peak shown in Fig. 1.9. At this incidence the suction peak began to drop at 30 mm in contrast to the height of 914 mm at 3.0° incidence, indicating leading edge separation has extended lower down on the wing due to increased adverse pressure gradient at high incidence. At this incidence, the leading edge laminar separation region at 914 mm is significantly greater than that seen at 3.0° incidence. Downstream of the leading edge laminar separation, the pressure coefficients at 914 mm are greater in magnitude than at the lower heights. Comparison with results at 3.0° incidence indicates a smaller effect of the junction flow at lower incidence.
1.2.4 Bernstein & Hamid Experiment

A similar experiment to Mendelsohn & Polhamus was conducted by Bernstein & Hamid at a Re = 0.86 x 10⁶. Their experiments took place in a 1.24 m wide by 1.0 m high test section. The wing was mounted on a 100 mm high false floor with width and length equal to the test section. The boundary layer on the floor was tripped and had a thickness of 30 mm at the wing location in an empty tunnel. The wing had a chord of 532 mm in the streamwise direction and spanned from the false floor up to close to the ceiling (gap not specified). The wing had a NACA 0015 cross section and a 20° swept back angle. Pressure measurements for the swept wing along the span for 3.0° incidence (Fig. 1.10)
and $12.0^0$ incidence (Fig. 1.11) are available up to a quarter span height. The reduction in the suction peak observed by Mendelsohn & Polhamus$^8$ was not seen by Bernstein & Hamid$^4$. Beside this, the trends for the swept wing were similar to the trends observed earlier (Fig. 1.6 and Fig. 1.8) for the non-swept wing, i.e. increase in suction peak with increasing distance from tunnel floor. Therefore, sectional properties of the swept wing should be a reflection of the non-swept wing characteristics.

At $3.0^0$ incidence the pressure distribution at 125 mm is the same as measured at 250 mm, indicating that the junction effects are limited to beneath the former height. Loss of lift due to downward shift of the suction surface pressures is mitigated by the shift of the pressure distribution on the pressure surface.

![Fig. 1.10: Cp distribution on NACA 0015 half wing, 3.0^0 incidence$^4$](image)

At $12.0^0$ incidence the suction peak increases upto 250 mm showing that the junction effects were observed at heights significantly greater than the onset flow, floor boundary layer height. On the pressure surface, the distribution is relatively insensitive to span and hence the effect on the lift coefficient will be greater than that observed at $3.0^0$ incidence.
These pressure coefficients were also converted into sectional lift and drag coefficients and are shown in Fig. 1.12. A decrease in the wing sectional lift coefficient and an increase in the pressure drag coefficient were seen as the junction region was approached. The junction effects were observed at heights significantly greater than the onset floor boundary layer thickness and increased with increasing incidence. No marked reduction in lift coefficient was seen up to 6.0° incidence. At 12.0° incidence the percentage decrease in the measured lift coefficient for a section in the junction, compared against values for an outboard section free from junction flow effects was calculated as 6.7 %; the increasing reduction in lift with incidence reduces the half wing lift curve slope ($C_{L\alpha}$). Compared to the lift coefficient, in percentage terms the pressure drag was more severely affected as the junction was approached. At 12.0°, the percentage increase in sectional drag was 129 %.
1.2.5 Wood & Westphal Experiment

Wood & Westphal tested a wing with a NACA 0012 profile in the Boundary Layer Wind Tunnel at NASA Ames Research Centre. The wind tunnel had a test section with the height of 200 mm and width of 800 mm. The wing with a chord of 100 mm spanned the wind tunnel height and was tested at a Re = 0.18 x 10^6. The floor boundary layer was tripped and had a thickness of 22 mm in the region of the wing. Pressure measurements on the wing surface were made, for 0.0°, 5.0° and 10.0° incidence, at various stations along the span ranging from a height of about half a millimetre (0.25 % of span) to a height of 95 mm (48 % of span).
span). Integration of the pressures along the chord showed that for the 0.5 mm height, the sectional lift coefficient decreased by 16% from a value measured at 95 mm height. At 10.0°, a peak Cp increase in magnitude of about 0.7 was measured at a height of 95 mm compared to values measured at 0.5 mm from floor. Cp values measured at a height of 65 mm (33 % of span) were the same as measured at 95 mm for 10° incidence. Furthermore, pressure measurements indicated no separation on the wing for heights as low as 2.5 mm (1.25 % of span).

1.2.6 Applin Experiment

Applin\textsuperscript{11} conducted a similar experiment to Wood & Westphal\textsuperscript{9} employing an untwisted rectangular half wing with a NACA 0012 cross section in the NASA Langley 14 x 22 ft subsonic wind tunnel\textsuperscript{11}. The height of the boundary layer on the tunnel floor, indicated from the wind tunnel user guide\textsuperscript{24}, was about 230 mm. The span of the wing was 2.95 m and chord 1 m resulting in an aspect ratio of 2.95. Transition strips on the upper and lower surfaces, extending over the entire span, were fixed at about 51 mm downstream of the wing leading edge. Surface pressure measurements were made at various stations ranging from 3.3 % to 47 % of span. The height of the first measuring station at 97.4 mm, (3.3% of span) was within the onset flow floor boundary layer. Tests were carried out for chord Reynolds numbers of 2.4 \times 10^6, 3.3 \times 10^6 and 4.71 \times 10^6. Results are presented here for Re = 2.4 \times 10^6 since this is closest to the typical Reynolds number achieved in the Loughborough University wind tunnel. Pressure coefficient results presented earlier, for NACA 651-012 and NACA 0015, did not show any definite trends significantly downstream of the leading edge. Hence for Applin’s experiment, presentation of the pressure distribution is limited to approximately 13 % chord.

At 4° incidence (Fig. 1.13), pressure coefficient values were relatively unaffected as the junction was approached with no conclusive trends seen in pressure variation. Loss of total pressure in the floor boundary layer prevents the local pressure coefficient achieving the stagnation. At 10.0° incidence, the reduction in peak pressure coefficient is under 5 % (Fig. 1.14) with a ‘Cp’ drop of 0.15 as the junction was approached, significantly smaller than the reduction of 0.7 measured by Wood & Westphal\textsuperscript{9}. Different measurement heights with respect to an onset flow boundary layer thickness in the two experiments are a plausible explanation for the difference. Wood & Westphal's\textsuperscript{9} measurements closest to the floor were done at a height equal to 2.3 % of the boundary layer.
thickness whereas the closest station in Applin's\textsuperscript{11} experiment was at 42 %. In Applin's experiment, at a height of 1391 mm, the pressure plateau is possibly due to leading edge laminar separation, and this may also contribute to the different behaviour noted.

![Figure 1.13: Cp distribution on NACA 0012 half wing, 4.0° incidence\textsuperscript{11}](image1)

At 18.0° incidence (Fig. 1.15), a marked reduction was seen in the magnitude of the junction region pressure coefficient values. At 18.0° incidence moving away from the junction, a distinct pattern of increasing pressure peak is followed up to midspan. It is observed that the junction effects manifest themselves at a height of 978 mm, i.e. more than 4 times the onset flow boundary layer thickness. Beyond 10.0°, at the lowest height of 97 mm, the suction peaks dropped with
increasing incidence (relative to the Cp peak further outboard) indicating an increasing loss of lift with incidence. The decrease in the suction pressure and flatter pressure profiles downstream of the suction peak at 97 mm, indicate junction region separation and loss of lift.

![Graph showing Cp distribution on NACA 0012 half wing, 18.0° incidence](image)

**Fig. 1.15: Cp distribution on NACA 0012 half wing, 18.0° incidence**

Since most of the experiments reviewed above have noted the largest effects on pressures in the regions likely to have been influenced by the horseshoe vortex feature mentioned earlier, this is discussed in more detail in the following section.

### 1.2.7 Formation of the Junction Horseshoe Vortex

Almost all of the work done in the area of horseshoe vortex formation is for symmetrical cross sections and hence this review has to be limited to such geometries. The origin and development of the vortex can be understood from the descriptions available in Eckerle & Langston \(^\text{12}\), Johnston \(^\text{13}\) and Kang et al \(^\text{14}\). The formation of a horseshoe vortex in a wing-floor junction originates in the wing leading edge region. The oncoming low momentum boundary layer flow separates as a result of high adverse pressure gradient created by the presence of the wing leading edge. A typical static pressure profile \(^\text{12,14}\) in the impingement plane is shown in Fig. 1.16. In this plane the oncoming flow (streamline ‘C’) separates at the ‘saddle point’. This point is a classic feature of a two dimensional separation; at this point shear stress drops to zero and flow downstream of this point is travelling in the opposite direction to the mainstream flow. In this plane downstream of the saddle point is filled by flow deflected by
the wing leading edge (streamline ‘A’). Above the floor the oncoming flow (streamline ‘D’) travels further downstream than streamline ‘C’ before it interacts with the deflected flow (streamline ‘B’). Interaction of oncoming and deflected flow results in the formation of a vortex core located some distance downstream of the saddle point. This is the origin of the 3D structure referred to as a junction horseshoe vortex. The rotating flow about the ‘eye’ of the vortex is bent downstream and around the wing where it moves off the impingement plane. It is for this reason that many studies on the junction horseshoe vortex involve investigating the saddle point and the separation line.

![Sketch of flow patterns in impingement plane](image)

**Fig. 1.16: Sketch of flow patterns in impingement plane**

Flow separation at the saddle point and the movement of the vortex into a horseshoe form may be illustrated via the skin friction lines on the tunnel floor (Fig. 1.17-showing only symmetric half of the flow patterns). Flow towards the saddle point, both from upstream and downstream, is diverted off the symmetry plane and moves around the wing along the ‘separation line’. The separation line forms a boundary on the floor between the oncoming flow and flow deflected by the wing. Unlike two dimensional separation, at this line the shear stress does not necessarily drop to zero since the flow is now three dimensional and separation may occur without all the three components of velocity going to zero. The region close to the separation line, on the wing side, is occupied by flow deflected by the leading edge of the wing as shown in Fig. 1.17. This affects the flow patterns in the neighbouring planes to the symmetry plane (see Fig. 1.17). In these planes the deflected flow on the floor separates before reaching the separation line (Fig. 1.18) as the region is already occupied by the flow deflected from the leading edge. The other effect of this is that now the flow approaches the vortex at a steeper angle. Hence, by this means the vortex propagates downstream around the wing root.
Formation of the horseshoe vortex is influenced by the wing cross section shape. This includes leading edge radius, maximum thickness and location of maximum thickness. The formation of the horseshoe vortex is also affected by the aspect ratio for wings where the size of the boundary layer is comparable to the wing span.

Olcmen & Simpson\textsuperscript{15} investigated the effect of the cross section shape on the floor flow patterns. The experiments were done in the Boundary Layer Tunnel at Polytechnic Institute and State University of New York. All the wings used were of different sizes and tested at a tunnel speed of 32.5 m/s and hence the resulting chord Reynolds number ranged from $0.69 \times 10^6$ to $1.4 \times 10^6$. Thickness to chord ratio of the wing wings ranged from 12 % to 43 % of chord and the leading edge radius ranged from 0.9 % to 21.3 % of chord. Tunnel floor flow visualisation pictures (Fig. 1.19) clearly show the existence of a saddle point and the separation line for all wings.
Fig. 1.19: Junction flow patterns, Olcmen & Simpson Experiment\textsuperscript{15}
Before discussing the flow visualisation in greater detail it is useful to consider Table 1.1. The table summarises non-dimensional separation distance between wing leading edge and saddle point \((x_s/c)\), the non-dimensional wing thickness \((t/c)\) and leading edge radius \((r_le/c)\). The trend seen is an increase in separation distance with increasing leading edge radius. In all cases, other than Sandia 1850, it is seen that the separation distance also increases with maximum thickness. The separation distance for the Sandia 1850 wing not following the trends of the other cross sections indicates that other parameters need to be investigated. This is discussed in more detail later.

<table>
<thead>
<tr>
<th>Section</th>
<th>Re</th>
<th>t/c</th>
<th>(r_{le}/c)</th>
<th>(x_s/c)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sandia 1850</td>
<td>1.40 x 10^6</td>
<td>0.18</td>
<td>0.009</td>
<td>0.028</td>
</tr>
<tr>
<td>NACA 0012</td>
<td>0.93 x 10^6</td>
<td>0.12</td>
<td>0.016</td>
<td>0.041</td>
</tr>
<tr>
<td>NACA 0015</td>
<td>1.40 x 10^6</td>
<td>0.15</td>
<td>0.025</td>
<td>0.045</td>
</tr>
<tr>
<td>Rood</td>
<td>0.70 x 10^6</td>
<td>0.24</td>
<td>0.078</td>
<td>0.106</td>
</tr>
<tr>
<td>Tear drop</td>
<td>0.69 x 10^6</td>
<td>0.43</td>
<td>0.213</td>
<td>0.220</td>
</tr>
</tbody>
</table>

Table 1.1: Effect of cross section shape on separation distance\(^{15}\)

For the Sandia 1850 wing the upstream separation and the separation line were quite distinct (Fig. 1.19a). For the NACA 0012 a very weak upstream separation was observed and there was no well-marked separation line around the wing (Fig. 1.19b). As the wing thickness increased to the NACA 0015 wing, the upstream separation and the separation line becomes more distinct (Fig. 1.19c). For the Rood aerofoil (Fig. 1.19d) and the tear drop shape (Fig. 1.19e) very distinct upstream separation and a marked separation line were apparent.

A similar experiment was conducted by Mehta\(^{16}\) who observed that the vortex size and strength increased with the half wing nose bluntness. Experiments were conducted in a test section with width and height of 762 mm x 127 mm, respectively. The floor boundary layer was turbulent and the estimated thickness, based on flat plate theory and given conditions, was 25 mm. The wings, spanning the tunnel height, had 325 mm chord; a 150 mm long nose was followed by a 175 mm long constant thickness section. The nose cross sections of the three shapes, here termed as M-Shape1, M-Shape2 and M-Shape3 are shown in Fig. 1.20.
The experiments were conducted at $0^\circ$ incidence and at a $Re = 0.58 \times 10^6$ with the boundary layer on the wing tripped at distances of 25 mm from the leading edge. Velocity contours at an axial location of $x = 175$ mm from the leading edge are shown in Fig. 1.21. No marked vortex or rotation was seen for ‘M-Shape3’ (Fig. 1.21a) indicating that the less blunt leading edge prevents upstream separation and hence avoids the formation of a horseshoe vortex. As the leading edge shape changed to ‘M-Shape1’ a distinct vortex appeared with an approximate centre at a height of 9 mm from the floor and 15 mm from the wing (Fig. 1.21b). As the nose bluntness was further increased the vortex became stronger and bigger with the centre moving away from the floor and wing surface (Fig. 1.21c).
The above experiments clearly demonstrated a strong dependence of junction horseshoe vortex strength and size on the wing cross section shape. Fleming et al.\textsuperscript{17} devised a correlation, given by Eq.(1.1), which gave an indication of the vortex strength.

\[
BF = \frac{1}{2} \frac{(r)_{le}}{X_{(t)\max}} \left[ \frac{(t)_{\max}}{S_{(t)\max}} + \frac{S_{(r)\max}}{X_{(r)\max}} \right] 
\]

where, ‘BF’ is the bluntness factor, $(r)_{le}$ is the leading edge radius, $X_{(t)\max}$ is the distance from the leading edge to the maximum thickness, $(t)_{\max}$ is the maximum thickness and $S_{(t)\max}$ is the distance from the leading edge to the maximum thickness along the aerofoil surface. Bluntness factor is a geometric...
property and hence independent of the flow conditions. It was observed that the larger bluntness factor the stronger the vortex\textsuperscript{18}. Bluntness factors for all the above discussed shapes are summarised in Table 1.2. Investigating these results show that the factor does give a reasonable indication of the vortex strength. All the wings tested by Olcmen & Simpson\textsuperscript{15} and Mehta\textsuperscript{16} showed increasing separation with increasing bluntness factor.

<table>
<thead>
<tr>
<th>Section</th>
<th>BF</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sandia 1850</td>
<td>0.013\textsuperscript{18}</td>
</tr>
<tr>
<td>NACA 0012</td>
<td>0.029\textsuperscript{18}</td>
</tr>
<tr>
<td>NACA 0015</td>
<td>0.045\textsuperscript{18}</td>
</tr>
<tr>
<td>Rood aerofoil</td>
<td>0.520\textsuperscript{18}</td>
</tr>
<tr>
<td>Tear drop</td>
<td>1.070\textsuperscript{18}</td>
</tr>
<tr>
<td>M-Shape3</td>
<td>0.0014</td>
</tr>
<tr>
<td>M-Shape1</td>
<td>0.0110</td>
</tr>
<tr>
<td>M-Shape2</td>
<td>0.2279</td>
</tr>
</tbody>
</table>

Table 1.2: Effect of Bluntness factor on separation distance

These measurements demonstrate the effectiveness of the parameter in predicting the separation distance. However, it should be noted that in all cases the wings were symmetrical and non-lifting, i.e. tested at $0^\circ$ incidence.

1.3 Techniques Used for Half Wing Testing

It is believed that half wing results may be improved either by eliminating or reducing the approach flow boundary layer on the tunnel floor or by eliminating the effects of the boundary layer, i.e. the junction horseshoe vortex formation. Some of the attempts made in this regard are described below.

1.3.1 Attempts to reduce the onset Boundary Layer Thickness

a) Splitter Plate

Use of a false floor, called a splitter plate, is a commonly recognised approach to improve half wing results\textsuperscript{19,20,21}. The net effect is a reduction in boundary layer thickness at the wing leading edge (Fig. 1.22). The splitter plate puts the wing out of the floor boundary layer but a new boundary layer is started at the leading edge of the splitter plate but this has significantly less thickness than the floor boundary layer at the wing leading edge. As an example of the effectiveness of splitter plates\textsuperscript{20}, the boundary layer thickness measured on the
floor and with a splitter plate at Mach 0.6 in the NASA Langley Transonic Dynamics wind tunnel was 304.8 mm and 31.75 mm, respectively.

Fig. 1.22: Reduction of boundary layer thickness with splitter plate

Bippes\textsuperscript{21} conducted experiments to investigate the effectiveness of a splitter plate in half wing testing in a low speed wind tunnel at DFVLR, Germany. The wind tunnel had a test section of 3 m x 3 m and an onset flow floor boundary layer thickness of 30 mm\textsuperscript{22}. The wing (NACA 4415 profile) had a chord of 0.6 m and a span of 1.86 m\textsuperscript{22}, resulting in an aspect ratio of 3.1. The wing was installed with an offset from the tunnel centre line; effects of this, if any, are not clear. The experiments were conducted at 21.0\degree incidence and at a Re = 2.1 \times 10^{6}; at these conditions the NACA 4415 profile is in the post-stall state\textsuperscript{23}. The incidence angle near stall was chosen to investigate the use of a splitter plate for cases with trailing edge flow separation, shown for a full span wing in Fig. 1.23a. For the half wing, interaction of the flow separation in the junction region and that on the wing surface resulted in flow patterns that were significantly different than those seen at the midspan of the full wing (Fig. 1.23b). The interaction resulted in reduction in the size of the separation region. The splitter plate was then employed to investigate if the half wing results could be brought closer to the full span results.
a): Full span wing  

b): Half wing

Fig. 1.23: Flow patterns on NACA 4415 wing, 21.0° incidence\textsuperscript{21}

The height of the splitter plate above the tunnel floor was set equal to the floor boundary layer thickness but the boundary layer thickness on the splitter plate was not measured. The wing was tested with varying splitter plate lengths upstream of the leading edge ranging from 1c to 3.2c. Using a splitter plate moved the floor/junction flow separation line (on the reflection plane) closer to the wing which moved even closer with decreasing length of the plate (Fig. 1.24). This separation line indicates the presence of the junction horseshoe vortex and the size of the vortex reduced as the separation lines moved closer to the wing. The vortex existed even for the shortest splitter plate tested as is indicated by the presence of a separation line on the splitter plate, although very close to the wing (Fig. 1.24b).

Fig. 1.24: Separation size for different splitter plate length, 21.0° incidence\textsuperscript{21}
Varying the splitter plate length modified the flow patterns on the wing suction surface as shown in Fig. 1.25. Large separation regions on the wing surface with the splitter plate were significantly different from the full span wing patterns (Fig. 1.23a).

![Flow patterns for half wing with splitter plate](image)

a): Splitter Plate Length = 3.2c  
b): Splitter Plate Length = 1c  

Fig. 1.25: Flow patterns for half wing with splitter plate, 21.0° incidence

Fig. 1.26 shows pressure measurements at a height of 71 mm (3.8 % of span) with and without a splitter plate. Only a marginal increase in suction peak was measured with the use of a splitter plate; the highest suction peak was achieved with the shortest splitter plate. The general trend was an increase in suction peak with decreasing splitter plate length upstream of the leading edge. Pressure measurements identified a pressure plateau starting at 50% chord without the splitter plate, the longer splitter plates brought the plateau upstream to 40% of the chord whereas the shortest splitter plate completely eliminated the pressure plateau. It was concluded in Bippes\textsuperscript{21} that for the test conditions studied the splitter plate did not prove very effective, although the longer splitter plate performed marginally better than the shortest splitter plate tested.
b) Distributed Suction in Junction Region

Bippes & Turk\textsuperscript{22} investigated the potential of distributed suction to improve half wing performance. The baseline experimental setup and test conditions were the same as described in Section 1.3.1. Only one suction rate was tested and no indication was given of the magnitude of suction rate. Suction was applied through a perforated disk on the tunnel floor in the wing root area (Fig. 1.27).

\textbf{Fig. 1.26:} Cp distribution at height of 71 mm (3.8 \% of span), 21.0\(^\circ\) incidence\textsuperscript{21}

\textbf{Fig. 1.27:} Sketch of Distributed Suction Setup, Bippes & Turk Experiment\textsuperscript{22}

Wing pressure plots, with and without suction, are only available at a height of 147 mm (7.9 \% of span). No effect of suction was observed on the pressure
distribution of the half wing. Possible reasons for this could be the large distance from the tunnel floor or that the suction used was not sufficient to cause a significant effect.

Fig. 1.28: Cp distribution at height of 147 mm (7.9 % of span), 21.0° incidence

c) **Boundary Layer Removal through Upstream Suction**

The idea of boundary layer suction is to replace the low momentum flow near the floor with more energetic flow from the free-stream (Fig. 1.29). Applin\(^{11}\) has investigated the effect of reducing boundary layer thickness using suction. Suction was applied through a slot spanning the tunnel width, located 2.5 m upstream of the wing leading edge. With the suction system in operation the floor boundary layer thickness in the empty wind tunnel at the wing location was reduced from 230 mm to 41 mm\(^{24}\), otherwise the setup was the same as described earlier. With the suction system in operation, the surface pressure profile measuring height closest to the floor, 97.4 mm (3.3 % of span), was outside of the onset boundary layer thickness.
At 14° incidence the only effect of suction was seen as elimination of a laminar separation bubble that originated at 1.2% of chord. Below this incidence, the suction effect was to cause a reduction in the pressure peak. At incidences greater than 14.0°, employing suction increased pressure coefficient values on the suction surface and hence improved the pressure coefficients from their no-suction values. Using suction, pressure trends with increasing incidence were as expected. At a height of 1391 mm (47% of span), the only effect of suction was seen at 10.0° and 14.0° in the form of a reduction in pressure peak (Fig. 1.31).
1.3.2 Attempts to Remove the Horseshoe Vortex

A number of attempts have been made to eliminate the horseshoe vortex\textsuperscript{6,7,25-28}. Among successful attempts, the use of localised wall suction, to eliminate the horseshoe vortex has the benefit that it gives the potential to vary suction and so adapt the technique for varying incidence and different half wing geometries. Philips et al.\textsuperscript{7} and Barberis et al.\textsuperscript{6} have both claimed success at removing the horseshoe vortex using this technique, and their work is briefly described here.

(a) Philips at al. Experiment with Localised Suction\textsuperscript{7}

The test set-up is the same as described in section 1.2.2. Suction was applied through a 150 mm wide x 190 mm long porous slot just upstream of the wing leading edge. The centre of the suction slot was aligned with the wing’s plane of symmetry (Fig. 1.32). The slot position was not given with reference to the wing or the no-suction saddle point location. No parametric studies regarding slot size or location were carried out. The measurement shown in the following figures were done at location of 0.24c rearwards of the wing’s leading edge.

![Fig. 1.32: Top view of Philips et al. setup with suction\textsuperscript{7}](image-url)
Velocity vectors in the junction region without suction are shown in Fig. 1.33a. It is pertinent to mention that these measurements were done with the porous plate in position but with no suction flow. It was observed by Philips et al.\textsuperscript{30} that in this no-suction case the presence of the porous surface increased the strength of the vortex. Suction rates ranging from 1.2Cq to 3.5Cq were tested; where ‘Cq’ is the suction coefficient and is defined as the ratio of mass flow in the boundary layer to the mass flow removed by suction. Here it is assumed that the boundary layer mass flow was calculated over the suction slot dimension in the cross flow direction in a plane located at hole centre. At low suction rates the size and strength of the vortex reduced but it was not until the suction rate was increased to 1.9Cq that the vortex was completely eliminated. This was observed by modification of the original vortex rotation (seen as ‘V1’ in Fig. 1.33a), in the measurement plane, to a resultant as shown by an arrow in Fig. 1.33b. Direction of the resultant, with suction, does not show the sense of rotation as seen for the no-suction case. Suction reduced the observed vortex strength in the measurement plane with increasing suction rate upto 1.9Cq; increasing suction beyond that rate started to increase the reduced vortex strength.

![Image](image_url)

**Fig. 1.33:** Effect of suction on junction velocity vectors, Philips et al. Experiment\textsuperscript{7}
(b) Barberis et al. Experiment with Localised Suction

Barberis et al.\textsuperscript{6} conducted an investigation into localised suction employing the same set-up as described in section 1.2.1. Suction was applied through a slot 100 mm wide x 80 mm long placed at two different locations. The first slot (Slot1) location coincided with the observed saddle point location for the no-suction case whereas the second position (Slot2) was between the wing and the saddle point.

The effects of suction were studied at $0^\circ$ incidence, using LDV measurements in the symmetry plane upstream of the leading edge, for a range of suction rates between $0.09C_q$ to $1.0C_q$. Suction had a marked effect on the separation in the wing plane of symmetry (Fig. 1.35).
Applying suction, through ‘Slot1’ at 0.09Cq, influenced the size and the location of the vortex, moving it closer to the floor and the wing as it reduced in size (Fig. 1.35b). As the suction rate was increased to 1.0Cq, the maximum suction applied, these effects were magnified, since the separation in the symmetry plane was significantly reduced although not completely eliminated. Applying a lower suction of 0.09 through ‘Slot2’ was significantly more effective than the higher suction through ‘Slot1’ (Fig. 1.35d). These results suggested a preferred position for the suction slot between the wing and the separation point. Furthermore, Barberis et al.\(^6\) concluded that further the slot was from the wing’s leading edge, the higher the suction rate needed.

**1.3.3 Discussion of Suction Experiments of Philips et al.\(^7\) and Barberis et al.\(^6\)**

The experiments of Philips et al.\(^7\) and Barberis et al.\(^6\) had a few significant differences (in experimental set-up) and hence resulted in different conclusions regarding the amount of suction required to eliminate the horseshoe vortex, (Philips et al.\(^7\)-1.9Cq, in contrast Barberis et al.\(^6\)-0.09Cq). In terms of Reynolds number the two experiments differ by an order of magnitude and its effect, if any, on the suction rate are not known although it seems unlikely to be the reason for such different conclusions. More convincing reasons are perhaps:
1) In Philips et al.\textsuperscript{7} experiment the location of the suction slot and the saddle point were not known, it may be that the slot was located significantly upstream of the saddle point and hence needed a higher suction rate. This is in line with the Barberis et al.\textsuperscript{6} observations. Barberis et al.\textsuperscript{6} studied the effect of suction by doing measurements only in the symmetry plane upstream of the leading edge whereas the measurements of Philips et al.\textsuperscript{7} were in a plane, perpendicular to the mainstream flow, located at $x = 0.24c$ downstream of the leading edge. Thus, one experiment examined the primary vortex core itself and the other the bent-around 'leg' of the horseshoe vortex, i.e. different parts of the horseshoe vortex were studied.

2) In Philips et al.\textsuperscript{7} experiment, measurements without suction were only available with the porous suction plate in place. Further information available from Ref. 30 highlighted that the clean flow junction vortex was strengthened due to the presence of the porous suction plate compared to the vortex seen with a solid tunnel floor. This increased strength was attributed to the increased boundary layer thickness caused by manufacturing imperfections. This could be another reason that Philips et al.\textsuperscript{7} had to use higher suction rates to eliminate the vortex.

3) It is not straightforward to capture a three dimensional flow picture when measurements are limited to a plane, although it was seen that suction successfully eliminated the vortex. Limited insight is available, from the Philips et al.\textsuperscript{7} experiment, on how the three dimensional flow field is modified by applying suction. No attempts were made to vary the slot shape, geometry or Reynolds number. Finally, no force or pressure measurements were done to indicate the benefits of suction in either study.

1.4 Aim and Objectives of the Project
The above reviews of experimental studies on horseshoe vortex formation and behaviour have identified a number of gaps in understanding such flows and how to manipulate them using suction. The absence of any attempts to use CFD to understand horseshoe vortex effects in half wing testing is also rather surprising, and identifies a further gap. Therefore, aims and objectives of the present study were selected as:
1. An evaluation of the capabilities of CFD for studying junction flows.
2. Use of CFD to visualise and gain an understanding of junction flow.
3. Use of CFD to evaluate the use of localised suction in improving half wing performance, and to aid the design of a wind tunnel experiment to assess the effectiveness of a localised suction system.
4. To conduct wind tunnel measurements of the proposed design and to evaluate its effectiveness.
5. To use the experimental results to access the accuracy of the selected CFD methodology in predicting junction horseshoe vortex flows and the effects of localised suction.

Completion of these aims and objectives is described in the following chapters.
2 CFD Methodology Selection and Validation

The initial task for the present CFD study focussed on an evaluation of the performance of a Reynolds Averaged Navier-Stokes (RANS) turbulence model for 2D aerofoil flow. The experimental measurements reported by McGhee & Beasley\(^ {31}\) for a modified version of an LS(1)-0413 aerofoil were selected since this aerofoil section was to be used later in the wind tunnel study (see below, Chapter 5). A 2D CFD analysis of this aerofoil would be a building block towards 3D lifting half wing computations. The second stage of validation of the CFD methodology was therefore to extend the 2D aerofoil predictions to a 3D wing on a solid surface, to examine the ability to predict a horseshoe vortex. For this purpose the experiments of Devenport & Simpson\(^ {32}\) for an uncambered NACA 0020 aerofoil on a wall were used.

2.1 CFD Computational Details-2D Aerofoil Flow

The commercial CFD package Fluent\(^ {33}\) was used to conduct the CFD computations, aided by panel method plus viscous boundary layer calculations using the XFOIL\(^ {34}\) code. Predictions were made for chord Reynolds numbers of 1 and 2 \(\times 10^6\) for the incidence range between -2.0\(^ 0\) and 12.0\(^ 0\) in 2.0\(^ 0\) steps. Incidence angle was varied by varying the onset velocity relative to the aerofoil chord via the boundary condition upstream of the aerofoil.

(a) Turbulence Model

A low Reynolds number Spalart-Allmaras turbulence model\(^ {35}\) was selected for analysis of the modified LS(1)-0413 aerofoil; low Reynolds number referred to the ability of the turbulence model to predict the flow within the boundary layer right down into the laminar sub-layer. This turbulence model is widely used for predictions of wall-bounded flows and it is believed to perform well for boundary layers subjected to adverse pressure gradients\(^ {36}\). This aspect of the turbulence model should be important in the planned three dimensional half wing application, since this involves junction flow separation which originates due to the adverse pressure gradient effects upstream of the wing leading edge.

(b) Free-stream Turbulence Level

When using the Spalart-Allmaras model, some estimate has to be made for the onset flow turbulent viscosity that must be set as an inlet flow boundary condition. No information on this was provided in the experimental data report\(^ {31}\). An estimate was thus made based on the free-stream turbulence values.
measured in the Loughborough University wind tunnel to be used for the experimental part of the present research, assuming this to be a typical aerodynamic experimental facility. Free-stream turbulent viscosity may be related to the turbulence intensity and length scale in the wind tunnel via:

\[ \nu_{r,\infty} = \sqrt{\frac{3}{2}} \left( \nu_{avg} \right) (I)(l) \]  

(2.1)

where:

'I' is the turbulence intensity-a value of 0.15 % has been recorded in for the Loughborough University wind tunnel.

'I' is the turbulent length scale; the hydraulic diameter (9.525 mm) of the cell size used in the honeycomb screens in the wind tunnel was chosen for this parameter.

'\( V_{avg} \)' is the mean flow velocity; 40 m/s

Thus, ‘\( \nu_{r,\infty} \)’ was calculated as 0.0007 m\(^2\)/s. This value is approximately 50 times the molecular kinematic viscosity; implying a reasonably turbulent onset flow. Perhaps the length scale chosen is incorrect (too large) but since no other information was available, this estimate was considered acceptable.

(c) Solution Domain and Computational Grid
The schematic solution domain shape used for the 2D analysis is shown in Fig. 2.1. The grid used is C-type with its origin at the aerofoil leading edge and expanding cell size downstream of the trailing edge. A semi-circular upstream boundary was used to avoid large grid cell deformation, particularly at the aerofoil leading edge. Expansion of the grid lines downstream of the trailing edge helped to maintain lower cell aspect ratio in this region.
Before the grid size to be used for final analysis was selected, a grid independence and grid sensitivity study at 0.0° and 10.0° incidence were undertaken at a Re = 2 x 10⁶. This investigation focused on the effects of grid changes on the three primary aerodynamic coefficients, i.e. lift, drag and pitching moment.

2.1.1 Grid Independence Study

A grid independence study was carried out to examine the effects of the following parameters:

(a) Distance of the far-field boundary from the aerofoil surface

(b) Total number of grid points

In all these initial studies, the Reynolds number, incidence angle and the value of ‘\( v_{\infty} \)’ quoted above were used to fix flow properties along the black velocity inlet boundary shown in Fig. 2.1. An outflow boundary condition was specified on the remaining blue part of the solution domain.
(a) Distance of far-field boundary from the aerofoil surface
Predictions were made for solution domains with far-field boundaries located at 25 and 50 chords distance (Fig. 2.2). Grid size was increased for the larger distance to ensure that the near aerofoil mesh stayed the same and only the far-field boundary’s distance changed. The grid sizes used in these predictions were 253 (wrapped around aerofoil) x 213 (transverse to aerofoil) and 259 x 247, respectively. The results indicate little difference (about 1.5 %) between boundary locations of 25 and 50 chords, see Table 2.1. This domain size and grid, designated as 'G0' were chosen as the datum case against which other predictions were subsequently compared. 'G0' was chosen since a far-field distance of 50 chord lengths was considered safer to prevent any interaction of the circulation generated by the aerofoil with the simple assumption made (fixed velocity) for the boundary condition on the far field boundary.

![Grids with varying far-field boundaries](image)

*Fig. 2.2: Grids with varying far-field boundaries*

<table>
<thead>
<tr>
<th>Grid No.</th>
<th>Far-field distance</th>
<th>0.0° Incidence</th>
<th>10.0° Incidence</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Cl</td>
<td>Cd</td>
<td>Cm</td>
</tr>
<tr>
<td>G-A1</td>
<td>25c</td>
<td>0.4049</td>
<td>0.0115</td>
</tr>
<tr>
<td>G0</td>
<td>50c</td>
<td>0.4052</td>
<td>0.0114</td>
</tr>
</tbody>
</table>

*Table 2.1: Variation of predicted aerodynamic coefficients with far-field distance*
(b) **Total number of grid points**

The cell number for the traverse direction (aerofoil surface up to the far-field boundary) was altered whilst keeping constant the number of points around the aerofoil surface. This resulted in the total grid node number varying from 25.4 K to 151 K (Fig. 2.3).

![Grids with varying traverse grid points](image)

Table 2.2 shows that the difference between the drag values at 10.0° incidence between 'G-B1' and 'G0' was greater than 30% but reduced to 4.5% between 'G0' and 'G-B2'. Other coefficients at both incidences were within 0.3%. Considering the increased computational time on the finer mesh, the results from 'G0' were considered acceptable and grid 'G0' was chosen for all further investigations and analysis.

<table>
<thead>
<tr>
<th>Grid No.</th>
<th>Total grid points</th>
<th>0.0° Incidence</th>
<th>10.0° Incidence</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Cl</td>
<td>Cd</td>
</tr>
<tr>
<td>G-B1</td>
<td>25.4 k</td>
<td>0.4022</td>
<td>0.012</td>
</tr>
<tr>
<td>G0</td>
<td>58.7 k</td>
<td>0.4052</td>
<td>0.0114</td>
</tr>
<tr>
<td>G-B2</td>
<td>151 k</td>
<td>0.4060</td>
<td>0.0114</td>
</tr>
</tbody>
</table>

Table 2.2: Variation of predicted coefficients with total number of grid points
2.1.2 Grid sensitivity study

Before the grid was finalised, further grid sensitivity calculations were conducted by assessing the sensitivity of predicted aerodynamic coefficients to variations in the following mesh design parameters:

(a) Number of grid points on the aerofoil surface
(b) Distance of the first cell from the aerofoil surface

(a) Number of grid points on the aerofoil surface

Enough grid points are required in the vicinity of the aerofoil leading edge to capture the sharply rising suction peak at high incidence. XFOIL\(^\text{34}\) predictions (Fig. 2.4) showed that the suction peaks were located within the first 10% of the chord for the \(0.0^0\) to \(10.0^0\) incidence range. This code is known to predict transition accurately for low to moderate Reynolds numbers\(^\text{39}\). The number of grid points over this distance, i.e. from leading edge up to 10% chord on both suction and pressure surfaces, was therefore varied in the RANS CFD calculations.

![Cp distribution for Modified LS(1)-0413 predicted by XFOIL code](image)

Fig. 2.4: Cp distribution for Modified LS(1)-0413 predicted by XFOIL code

The total number of points around the aerofoil was varied from 20 to 160 distributed equally on suction and pressure surfaces. Fig. 2.5 shows the aerofoil leading edge region with 20 and 80 grid points. The results of the predictions are shown in Table 2.3.
No simulations were conducted at 10.0° for 'G-C1' as the grid did not capture the suction peak at 0.0° incidence. As the number of aerofoil surface points increased the coefficients asymptoted to constant values. Coefficients at 0.0° incidence converged with fewer grid points compared to 10.0°. Grid 'G0' produced results within 1% compared to the finest resolution used, 'G-C4', at both incidence angles; thus once again grid 'G0' was considered adequate for further predictions.

(b) Distance of the first cell from aerofoil surface
To obtain acceptable predictions of the skin friction component of drag with low Reynolds number RANS turbulence models, the near wall cell size was chosen using the recommendation\(^{10}\) that a \(y^+\) value of 1.0 or less should be targeted (\(y^+\) is a non-dimensional wall distance, using the friction velocity and fluid kinematic...
viscosity as non-dimensional factors). To achieve this, the height of the first cell above the surface was initially estimated as 0.0087 mm using the following calculation\textsuperscript{41}.

\[
\text{First Cell Height} = \frac{(y^+)(L^{0.125})(\mu^{0.875})}{(0.199)(V^{0.875})(\rho^{0.875})}
\]  \hspace{1cm} (2.2)

where,

'L' is the characteristic length-aerofoil chord

'\mu' is fluid viscosity

'V' is free-stream velocity

'\rho' is fluid density; 1.225 kg/m\textsuperscript{3}

A lower value of 0.006 mm had previously been used in grid 'G0' to allow for the increase in \(y^+\) at higher incidence. Sensitivity predictions were made by altering the distance of the first cell to 0.004 mm and 0.008 mm. No significant difference was found in the results (Table 2.4). The 'G0' grid with 0.006 mm first cell height was considered suitable.

<table>
<thead>
<tr>
<th>Grid No.</th>
<th>First cell height (mm)</th>
<th>0.0° Incidence</th>
<th>10.0° Incidence</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>(C_l)</td>
<td>(C_d)</td>
<td>(C_m)</td>
</tr>
<tr>
<td>G-D1</td>
<td>0.004</td>
<td>0.4053</td>
<td>0.0115</td>
</tr>
<tr>
<td>G0</td>
<td>0.006</td>
<td>0.4052</td>
<td>0.0114</td>
</tr>
<tr>
<td>G-D2</td>
<td>0.008</td>
<td>0.4052</td>
<td>0.0115</td>
</tr>
</tbody>
</table>

\textbf{Table 2.4: Predicted coefficients with varying first cell height}
2.1.3 Final Grid Selected

The final grid selected 'G0' had a total of 58.7 K nodes; Fig. 2.6 shows the grid over the complete domain and zoomed-in views near the aerofoil.

![Grid images](image.png)

Fig. 2.6: G0-Grid selected for final CFD Studies

2.2 Analysis of Predicted Results-2D Aerofoil Flow

(a) Fully turbulent predictions

The target $y^+$ value of 1 was maintained for low incidence angles. At higher incidence the value increased to a maximum of a little less than 2.5 for 10.0° incidence. A typical $y^+$ distribution over the aerofoil, not including the trailing edge values, for 0.0° and 10.0° is shown in Fig. 2.7. The orientation of the blunt trailing edge and particularly the flow separation behind the trailing edge (Fig.
2.8) makes its contribution to skin friction drag almost negligible and therefore the $y^+$ values on the trailing edge are not considered significant.

![Fig. 2.7: Wall $y^+$ plot, Re=2 x 10^6](image)

Values of $y^+$ larger than 1.0 at the leading edge were caused due to the high acceleration at large incidences leading to increased velocity and hence increased wall shear stress at the aerofoil leading edge. It was not possible to achieve a value of less than 1.0 without greatly distorting the grid. However, since the near leading edge flow is laminar at the Reynolds number being considered, a better approach would be to introduce some allowance for transition and this is described below.

![Fig. 2.8: Flow separation predicted behind trailing edge, Re=2 x 10^6](image)
Aerodynamic characteristics obtained from the fully turbulent CFD predictions were compared with the experimental data from Ref. 31. Spalart-Allmaras drag coefficient predictions, based on all turbulent flow (i.e. no allowance for transition), are significantly higher than the experimental values in which natural transition occurred\textsuperscript{31} (Fig 2.9). The predicted lift curve slope ($C_{L_{\alpha}}$) between -2.0$^\circ$ to 6.0$^\circ$ was 0.112 per degree as compared to the measured value of 0.107 per degree. Predicted zero lift angle of attack was 0.45$^\circ$ greater than the experimental value, as shown in Fig. 2.10. The pitching moment coefficient comparison, Fig. 2.11, shows a slightly increased nose-down prediction.

![Graph](image-url)  
**Fig. 2.9:** Drag Coefficient, $Re=2 \times 10^6$
Fig. 2.10: Lift coefficient, Re=2 x 10^6

Fig. 2.11: Pitching moment coefficient, Re=2 x 10^6
(b) Predictions considering laminar/turbulent transition
Further RANS CFD analysis was carried out by introducing an allowance for boundary layer transition. The XFOIL code was used to make preliminary assessment of the transition location. This code uses an \( e^N \) method for boundary layer transition prediction\(^{42} \). On the suction surface the high suction peak at incidence is often the cause of transition occurring close to the leading edge. On the pressure surface a large region of favourable pressure gradient downstream of the leading edge causes the transition to be closer to the trailing edge. As the favourable pressure region grows with increasing incidence, transition moves downstream on the pressure surface. Fig. 2.12 presents results derived from XFOIL for the predicted transition location at various angles of attack for a \( \text{Re} = 2 \times 10^6 \). On the suction surface the transition occurs at approximately 0.6 chord at 0.0\(^0 \) incidence and moves forward rapidly as incidence increases until it is effectively at the leading edge beyond 8.0\(^0 \). On the pressure side at 0.0\(^0 \) the transition is predicted to occur slightly further downstream, about 0.7c, and moves towards the trailing edge as incidence increases.

![Modified LS(1)-0413, Reynolds Number: 2 x 10^6](image)

**Fig. 2.12: XFOIL prediction of transition location, Re=2 x 10^6**

In the present RANS CFD calculations a simple approach to introducing the influence of transition has been adopted. Transition was modelled by dividing the solution domain, into separate laminar and turbulent zones at the transition locations predicted by XFOIL, and suppressing turbulence in the laminar region but allowing the full level turbulence prediction by the Spalart-Allmaras model to enter from the transition location and further downstream. This models transition
over a zero length. Since the transition was set at the location of natural transition predicted by XFOIL, these predictions are termed 'CFD (Natural Transition)' in the following figures. A sketch of the approach adopted, with a close up view of the aerofoil, is shown in Fig. 2.13 for $0^0$ incidence. A similar approach was adopted for other incidence angles using the XFOIL predicted locations from Fig. 2.12.

Predicted contours of turbulent viscosity ($v_t$) around the aerofoil at $10.0^0$ are shown in Fig. 2.14a with a zoomed-in view in Fig. 2.14b. High turbulent viscosity values indicate high turbulent mixing regions in the flow. Regions with zero turbulent viscosity represent laminar flow regions. Laminar and turbulent regions are separated at the boundary where abrupt transition occurs. The $v_t$ contours show regions of high turbulence in the boundary layers on the aerofoil and in the wake region. In the boundary layer region the predicted turbulent viscosity is about 500 times the molecular viscosity; in the wake higher turbulent viscosity values, approximately $10^3$ times the molecular viscosity, are observed. These values mean that the somewhat large values of 50 times the molecular viscosity in the onset flow boundary conditions does not cause any problems in the CFD results presented.
Fig. 2.14: 10.0° incidence-CFD (Natural transition), Re=2 x 10^6
The transition location estimated from XFOIL always lay between two CFD grid points on the aerofoil. For simplicity, in all CFD predictions shown the upstream point was chosen as the transition location.

The improvement in the drag coefficient predictions, when transition was allowed for, was significant, with predictions as close as about ~5 % of the experiment data at lower incidence and within ~25 % at higher lift (Fig. 2.15). Zero lift drag was 84 drag counts compared to an experimental value of 82. At a lift coefficient value of 1.2 (about 8\(^0\) incidence) the predicted value was 150.6 drag counts compared to the measured value of 125.2. Introducing transition into the Spalart-Allmaras predictions marginally improved the lift coefficient comparison by reducing the lift curve slope from 0.1121 per degree to 0.1119 per degree and improving the zero lift incidence comparison by 0.25\(^0\) (Fig. 2.16). Spalart-Allmaras predicts a delayed stall as compared to wind tunnel measurements. Pitching moment predictions showed an increased nose down trend, but was within 10 % of the experimental data (Fig. 2.17). Pitching moment coefficient (about quarter chord) for most of the incidence operating range was -0.098 compared to the experimental value of -0.09.

![Fig. 2.15: Predicted Drag Coefficient, Re=2 x 10^6](image)
Fig. 2.16: Predicted Lift Coefficient, Re=2 x 10⁶

Fig. 2.17: Predicted Pitching Moment Coefficient, Re=2 x 10⁶
Predicted particle pathlines and wall shear stress trends at 10.0° incidence (Fig. 2.18) showed no separation. A zoomed-in view of pathlines at 12.0° incidence showed a small and thin separation bubble on the suction surface towards the trailing edge. This is mirrored in the shear stress plots where the velocity changes direction on the suction surface (Fig. 2.19).

![Pathlines and Shear Stress Graphs](image)

**Fig. 2.18:** 10.0° incidence, CFD (Natural Transition) predictions, Re=2 x 10^6
CFD simulations carried out at the lower Re = 1 x 10^6 showed similar comparison with the experiment as at 2 x 10^6. Predicted drag values were within 15% of experimental data for almost the complete lift range (Fig. 2.20). The predicted lift curve slope was 0.1107 per degree compared to 0.1035 per degree from experiment (Fig. 2.21). The error in pitching moment coefficient prediction for lift values up to 0.8 was about 7% and increased to 15% for higher lift (Fig. 2.22). Finally, a trend comparison of the aerodynamic coefficients with Reynolds number was carried out. Experimental lift and drag values are shown in Fig. 2.23 where lift coefficient is plotted against incidence angle and drag coefficient against lift coefficient. A reduction in drag and an increase in the lift curve slope is seen with increasing Reynolds number. A similar trend for lift and drag are seen in the RANS predictions (Fig. 2.24).
Measured pitching moment coefficients do not show any definite trend with Reynolds number (Fig. 2.25). Spalart-Allmaras RANS CFD predictions show a more nose-down moment for the complete lift range at Re = 2 x 10^6 (Fig. 2.26).

**Fig. 2.20: Predicted Drag coefficient, Re=1 x 10^6**

**Fig. 2.21: Predicted Lift coefficient, Re=1 x 10^6**
**Fig. 2.22:** Predicted Pitching moment coefficient, Re=1 x 10^6

**Fig. 2.23:** Experimental Lift and Drag Coefficient
CFD Methodology Selection and Validation

**Fig. 2.24**: Predicted Lift and Drag Coefficient, Spalart-Allmaras

**Fig. 2.25**: Experimental Pitching Moment Coefficient\(^{31}\)
2.2.1 Summary-2D Aerofoil flow

The performance of the Spalart-Allmaras turbulence model in a RANS CFD code was evaluated for a modified LS(1)-0413 2D aerofoil. A simple method for allowance for transition was introduced, involving a combination of RANS CFD and the XFOIL code. Subsequent predictions significantly improved the comparison with the experiment particularly for drag coefficient. Performance of the turbulence model was considered acceptable when compared to the experimental data for a range of incidence angles and Reynolds numbers.

2.3 Evaluation of Spalart-Allmaras Model for Junction Flow Predictions

The 2D RANS CFD aerofoil predictions reported above have demonstrated that the modelling approach adopted can be used for aerofoil aerodynamics. It was now necessary to evaluate the performance of this approach for 3D wing flow including onset floor boundary layer/horseshoe vortex elements. A well-chosen test case was therefore needed. A wing(floor) junction horseshoe vortex flow has been studied experimentally by Devenport & Simpson\textsuperscript{32} with the recent contribution to the database made by Ölçmen & Simpson\textsuperscript{43}. The study was carried out at $0^\circ$ incidence and at Re = $0.58 \times 10^6$. The measurements of the Devenport & Simpson\textsuperscript{32} experiment have been extensively used to evaluate
performance of various turbulence models in junction flow predictions\textsuperscript{44-49}. Among these references, considering the physical and the turbulence model used, the work of Apsley & Leschziner\textsuperscript{44} and Paciorri et al.\textsuperscript{45} is most relevant to the current research and are hence discussed here.

Apsley & Leschziner\textsuperscript{44} reported various computational studies carried out in partnership with UMIST, Loughborough University, BAE Systems, DERA, ARA and Rolls Royce Aeroengines. The relative performance of various RANS turbulence models, i.e. linear eddy viscosity models (variants of $k$-$\varepsilon$ and $k$-$\omega$), non-linear eddy viscosity models (all of the $k$-$\varepsilon$) and Reynolds Stress Models (RSM). The comparison showed that all the turbulence models capture the basic structure, i.e. separation upstream of the wing leading edge which rolls into a vortex and is conveyed downstream around the wing. In contrast to a good qualitative comparison, quantitative comparison is poor for all the models, under-predicting the shape and size of the vortex and with only an approximate estimation of the turbulent quantities; although the RSM models perform marginally better than the eddy viscosity models. The predictions of the surface pressure on the wing and the floor is affected accordingly. Paciorri et al.\textsuperscript{45} study also showed that RANS predictions can effectively capture the junction horseshoe vortex. Comparison of predictions with experiment for pressure distributions on the floor around the wing in Ref. 45, showed that the one equation Spalart-Allmaras slightly outperformed the two equation $k$-$\omega$ turbulence model in predicting the locations of the floor separation line and the vortex in the junction.

Gand et al.\textsuperscript{50} also conducted a similar study using wind tunnel measurements and CFD, employing Large Eddy Simulations (LES), for a NACA 0012 wing/floor junction. The study was conducted at $7^0$ incidence and at $Re = 0.28 \times 10^6$ with more emphasis on the unsteady flow characteristics. The investigations showed that the LES predictions were not perfect but agreed well with the experimental results.

In the current study, CFD is employed as a tool for junction flow study with more focus on its ability to predict flow structures rather than actual values of the involved variables. The above studies\textsuperscript{44,45} showed that RANS based eddy viscosity turbulence models may effectively be used to predict the basic flow features of a junction flow. Hence these methods may be used as a tool in the current study. Therefore, the turbulence model previously used for 2D aerofoil
calculations (Spalart-Allmaras) has been applied to the Devenport & Simpson\textsuperscript{32} experiments.

The aerofoil section used in the experiments of Ref. 32 was the so-called “Rood” aerofoil, (named after its designer E.P. Rood) which is a modified version of the NACA 0020 section\textsuperscript{18}. The original NACA profile, upstream of its maximum thickness, was replaced with an elliptic profile having a cross section ratio of 3:2, with its major aligned with the chord of the aerofoil (Fig. 2.27).

![Fig. 2.27: Rood Aerofoil Section](image)

### 2.4 Computational Details-3D Junction Flow

The onset flow conditions and wing geometry were the same as those used in the experiment of Devenport & Simpson\textsuperscript{32}. The junction was formed from a rectangular wing, of the aerofoil cross-section described above, mounted on a flat plate on which an oncoming turbulent boundary layer developed. The wing, with a span 0.229 m and a chord of 0.305 m, was tested at 0.0\textdegree incidence (hence non-lifting flow) and with a free-stream velocity of 27 m/s, resulting in a $Re = 0.58 \times 10^6$. The wing was tested in a test section of height and width of 250 mm and 1000 mm, respectively. In the current study, the experimental results of Devenport & Simpson\textsuperscript{32} have been taken from the report by Jones & Clark\textsuperscript{48} and Veloudis\textsuperscript{49}. The experimental setup and the corresponding coordinate frame used to present the predictions are shown in Fig. 2.28.
Since predictions were to be made for a symmetric aerofoil at $0.0^\circ$ incidence, only a symmetric half of the aerofoil cross section needed to be considered. With the x-axis in the streamwise direction the symmetry plane (x-y plane) was represented as a non-viscous wall and the wind tunnel wall, floor and ceiling were modelled as viscous walls. The grid was constructed by extruding a two dimensional grid, in the x-z plane, in the spanwise (y) direction. The solution domain dimensions are shown in Fig. 2.30. To reproduce exactly the experimental setup, the predictions were made with a gap between the wing and the wind tunnel ceiling (since the wing is not producing any lift, no wing tip vortex is formed, and so the grid in this region did not need to be fine). The distance of the upstream inflow boundary from the wing leading edge was the same as used by Veloudis$^{49}$ to allow the development of a boundary layer on the tunnel floor. A ‘Velocity Inlet’ boundary condition with a flat velocity profile was selected as opposed to a velocity profile based on the experiment measurements of the floor boundary layer. This was to allow the development of a boundary layer on the tunnel floor which could be compared in the CFD predictions with measured data at the wing leading edge.

The selected grid has a total of 60 points along the wing span (y) concentrated near the floor, with at least 25 points within the region formed by the onset flow boundary layer. The height of the first cell above the floor was 0.1 mm, a value used by Jones & Clark$^{48}$ in a similar study of the Rood junction flow. The distance of the first cell from the wing surface (both in x and z directions), was 0.01 mm, resulting in surface $y^+$ values of less than 1.0. The total number of grid
points wrapped around the aerofoil was 193 and 45 points traverse to the aerofoil. These were selected so as to avoid any abrupt changes in grid cell size of neighbouring cells. Grid point distribution around the aerofoil is shown in Fig. 2.29.

![Overall view](image1.png) ![Leading edge close-up view](image2.png)

**Fig. 2.29: Grid around the Rood aerofoil**

![Top view](image3.png) ![Side view](image4.png)

**Fig. 2.30: Three dimensional flow domain**

### 2.5 Analysis of predicted Results-3D Junction Flow

As described in Chapter 1, the horseshoe vortex originates due to flow reversal of the tunnel floor boundary layer. This occurs in the plane of symmetry just upstream of the wing leading edge ($x = 0$) and is indicated by negative axial velocity in the near wall region. Fig. 2.31 shows the predicted boundary layer profile at three axial locations compared with experiments. For both
predictions and experiment reverse flow is present at \( x/c = -0.036 \) and \( x = -0.012 \) upstream of the leading edge and may be noted in the zoomed-in view (Fig. 2.32). The upstream flow development and prediction of negative velocity is similar to the experimental measurements, but a fuller predicted profile perhaps indicates a relatively higher turbulence compared to that seen in experiments. The simple inflow boundary conditions as described above have given a reasonable prediction of the approach flow boundary layer (see \( x/c = -0.2 \) station), but this is not perfect. The limitations of the inlet eddy viscosity used will contribute to this discrepancy.

**Fig. 2.31: Boundary layer development in symmetry plane**

**Fig. 2.32: Boundary layer development in symmetry plane (Zoomed-in view)**
The horseshoe vortex creates a strong three dimensional flow structure. When studied via flow visualisation of predicted particle paths close to the tunnel floor, distinct schematic patterns on the wind tunnel floor are created, that provide valuable information about the junction flow. These patterns are characterised by the presence of a saddle point, a nodal point of attachment, a separation and an attachment line\textsuperscript{51}. These characteristics are easily identified in the present junction flow predictions (Fig. 2.33).

![Diagram showing patterns around wing and saddle point zoomed view]

a): Patterns around wing  
b): Saddle point – Zoomed view

Fig. 2.33: Predicted patterns on tunnel floor for Rood half wing

The saddle point is a point of zero skin friction and two sets of skin friction lines pass through it; one brings flow towards the point and the other takes flow away. The saddle point is a feature of two dimensional separation where the skin friction drops to zero and $180^\circ$ flow reversal occurs\textsuperscript{13}. Lines approaching the saddle point from the upstream direction are diverted away from the aerofoil side surface. The line through the saddle point is referred to as the separation line. The surrounding flow moves along the separation line. The nodal attachment point sits just forward of the wing leading edge and is characterised by an infinite number of skin friction lines passing through it (Fig. 2.34a). The direction of the streamlines, i.e. away from the nodal point, shows that it acts like a flow source.

This picture of the flow pattern on the floor may be complemented by velocity vectors in a vertical plane through the symmetry line. This shows velocity vectors at the rear of the horseshoe vortex flow and the flow behaviour adjacent to the wing leading edge (Fig. 2.34b). The flow which moves away from the
nodal point is fed from the vertical (y) direction. At this point the flow moving on the floor bifurcates, moving along the floor and away from the nodal point in both upstream and downstream directions.

Flow moving in the downstream direction very close to the leading edge rolls up into a vortex termed a ‘secondary vortex’. This vortex bends downstream as shown in Fig. 2.35, and stays close to the wing root surface as it travels downstream. Fig. 2.34b indicates the rotational direction of the vortex, as it sweeps flow from the floor onto the wing.

The flow travelling in the upstream direction from the nodal point separates from the floor at the saddle point, which in the present prediction is located approximately at x/c = -0.115 (x = 35 mm) upstream of the wing leading edge.
(Fig. 2.33). This separation rolls into the horseshoe vortex core, visible in the symmetry plane, with the eye of the vortex located at \( x/c \approx -0.054 \) (\( x = 16.5 \text{ mm} \)) and \( y/\text{span} \approx 0.0096 \) (\( y = 2.2 \text{ mm} \)) as is shown in Fig. 2.36a. A zoomed-in view of the predicted velocity vectors in the symmetry plane is shown in Fig. 2.36b.

\[ \text{a): Overall View} \quad \text{b): Zoomed-in view} \]

**Fig. 2.36: Predicted velocity vectors in the symmetry plane**

The vortex core shown in Fig. 2.36a is the origin of the junction horseshoe vortex shown in Fig. 2.37 which is originally confined between the saddle point and the nodal attachment point before it bends downstream. The rotation direction of this vortex is opposite to the secondary vortex as indicated in Fig. 2.36a. The horseshoe vortex is formed upstream of the secondary vortex and when bent downstream is further from the wing root compared to the latter.

\[ \text{a): Top view} \quad \text{b): 3D view} \]

**Fig. 2.37: Junction horseshoe vortex in Rood junction**
Experimental\textsuperscript{32} velocity vectors in the symmetry plane are shown in Fig. 2.38. The size and location of the predicted flow features are comparable to the experimental values - the distance between the wing and saddle point from experiment, (approximately \(x/c = -0.115\textsuperscript{52}\)), is the same as obtained from the predictions. The predicted axial location of the vortex core is slightly larger than the experimental value of \(x/c \approx -0.047\textsuperscript{52}\) and its height is less than that measured (\(y/\text{span} \approx 0.016\textsuperscript{52}\)).

Thus, compared with experimental results the predictions result in a vortex core circulation region that is more confined towards the floor and the wing. This is also evident when examining the comparison between predicted and measured axial velocity contours (Fig. 2.39) in which the predicted negative velocity contour bubble is smaller in size and more confined in the junction corner compared to experimental measurements.
The vortex whose symmetry plane core is visualised above is convected around the aerofoil in the wing/floor junction developing into the 'legs' of a classical horseshoe vortex. Flow features in the three axial section axial planes located along the aerofoil, as shown in Fig. 2.40, are used to portray the streamwise development of this vortex.

Fig. 2.40: Axial planes of interest

Fig. 2.41a shows predicted axial velocity contours (in the y-z plane) and streamlines at the first chordwise station location of 0.179c (In Fig. 2.41 the wing surface is located at z/c = 0.12). This station is located just downstream of the aerofoil maximum half thickness (35.8 mm) at x = 0.176c. The predicted boundary layer thickness on the wing, at large values of y, is small, less than 0.5 mm (z/c = 0.0016). In the junction region the boundary layer thickness increases to about 2 mm (z/c = 0.0066) as it interacts with the floor boundary layer. On the other hand the boundary layer thickness on the junction floor is small but increases greatly away from the wing. The reduction in boundary layer size near the wing surface is because of flow acceleration at the wing leading edge. The predicted boundary layer on the floor may be compared with the measurements shown in Fig. 2.41b. Experimental measurements are not available close to the wing.
Axial velocity contour predictions and pathlines at the second axial location of 0.64c are shown in Fig. 2.42a. It is noted that compared to the upstream location the flow pathlines have changed direction (z-direction). This is attributed to the flow now following the inward oriented shape of the aerofoil profile, at this midspan station. Pathlines in the lower left corner of the plot (Fig. 2.42a) indicate development of the horseshoe vortex, as will be discussed further below. Comparison of the predicted contour shapes and the experiment (Fig. 2.42b) again indicate under-prediction in the size of the vortex at this axial location, but the general shape in vortex development is comparable.
Axial velocity and junction flow patterns at the final location of 1.05c are shown in Fig. 2.43. The horseshoe vortex which had become just visible at 0.64c has become fully formed at 1.05c. The boundary layer thickness variation along the floor is explained by the pathlines which indicate the presence of the ground junction horseshoe vortex. Low momentum fluid drawn away from the floor by the vortex near z/c = 0.23 (z = 70 mm) disturbs the shape of the low momentum region above, whereas the downward flow near the wing brings high momentum fluid towards the floor. These predicted trends are supported by measurements although the larger height of the vortex in the measurements is still indicated.

Predicted flow patterns on the wing surface itself in the junction region are shown in Fig. 2.44. Marked flow features are the secondary vortex and a flow separation at the trailing edge. The separation is the result of the high adverse pressure gradient seen towards the wing trailing edge (Fig. 2.45). Close proximity of the wing tip with the tunnel roof modifies the flow patterns towards the tip but these are significantly different compared to the floor junction flow due to the presence of a gap.

![Fig. 2.43: Axial velocity contours in x = 1.05c plane](image-url)
The presence of the horseshoe vortex affects the sectional lift and drag coefficients in the junction region; reduced lift and enhanced pressure drag are expected in the junction region\(^4\). In the current study a reduced suction peak at a ‘y/span’ location of 0.042 (y = 9.5 mm) from the tunnel floor showed reduced area under the lift coefficient curve compared to a ‘y/span’ location of 0.54 (y = 123.8 mm) as is shown in Fig. 2.45a. The predicted trends of the pressure coefficient with span location are similar to the experimental measurements (Fig. 2.45b), although the Spalart-Allmaras model under-predicts the loss in lift as compared to the experimental values, presumably because of the smaller size vortex prediction.
A composite overview of the junction flow obtained by putting together the individual flow features discussed above is presented in Fig. 2.46. The figure shows flow patterns in the symmetry plane, tunnel floor, wing surface and an axial plane located at the trailing edge.

![Fig. 2.46: Overview of junction flow prediction](image)

### 2.6 Summary-3D Junction Flow

The aim of the current CFD investigation was to assess the potential of the Spalart-Allmaras turbulence model for junction flow predictions. This was accomplished by comparing CFD predictions with experimental measurements\(^{32}\). The turbulence model together with a RANS CFD code captured the general features of a 3D separation, e.g. nodal attachment point, saddle point and the separation line\(^{51}\). The predictions underestimated the size of the circulation in the symmetry plane where the horseshoe vortex originates and hence the magnitude of the junction flow effects as the flow developed in the streamwise direction. These trends were evident in the wing surface pressure distributions where CFD predictions showed smaller losses in the junction compared to the experiment. The study supports the findings of Apsley & Leschziner\(^{44}\) that showed good qualitative comparison. Also, under-prediction of vortex size is in line with the Paciorri et al.\(^{45}\) findings for the Spalart-Allmaras turbulence model predictions of junction flow. However, it was concluded that the Spalart-Allmaras model performed reasonably well in capturing the junction flow effects and hence it was decided to use this CFD approach to investigate the more complex flow of a lifting aerofoil, without and with suction, as described in the following two chapters.
3 CFD Predictions of Lifting Half Wing Characteristics

Flow over a lifting half wing, mounted in the test section of the Loughborough University wind tunnel has been modelled using essentially the same RANS CFD approach applied to the non-lifting Rood aerofoil half-wing. Predictions have been made with and without the wind tunnel floor boundary layer to represent a "real life" half-wing and an ideal half-wing, respectively. In the following, the former is termed as "Half-Wing" and the latter as "Ideal Half-Wing". These investigations not only provided aerodynamic coefficients for various cases but more importantly a detailed insight into wing/floor junction flow under lifting conditions.

3.1 3D Computational Details
The half wing, with a modified NASA LS(1)-0413 cross section, had a rectangular planform with a physical aspect ratio of 3.02. Predictions were made for a $Re = 0.89 \times 10^6$ at $0.0^\circ$ and $10^\circ$ incidence. The half wing was located in a wind tunnel cross-section with dimensions representing Loughborough University wind tunnel in which measurements were to be made (details in Chapter 5).

3D Streamwise/Transverse Grid (x-z)
The grid for 3D predictions was build up from a 2D aerofoil section grid in the x-z plane, extruded in the third (y-spanwise) direction; for convenience in these 3D predictions the y-axis was aligned with the wing span. The 2D grid was a modified version of the grid used earlier to predict the 2D unconfined aerofoil flow (in the first part of Chapter 2) tailored to suit the wind tunnel geometry and dimensions. Thus, the solution domain width was set equal to the test section width and the inflow boundary location upstream of the wing leading edge was set so that the floor boundary layer in the half wing predictions had sufficient distance to grow to a thickness representative of the measured wind tunnel boundary layer close to the wing leading edge location. The distance upstream of the wing leading edge was estimated using flat plate theory.

Around the wing, up to and just downstream of the trailing edge, the same grid distribution used in earlier 2D predictions was maintained for about three downstream chord distances. Incidence angle was introduced by using wing
rotation around its quarter chord position in order to mirror the approach adopted in the experiment. Lateral boundaries in the modified grid were modelled as viscous walls to represent the blockage effect of the wind tunnel. Modelling with the modified 2D grid represented the classical scenario of testing a 2D wing in a wind tunnel where flow is constrained by wind tunnel walls and measurements require corrections to obtain free-air values. To check this, applying lift interference and blockage corrections, based on linearised potential flow theory\textsuperscript{53}, to predictions from the modified (x-z) grid effectively reproduced those reported earlier for unconstrained free-stream conditions. The modified grid has a total of 215 × 262 cells in x and z directions, respectively. The grid around the wing in the x-z plane for 10.0° incidence is shown in Fig. 3.1.

![Grid distribution in x-z plane](image)

Fig. 3.1: Grid distribution in x-z plane
Spanwise (x-y) Grid
The grid in the 3rd (y) spanwise direction had a total of 118 grid points with the height of the first cell above the floor being 0.1 mm. Wing-tip and wing/wall junction were regions where highly three dimensional flows were expected and hence grid points were concentrated in these region. At the wing leading edge location there were about 25-30 grid points in the onset boundary layer region, based on boundary layer measurements in an empty tunnel. Outside of the boundary layer and up to the wing tip there were 39 points along the span. The distance of the first cell on either side of the wing tip (y direction) was 0.5 mm. Between the wing tip and the tunnel ceiling there were a further 49 grid points with the distance of the first cell from the wind tunnel wall being 0.1 mm. The distribution of the grid points in the spanwise direction is shown in Fig. 3.2, and the solution domain used for the 3D predictions is shown in Fig. 3.3.

![Fig. 3.2: Grid distribution in x-y plane](image-url)
3.1.1 Half Wing and Ideal Half Wing predictions

The Spalart-Allmaras turbulence model without incorporating transition effects was selected for the half wing predictions. In this study the main focus was on the junction flow and not on the wing aerodynamic loads and since the onset flow had a relatively thick and turbulent boundary layer, it was not expected that transitional effects would be important. This was supported by Mendelsohn & Polhamus' observation that the turbulent mixing of the floor boundary layer with the wing boundary layer prevented any laminar separation or transitional effects on the fraction of the span located within the boundary layer. For inlet conditions a flat velocity profile of magnitude 40 m/s was specified. Prior to making predictions for the half-wing, the predicted boundary layer profile on the wind tunnel floor in an empty tunnel was compared (Fig. 3.4) with measurements in an empty tunnel made at the turntable centre coinciding with the wing quarter chord location (see Chapter 5). The profile measured in the present study compared well with earlier measurements made by Johl indicating that no significant change in the boundary layer characteristics had occurred. The measured boundary layer thickness of 57.5 mm was slightly greater than the predicted thickness of 52 mm, displacement thickness in the experiment was 7.1 mm as compared to a predicted value of 5.8 mm. The predictions showed a fuller boundary layer profile as compared to measurements indicating possibly higher turbulence levels in the predictions, although no measurements were conducted to confirm this.
3.1.2 CFD Results and Analysis

(a) Ideal Half Wing flow patterns

Ideal half wing predictions were made with the same grid but with the tunnel floor modelled as a 'non-viscous wall' and without any onset flow boundary layer (Fig. 3.5a). Thus there were no signs of a junction horseshoe vortex on the tunnel floor (Fig. 3.5b). Flow patterns on both wing surfaces showed no evidence of horseshoe vortex or secondary vortex. The only flow feature observed on the wing surface was the tip vortex (Fig. 3.6).

Fig. 3.5: Predicted flow patterns for ideal half wing, 10.0° incidence
The trends in aerodynamic coefficients for lift and drag, for half wing versus ideal half wing predictions were as expected from literature\(^1\),\(^4\), i.e. a decrease in lift and increase in drag (Table 3.1). Losses at \(0^\circ\) incidence were small but increased significantly at \(10.0^\circ\) incidence. At this incidence, the pitching moment coefficient for the ideal wing was more nose down. Increase in the total drag was due to an increase in pressure drag for the half wing as shown in the drag breakdown in Table 3.2.

<table>
<thead>
<tr>
<th></th>
<th>(0^\circ) incidence</th>
<th></th>
<th>(10.0^\circ) incidence</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>(C_L)</td>
<td>(C_D)</td>
<td>(C_{M,a.c.})</td>
</tr>
<tr>
<td>Ideal Half Wing</td>
<td>0.303</td>
<td>0.01789</td>
<td>-0.0819</td>
</tr>
<tr>
<td>Half Wing</td>
<td>0.2999</td>
<td>0.0182</td>
<td>-0.0815</td>
</tr>
</tbody>
</table>

**Table 3.1: Force and moment coefficient predictions**

<table>
<thead>
<tr>
<th></th>
<th>Ideal Half Wing</th>
<th>Half Wing</th>
</tr>
</thead>
<tbody>
<tr>
<td>(0.0^\circ) incidence</td>
<td>(C_D)\text{Pressure}</td>
<td>0.0077</td>
</tr>
<tr>
<td></td>
<td>(C_D)\text{Viscous}</td>
<td>0.0102</td>
</tr>
<tr>
<td>(10.0^\circ) incidence</td>
<td>(C_D)\text{Pressure}</td>
<td>0.063</td>
</tr>
<tr>
<td></td>
<td>(C_D)\text{Viscous}</td>
<td>0.0096</td>
</tr>
</tbody>
</table>

**Table 3.2: Drag breakdown predictions for half wing and ideal half wing**

The predicted coefficient trends were in accordance with the observed flow pattern; the half wing flow pattern was characterised by the presence of a junction horseshoe vortex that grows rapidly in size as incidence increases from \(0.0^\circ\) to \(10.0^\circ\), as is discussed next.
(b) Half Wing junction flow

0° Incidence
Predicted skin friction lines on the floor for 0.0° incidence (Fig. 3.7) indicate the clear presence of a junction horseshoe vortex flow structure as observed in the non-lifting Rood aerofoil study. The modified LS(1)-0413 aerofoil profile is not symmetric and hence the circulation generated creates an angle to the onset flow. The saddle point is located at about 10 mm, i.e. 25% of wing aerofoil thickness from the wing leading edge. A close-up of the trailing edge region (Fig. 3.7b) shows that small separations are predicted on both wing surfaces as well as a separation caused by the finite width trailing edge. Flow around the trailing edge from the lower surface feeds into the separation on the upper surface. The flow around the trailing edge eventually reverses and is entrained into the downstream wake. Unlike with the Rood aerofoil, it is not possible to investigate flow patterns in a plane of symmetry (since this is absent) and hence flow patterns were studied in the stagnation plane. The stagnation plane is similar to a plane of symmetry for an uncambered aerofoil in that it is perpendicular to the floor and contains the saddle and nodal points of the junction flow. Flow vectors and pathlines in the stagnation plane at 0.0° incidence are shown in Fig. 3.8a. In this case the vortex core is much thinner than was seen in the non-lifting case. The circulation of the flow upstream of the leading edge again gives rise to the junction horseshoe vortex as clearly illustrated in Fig. 3.8b.

Fig. 3.7: Predicted flow patterns on floor for half wing, at 0° incidence
The significance of the separation line is limited to the flow on the floor, i.e. it does not provide any information about the flow just above the floor. Fig. 3.9 shows a three dimensional view, for the first 10 mm above the floor, of the junction flow pattern. Above the floor, the oncoming flow (shown for planes other than the stagnation plane) is deflected by the wing and moves towards the separation line on the floor; it is then diverted around the wing but only touches the floor at the separation line. On touching the floor the velocity drops to zero, represented by the flow lines terminating at the separation line. The separation line forms the locus of termination of these flow pathlines. On the floor, the flow deflected around the wing stays between the separation line and the wing, whereas just above the floor, the flow may move across the separation line.

Fig. 3.9: 3D view of oncoming flow deflected from leading edge, $0^\circ$ incidence
Flow patterns in the corner of the wing/floor junction, within a small area of 1 mm x 1 mm, show as expected the presence of a second vortex core located between the nodal point of attachment and the wing Fig. 3.10a. This is as before in the non-lifting case the secondary vortex which rotates in the opposite direction to the primary horseshoe vortex. This vortex travels downstream significantly closer to the wing (Fig. 3.10b). In the leading edge region high rotation is seen in the two vortices which decrease as they travel downstream (Fig. 3.11).

Fig. 3.10: Secondary vortex, 0.0° incidence

Fig. 3.11: Zoomed-in top view of two vortices, 0° incidence
Flow patterns on the wing are influenced and altered, although in a fairly limited manner at this incidence, by the flow structures in the junction area. The close proximity of the secondary vortex and its rotation direction causes the flow to move from the floor onto the wing. Other than the flow moving onto the wing and the small regions of flow separation near the trailing edge on both surfaces, no marked flow patterns are seen on the wing surface (Fig. 3.12).

![Fig. 3.12: Predicted flow patterns on half wing surface at 0.0° incidence](image)

The secondary vortex has about the same height on both suction and pressure surfaces (Fig. 3.13). On the suction surface, this vortex merges into the trailing edge separation (Fig. 3.14a). In contrast to this, the secondary vortex is not seen at the trailing edge on the pressure surface. Instead it is replaced by a separation zone which is closed before the trailing edge (Fig. 3.14b). This is due to the aerofoil surface profile resulting in a favourable pressure gradient towards the trailing edge on the pressure surface.
As they travel downstream the two vortices (Fig. 3.11) gradually induce the surrounding flow into a circular motion which becomes easily identifiable in the wake at an axial location of \( x/c = 2 \); this circulation was not readily visible at the upstream axial location of \( x/c = 1 \) (Fig. 3.15).
The predicted static pressure coefficient for various stations along the half wing and ideal half wing are shown in Fig. 3.16 to Fig. 3.18. At 75 mm the pressure coefficients for the two half wings are identical suggesting no influence of the junction flow at this height for 0° incidence. As the floor is approached the effects of the junction flow start to appear as seen at heights of 10 mm (Fig. 3.17) and 5 mm (Fig. 3.18). The predicted effects are quite small since no significant reduction in the enclosed area is predicted indicating no significant loss of lift in the junction and hence marginal reduction in half wing coefficient compared to the ideal half wing, as previously noted in Table 3.1. This is similar to the Bernstein & Hamid finding where no reduction in the junction lift coefficient was seen until a sectional lift coefficient of 0.6 was achieved at 6.0° incidence (the lift coefficient of the modified LS(1)-0413 at 0.0° incidence is ~0.4). Furthermore, small variations in the pressure magnitude along the span suggest marginal changes in drag.
Fig. 3.16: Cp distribution at height=75 mm (7.9 % Span), 0° incidence

Fig. 3.17: Cp distribution at height=10 mm (1.1 % Span), 0° incidence

Fig. 3.18: Cp distribution at height=5 mm (0.5 % Span), 0° incidence
The empirical correlation for the bluntness factor developed by Fleming\textsuperscript{17} was only derived from symmetrical aerofoil data which hinders the use of this correlation for a cambered aerofoil. However, an estimate was made for the bluntness factor for the modified LS(1)-0413. The values obtained were 0.025 and 0.024 based on distances on the suction and pressure surfaces, respectively. The value is even smaller than the bluntness factor for the NACA 0012 (Table 1.2). Flow patterns on the floor for NACA 0012 (Fig. 1.19) were not as marked as for thick wings hence the weak vortex seen at 0\textdegree incidence for the modified LS(1)-0413 is of little surprise.

10\textdegree incidence
For 10.0\textdegree incidence a significantly larger and thicker onset flow vortex circulation compared to 0\textdegree incidence is seen in the stagnation plane (Fig. 3.19a). In this case the junction flow horseshoe vortex is located further from the wing than was seen earlier at 0\textdegree incidence (Fig. 3.8a). Fig. 3.20 shows the formation and path of the secondary vortex. On the suction surface the vortex travels along the wing root for just 15 mm (x/c = 0.048) before it is displaced away from the wing towards the junction horseshoe vortex.

![Diagram](image)

\textbf{Fig. 3.19:} Junction horseshoe vortex, 10.0\textdegree incidence
Fig. 3.20: Secondary vortex, 10.0° incidence

Fig. 3.21 shows skin friction line flow patterns on the tunnel floor which now displays significantly different flow features compared to 0° incidence (Fig. 3.21b). The near leading edge picture is essentially the same as at 0° incidence except at 10° incidence the predicted horseshoe vortex has moved away from the suction surface. The general pattern shows large separation areas on the wing suction surface side with a significantly increased trailing edge separation region. The separation region between the trailing edge separation and the horseshoe/secondary vortex line on the floor is here termed a ‘mid–chord separation’.

Fig. 3.21: Predicted flow patterns on floor for half wing, 10.0° incidence
Flow patterns above the tunnel floor are shown up to a height of 20 mm (Fig. 3.22). The patterns are similar to those seen on the floor; however, as height increases the different separation zones change in size and gradually grow smaller as the visualised plane is moved in the traverse direction (compare 1 mm and 20 mm).

![Flow patterns above the floor, 10.0° incidence](image)

**Fig. 3.22:** Predicted flow patterns above the floor, 10.0° incidence

Flow patterns on the wing suction surface (Fig. 3.23) also show evidence of much larger separation regions compared to 0° incidence (Fig. 3.12). These regions are classed as trailing edge and mid-chord separation.
On the pressure surface, there is no separation over most of the surface because of the generally favourable pressure gradient (Fig. 3.24). At this incidence, the secondary vortex is seen close to the aerofoil surface up to the trailing edge.
The flow patterns on the wing surface are mirrored in the pressure coefficient trends shown in Fig. 3.25. Evidence of the large separation regions at heights of 5 mm and 75 mm are seen in the suction surface pressure plateaus. At 5 mm the pressure plateau extends from an axial location of about 30 mm (x/c = 0.095) up to the trailing edge. At 75 mm pressure coefficients show separation originating from an axial location of approximately 120 mm (x/c = 0.38). At 475 mm (midspan), just downstream of the leading edge, the pressure coefficients for the ideal half wing and half wing are identical showing that this span location is free from the junction effects. Note that at 5 and 10 mm the ‘Cp’ suction peak values are reduced compared with the ideal case.
Fig. 3.25: Cp distribution at height=5 mm (0.5 % span), 10.0° incidence

Fig. 3.26: Cp distribution at height=75 mm (7.9 % span), 10.0° incidence

Fig. 3.27: Cp distribution at height=475 mm (50 % span), 10.0° incidence
3.2 **Summary-3D Junction Flow Predictions for a Lifting Half Wing**

Predictions for an ideal wing and a junction flow affected half wing for a lifting aerofoil were made using the Spalart-Allmaras turbulence model in RANS CFD predictions at 0° and 10.0° incidence. As expected from literature, half wing results showed extra aerodynamic losses and were explained by the presence of a horseshoe vortex in the wing/floor junction. The vortex grew in size with increasing incidence. At 0° incidence the vortex was small and hence losses were nominal. The greatly increased size and strength of the vortex predicted at 10.0° incidence significantly increased the losses and made the half wing results unrepresentative of an ideal aspect ratio 6 wing. Hence, 10° incidence was chosen as the design case for suction investigations to be investigated next.

It has to be acknowledged that the presence of such a large separation flow in the 10° incidence case raises questions about the predictive accuracy of the current RANS predictions. Eddy viscosity turbulence model RANS CFD is not generally accepted as accurate for large scale separated and thus inevitably highly unsteady flows and more advanced techniques such as Large Eddy Simulations (LES) are considered more appropriate. The claim is supported by the LES predictions of Gand et al.\(^5\), although to a limited extent as they were made for a symmetric wing at a lower incidence (7°). However, time constraints did not allow the use of such advanced CFD in the present project. The current RANS approach performed reasonably well at lower incidence and it was decided to remain with this level of modelling for further CFD studies. It is argued that eliminating or weakening the horseshoe vortex would alleviate the junction losses and bring half wing performance closer to ideal half wing. Hence, the next step taken was to use the current CFD approach to model localised suction upstream of the wing leading edge.
4 CFD Predictions of Junction Flow Modification using Localised Suction

Bippes & Turk\textsuperscript{22} (1982) has previously suggested the use of localised suction just upstream of the wing leading edge to improve half wing wind tunnel test results and bring them closer to an ideal half-wing. Following the findings of Philips et al.\textsuperscript{7} and Barberis et al.\textsuperscript{6}, localised suction seemed to show promise and is further investigated as part of the present work. Prior to testing in the wind tunnel, the effectiveness of the technique was examined using the same CFD methodology applied in Chapters 2 and 3. In particular CFD was used to investigate elimination of the horseshoe vortex or reduction in separation that could be achieved via application of suction and the consequent effects on the aerodynamic coefficients. In the process the effect of design parameters such as suction hole location, size, shape and the required suction rate were also studied. A design of the suction setup to be tested in the wind tunnel was then selected.

4.1 Half Wing CFD Predictions with Suction

10.0\(^{0}\) incidence was chosen as the suction case to be studied since at this incidence the larger aerodynamic loses predicted in the previous chapter would be more amenable to correction from the inherent problem of floor boundary layer interaction with the half wing flow. All initial predictions were made with a circular shaped hole, which maintains its geometric shape for the incidence sweep. The hole was modelled with a fillet radius, included to reduce losses in the suction system. A circular shape would also minimise any vorticity shed from a sharp streamwise edge, as was observed by Philips et al.\textsuperscript{7} for a rectangular slot. The effect of suction was investigated by varying hole diameter and location for given suction rates. Suction rates are represented in terms of suction coefficient, Cq, (illustrated in Fig. 4.1) defined as the ratio of mass flow through the boundary layer to suction mass flow (Eq. (4.1)).

\[
Cq = \frac{\dot{m}_{bl}}{\dot{m}_{suction}} \tag{4.1}
\]
where, \( \dot{m}_{bl} \) and \( \dot{m}_{suction} \) are the mass flow in the boundary layer and mass flow removed through suction, respectively. This is illustrated for a circular suction hole with diameter ‘d’ and having a filleted radius at the inlet (Fig. 4.1).

In Fig. 4.1, ‘\( \delta \)’ and ‘\( \delta^* \)’ represents tunnel floor boundary layer thickness and displacement thickness, respectively, in an empty wind tunnel on a solid floor, i.e. without suction hole. Displacement thickness is estimated by integrating the boundary layer velocity profile\(^{54} \) as represented by Eq. (4.2).

\[
\delta^* = \int_0^\delta \left(1 - \frac{u}{V_\infty}\right) dy
\]  

(4.2)

Hence mass flow in the boundary layer over a width equal to the hole diameter is given as:

\[
\dot{m}_{bl} = (\rho_\infty)(V_\infty)(\delta - \delta^*)(d)
\]  

(4.3)

where, ‘\( \rho_\infty \)’ is the free-stream density, ‘\( V_\infty \)’ is the free-stream and ‘d’ is the suction hole diameter.

Initial hole locations were chosen based on the findings of Barberis et al.\(^{6} \) with respect to the wing flow and floor saddle point and involved low suction rates of
the order of 0.2Cq. Subsequently, the hole size, location, shape and suction rate were varied to study the predicted effect of these parameters.

The final setup had a hole diameter of 58 mm, hole diameter to aerofoil thickness (d/t) of 1.4, with its centre aligned with the wing chord and located 30 mm (x/t = -0.73) upstream of the leading edge. Hole diameter to aerofoil thickness ratio (d/t) of 1.4 was greater than the value of 1.1 employed in Philips et al.\textsuperscript{7} experiment. The ratio was also significantly greater than that employed by Barberis et al.\textsuperscript{6} (0.28). It was judged that a larger hole would help combat effects of varying saddle point location with incidence. The hole diameter of 58 mm was chosen so as to be able to use a standard pipe size in the suction system. The suction rate was varied from 0.2Cq to 5.0Cq and its effects on the aerodynamic coefficients and junction flow features studied. Among the suction rates tested, a rate of 4Cq was observed to modify the half wing aerodynamic loads and junction flow features such that these approached those of an ideal half wing. All the half wing results with suction presented below are thus for 4Cq. The aerodynamic performance of the suction system was also not influenced as the hole shape was modified to a square so only circular hole results are presented. A summary of the predicted aerodynamic coefficients with suction is shown in Table 4.1. Suction was predicted to return the lift, drag and pitching moment coefficients to close to their ideal values. The suction induced flow features are now discussed in more detail.

<table>
<thead>
<tr>
<th></th>
<th>CL</th>
<th>CD</th>
<th>CM</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ideal Half Wing</td>
<td>1.15</td>
<td>0.073</td>
<td>-0.0818</td>
</tr>
<tr>
<td>Half Wing</td>
<td>1.018</td>
<td>0.0868</td>
<td>-0.0861</td>
</tr>
<tr>
<td>Half Wing-With suction</td>
<td>1.119</td>
<td>0.0754</td>
<td>-0.0828</td>
</tr>
</tbody>
</table>

Table 4.1: Effect of suction on half wing aerodynamic coefficients, 10.0\textdegree incidence

### 4.2 Flow Conditions and Grid Description

All predictions were made at a Re = 0.89 x 10\textsuperscript{6}. The grid distribution in the spanwise direction was the same as used earlier in no-suction cases. The grid in the horizontal plane was modified to incorporate the suction hole with fillet radius. The modifications were limited to the region around the hole as shown in Fig. 4.2. Tetrahedral cells were used to connect the hexahedral grid cells on the floor with the hexahedral cells in the suction hole; the suction system pipe was extended 12 mm downward beneath the floor. These modifications in the grid were then transferred along the span as shown in Fig. 4.3a, which shows an x-z
plane at midspan location; the equivalent grid for the no-suction case is shown in Fig. 4.3b.

The insensitivity of the no-suction prediction presented earlier to the grid modifications seen in Fig. 4.3a was established before the grid was used for half wing suction predictions. This was done by comparing the 2D aerodynamic coefficients reported in Chapter 2 (using a grid similar to that shown in Fig. 4.3b) with 2D coefficient predictions obtained using the modified x-z plane grid, (Fig. 4.3a, but without applying suction). Only a small effect was seen with the modified grid for all coefficients (less than 0.5 %, Table 4.2) and this was considered insignificant.
<table>
<thead>
<tr>
<th>Description of 2D Grid</th>
<th>10.0° Incidence</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Cl</td>
</tr>
<tr>
<td>All hexahedral cells (Fig. 4.3b)</td>
<td>1.4578</td>
</tr>
<tr>
<td>Hybrid grid (Fig. 4.3a)</td>
<td>1.4568</td>
</tr>
</tbody>
</table>

Table 4.2: Effect of grid modification on aerodynamic coefficients

The boundary condition at the lower end of the suction outlet pipe was treated as a velocity outlet with a specified constant velocity profile across the cross section defined from the suction mass flow chosen. This was the reason for extending the mesh 12 mm below the floor, to allow adjustment of the flow at entry to the suction hole to the suction outlet boundary condition. Other than this, the same boundary conditions from the no-suction case were used.

### 4.2.1 Analysis of the Results

Using suction significantly modified the flow patterns around the wing as indicated by the elimination of the large separation regions on the floor (Fig. 4.4a). No nodal point of attachment or attachment line is shown in this figure. Two additional saddle points were created compared to the no-suction case, more clearly seen in Fig. 4.4b. These saddle points are linked to extra vortices generated (shown later in Fig. 4.7a). Compared to the no-suction case, on the floor the saddle point has moved away from the wing and the vortex core has moved away from the wing surface and is larger in size (no-suction case shown in Fig. 4.5); similarly the size of the secondary vortex core increases with suction (Fig. 4.6).

![a): Overall view around the aerofoil](image1)

![b): Zoomed-in view around hole](image2)

**Fig. 4.4:** Flow patterns on the floor with suction
The complex vortex system created by suction is illustrated in Fig. 4.7a. The junction horseshoe vortex has partially disappeared since only the pressure side leg of the vortex can be identified. The leg of the horseshoe vortex has also been moved further away from the wing as compared to the no-suction case (Fig. 4.7b). The significant effect of suction is seen as elimination of the large separation regions on the floor. The mid-chord separation region is completely eliminated and the trailing edge region has been significantly reduced. Elimination of the mid-chord separation helps keep the secondary vortex, on the suction surface, close to the wing root up to the trailing edge. The reduction in these large separation regions is believed to be the main reason that the
predicted half wing aerodynamic coefficients are now much closer to the ideal half wing results.

Flow patterns in a chordwise plane, which coincides with the centre of the suction hole, show elimination of the horseshoe vortex core from the near vicinity of the wing leading edge (Fig. 4.8). Here it may argued that a preferred hole location may be further out along the stagnation plane so that it may eliminate the origin of the horseshoe vortex. However, predictions made by moving the hole centre on the stagnation plane showed that hole displacement did not have a significance influence.
Flow patterns, for the current suction case, on the wing pressure and suction surface are shown in Fig. 4.9. Flow patterns on the suction surface are greatly affected by applying suction, which has eliminated the mid-chord separation and significantly reduced the trailing edge separation. Applying suction seems to have had little effect on the pressure surface patterns.
On the suction surface the height of the secondary vortex remains largely unaffected (Fig. 4.10a&b). In the no-suction case the secondary vortex was displaced away from the wing; in the suction case the secondary vortex travels up to the trailing edge where it merges with the significantly reduced trailing edge separation (Fig. 4.10d).
On the pressure surface, the height of the secondary vortex is increased at the leading edge (Fig. 4.11a&b). In the suction case the height of the secondary vortex increases along the chord whereas no significant increase was seen without suction (Fig. 4.11c&d).
These modifications in the junction flow patterns are reflected in the predicted levels of turbulent viscosity generated by the flow. Fig. 4.12 compares these for an axial location of $x/c = 1$. Significantly high viscosity values are seen in the suction surface junction area without suction and are greatly reduced by applying suction. Overall on the pressure side the values are significantly reduced although locally $\sim 100$ mm from the wing slightly higher values appear. This overall reduction when suction is applied is due to the much lower velocity gradients observed in the vicinity of the wing surface where the flow velocities are much more closely aligned to the surface (although not perfectly). In contrast, for no-suction, a large separation region (Fig. 4.10a) exists resulting in regions of high shear creating high turbulence and excess drag.
The flow patterns on the wing surfaces are mirrored in the spanwise pressure coefficient distributions. Pressure profiles are presented for three heights, i.e. 5 mm (Fig. 4.13), 75 mm (Fig. 4.14) and 475 mm (Fig. 4.15) from the floor. At 5 mm predicted suction surface pressure coefficients show elimination of the pressure plateau associated with the separation regions in the no-suction case. A reduction in suction peak (from -2.3 to -1.9) is also seen. Fig. 4.10a&b show that the leading edge does not interact with the secondary vortex for either suction or no-suction cases. Hence the reduction in suction peak is probably due to slow recovery from the negative streamwise velocity region caused just downstream of the hole. On the pressure surface, the maximum value of the pressure coefficient moved closer to a stagnation value of 1.0 indicating that the local free-stream speed was closer to the undisturbed tunnel free-stream speed.

Fig. 4.12: Turbulent viscosity contours at x/c=1

Fig. 4.13: Cp distribution comparison at height=5 mm (0.5 % span)
At a height of 75 mm the pressure coefficient profile again shows elimination of separation on the suction surface and also now an increase in the leading edge suction peak (from -2.3 to -2.9). Hence, the flow over the wing at this height is starting to resemble that at higher heights, i.e. closer to the ideal wing case. At a height of 475 mm (midspan) the pressures remain largely unaffected by applying suction but an increase (from 2.9 to 3.4) is seen in the magnitude of $C_{p_{\text{min}}}$ at the leading edge.

![Fig. 4.14: Cp distribution comparison at height=75 mm (7.9 % span)](image1)

![Fig. 4.15: Cp distribution comparison at height=475 mm (50% span)](image2)
4.3 Summary- CFD Predicted Effect of Suction on Junction Flow for a Lifting Half Wing

Predictions were made for the effects of localised suction on a lifting half wing junction flow at $10.0^\circ$ incidence. Localised suction was applied just upstream of the wing leading edge in an effort to eliminate junction region separation and modify the horseshoe vortex. Following a parametric study involving suction hole location, size, geometry and suction rate a combination was selected that produced the desired results, i.e. elimination of the large flow separations predicted in the half wing junction region. This in turn increased the lift coefficient and reduced the drag and pitching moment coefficient, bringing the half wing results significantly closer to those of the ideal half wing. On the basis of these CFD predicted trends in this and the preceding chapter a wing tunnel experiment was designed and carried out as is described in the following chapter.
5 Test Facilities and Experimental Setup

Experimental work was conducted in Loughborough University’s indraft wind tunnel whose basic design is fully described in Ref. 38. The wind tunnel draws air from the atmosphere through an inlet. The airstream is then straightened as it passes through screens. The flow (Fig. 5.1) passes through a contraction and enters the test section. Following the test section the flow exits through three diffusers and two 90.0° corners before it is discharged back into the atmosphere. After the second diffuser the flow goes through a section that changes the non-circular shape of the tunnel circuit into a circular shape before it passes through the fan which drives the flow.

![Model of Loughborough University indraft wind tunnel](image)

**Fig. 5.1: Model of Loughborough University indraft wind tunnel**

The tunnel has a working cross-section area of 1.9 m x 1.3 m (width x height) and a test section length of 3.6 m. Free-stream turbulence intensity values at the centre of the test section have been measured as 0.15%\(^\text{38}\). The maximum velocity achievable is 45 m/s, resulting in a Reynolds number per metre of 3.18 \times 10^{6}. The tunnel is generally used for half model testing of aeronautical models and also for automotive models. A right-handed coordinate system (as shown in Fig. 5.2) is used in the test section in which ‘x’ is in the free-stream direction, ‘y’ is along the wing span and ‘z’ is normal to the x-y plane. The centre of the
coordinate system coincides with the centre of the turntable. A photograph of the test section looking in the downstream direction is shown in Fig. 5.3.

Fig. 5.2: Test Section Reference Axis System

Fig. 5.3: View of test section looking into the downstream

5.1 Half Wing Design and Geometry
The design of the half wing was a compromise between several conflicting requirements. It was desirable to have as large a model as possible, producing high loads and thus increasing the accuracy of measurements, but the size was limited by the size of the test section and maximum load limits on the balance. Furthermore, it was desired for the aspect ratio be representative of real life
Testing Facilities and Experimental Setup

a aircraft, which generally range between 6 and 8\(^5.5\). The final design chosen had a span of 950 mm and an effective full span aspect ratio of 6.03. The gap between the wing tip and the tunnel wall was 27 % of tunnel height, i.e. greater than the minimum (10 % of tunnel height) recommended by Pope\(^5.6\) to minimise interaction of the flow between the wing and the tunnel wall. The aerofoil used for the wing, as cited above, was a modified version of a NACA LS (1)-0413. The original aerofoil has been used on the Glassair III\(^5.7\). The aerofoil has a trailing edge thickness to chord ratio of 0.0051, which resulted in a trailing edge thickness of 1.6 mm for the 315 mm chord length used; a trailing edge thickness of greater than 1 mm is desired for ease in manufacturing (Fig. 5.4). Sectional properties of the aerofoil are summarized in Appendix A.

The wing was manufactured from pre-set resin, machined to achieve the desired aerofoil shape, reinforced with steel rods as spars (Fig. 5.5). These rods extended outwards from the wing root chord and were secured into an interface block which was bolted on the balance through a mounting strut. The reinforcement rods were sized and positioned to have minimum bending and twisting of the wing under aerodynamic loads assuming simple beam calculations\(^5.8\). The location and size of the steel spars is shown in Fig. 5.4, wing engineering drawings are shown in Appendix B.
5.2 Half Wing Measurements

A sketch of the half wing mounted in the tunnel is shown in Fig. 5.5. Conducting balance measurements on half wings requires a minimum gap between the wing and tunnel floor to prevent fouling of loads measurements by the loads on the floor. Pope\textsuperscript{56} suggests that a small gap of the order of 0.5 \% of the span may be allowed which could be a few millimetres for a wing with a span of the order of 1 m.

![Fig. 5.5: Sketch of model setup for basic half wing tests](image)

On the other hand Kuppa & Marchman\textsuperscript{59} observed that a gap as small as 0.1 mm can still influence half wing measurements. Hence it was decided to conduct experiments without any gap. The absence of a gap prevents any meaningful balance data being collected and hence in the current study experimental work was limited to flow visualisation and wing surface pressure measurements; the balance was only used to mount the wing and vary the incidence angle. The arrangement employed to avoid any gap between the wing and the floor is sketched in Fig. 5.6. A plate was fixed to the wing root level with the tunnel floor. The gap, approximately 5 mm, between the tunnel floor and the plate was filled with a soft modeling compound commonly marketed as ‘Playdoh’. This was made flush with the tunnel floor so the aerodynamic effect of the filled gap was assumed to be minimised and the soft nature of the filler prevented any loads transfer from the floor onto the balance. The setup allowed a perfect seal between the wing and the floor. The plate was attached to the wing root and prevented any flow leaks from the pressure surface onto the
suction surface. The plate was sized large enough that the entire expected junction flow features would lie within the plate area. Thus the plate design was more extended on the suction surface than on the pressure surface.

Fig. 5.6: Sketch of half wing setup (without suction)

A photograph of the half wing installed in the wind tunnel indicating the wing root plate design is shown in Fig. 5.7
Wing surface pressures were measured via metallic tubes embedded into the wing surface and routed out through the interface block (Fig. 5.8). The tubes were sealed at the embedded end and open at the other which was connected to a pressure scanner through a ‘scanner disconnect’ interface (Fig. 5.9). Initially the pressure tappings were drilled 75 mm from the floor to record pressure measurements at a height expected to be outside of the floor boundary layer. These holes were then filled and re-drilled at 5 mm height to record measurements within the junction flow affected zone.
Wing surface pressures were measured using a Chell pressure scanner at a sampling rate of 312 Hz. The pressure scanner measured up to 64 channels with individual transducers. Signals from these transducers were multiplexed at speeds up to 20 KHz to a data acquisition system which enabled the data to be viewed in real time or logged on a PC. The scanner had an accuracy of ±0.12 mm of water, which is ±0.05 % of the full scale range.

5.3 Suction System

In the experiments the suction hole geometry and location were initially chosen to be the same as used in the CFD predictions, i.e. a suction hole with a diameter of 58 mm with centre located 61 mm upstream of the leading edge along the chord line. An existing fan was used for the suction system. Initial pressure loss estimations of the suction duct system showed that the capacity of the fan was such that it was sufficient to remove the required mass flows at tunnel operating speeds up to 20m/s.

5.3.1 Calibration of the Suction System

Prior to installation of the suction system in the wind tunnel, the mass flow through the suction hole exhaust duct work was calibrated with reference to a velocity measurement at the centre of the duct. The velocity was measured at Station ‘A’ (see Fig. 5.10) located 83 mm downstream of the bellmouth of the suction inlet. The duct’s centre line velocity was measured using a pitot probe.
and a ring of static pressure tappings at Station ‘A’ around the ring. Static pressure was taken to be the average of measured values (Fig. 5.10a). During calibration a wooden sheet (not shown in Fig. 5.10) containing a hole with the same geometry as selected for use in the final suction system was bolted onto the metallic plate. The size of the plate was 2 m x 2 m with a hole located in its centre and this was used to replicate the surroundings of the hole in the wind tunnel. Pressure measurements were made in 1 mm and 10 mm radial increments near and away from the duct sidewalls, respectively.

The local total pressure was measured using an electronic manometer (type Furness Controls FC 0332), whose accuracy was better than 0.5 % of reading. The position selected for the total pressure measurement was just downstream of the suction hole intake - the flow would not be fully developed over such a short length of duct. However, due to the limited space underneath the test section and the shape of the suction ductwork this was the best possible location. Fig. 5.10b shows all the suction system components: the intake, first and second diffusers, flexible duct lengths and the extraction fan. Diffusers were
incorporated to reduce the pressure losses in the system. Excess length in the first flexible duct was introduced which allowed free rotation of the turntable and the connected duct work when the system was installed in the wind tunnel. The second flexible duct allowed for placement of the fan at a convenient location.

The mass flow in the suction system was calculated by integrating the velocity profiles measured at Station ‘A’ along two orthogonal directions. Typical velocity profiles inside and outside the wind tunnel are shown in Fig. 5.11. Core suction velocity was maintained for most of the duct diameter but the velocity drops close to the duct walls. This supports the technique used to apply suction in the CFD predictions where a constant velocity profile was defined as the suction boundary condition.

Calibrating the system involved correlating the measured integrated mass flow against the measured velocity at the duct centre which could be measured during wind tunnel testing. The calibration process was repeated with the entire suction system installed in the wind tunnel, with the wing mounted and with the tunnel running at 20 m/s. Repeatability of setting the suction system at a specific mass flow was within 1.5 %. Results for the calibration outside and inside the wind tunnel are shown in Fig. 5.12. The variation of measured mass flow with core velocity was closely linear. Calibration done inside the wind tunnel showed similar results to those obtained in the outside calibration. The plot also shows an estimate of the mass flow in the approach flow boundary.
layer at a free-stream speed of 20 m/s (over the width of the suction hole diameter). It can be seen that the suction system was capable of removing up to 3 times the boundary layer mass flow.

![Graph showing the relationship between core speed and air flow](image)

**Fig. 5.12:** Calibration measurements outside and in the wind tunnel

### 5.3.2 Installation of Suction System in Wind Tunnel

An initial estimate of the load on the tunnel floor with the suction system installed indicated that the suction system could not be live on the balance, which would be the case if the suction system was connected directly to the wing root plate. Hence the plate attached to the wing root was modified from the design shown in Fig. 5.6a so that the suction hole could be installed as part of the tunnel floor. The redesigned plate is shown in Fig. 5.13 and a sketch of the suction system installed in the wind tunnel is shown in Fig. 5.14.
5 Testing Facilities and Experimental Setup

NOTE: Not to scale
Dimensions in millimeter

Tunnel Wall
Flow
Tunnel Centre Line
Tunnel Floor

a): Top view sketch of the modified plate

Fig. 5.13: Suction hole arrangement in Loughborough University wind tunnel
Fig. 5.14: Sketch of suction system installation

Photographs of the overall suction system installation in the wind tunnel are presented in Fig. 5.15, Fig. 5.15a shows a view of the mounting strut looking from the downstream direction. The pressure scanner, on one side, is plumbed into the tubes from the pressure tappings in the wing through the ‘scanner disconnect’ interface; on the other side it is connected to the data acquisition system which also supplies power to the scanner. An Ethernet connection from the acquisition system carries data to the tunnel PC. Also seen is the lower side of the plate fixed to the wing root. Fig. 5.15b&c show the first diffuser connected to the bottom surface of the tunnel floor. On the other end this is attached to a flexible duct which is fed through the balance room door to the second diffuser (Fig. 5.15d), which is secured on the floor in a wooden frame. Finally the second diffuser is connected directly to the suction fan.
Testing Facilities and Experimental Setup

5.4 Flow visualisation

Surface flow visualisation was conducted using an oil flow technique. This used a mixture of Titanium Dioxide, paraffin and linseed oil. Titanium dioxide is considered most suitable for flow visualisation on dark coloured models as it is a white and opaque pigment\(^{64}\). The ratio of mixture ingredients was varied for different studies with an aim to achieve the optimum ratio of air viscosity to mixture viscosity. The right mixture, obtained by trial and error, should be
viscous enough not to flow due to gravity on curved surfaces, yet flow easily due to the wind tunnel flow applied shear stress. Following a test run, the dried streaky deposits of the mixture provided the surface flow pattern created by the shear stress between the model and the air. It is known that the oil layer on the model surface does interfere with the flow, and the results are also affected by mixture movement under gravity or the generated pressure field, these effects are however considered small. The dried patterns require careful analysis to interpret the results. The mixture was usually applied with a brush over the surfaces of interest. Care was taken to align the brush strokes perpendicular to the flow direction so as not to confuse, following a run, any brush stroke with the dried flow pattern. The effects of gravity for vertically mounted models make the technique difficult to use below speeds of 45 m/s\textsuperscript{65}. In the current situation, where the free-stream was less than the recommended minimum speed, the photographs of the dried streaks were complemented by recording video images as the tunnel speed was built up from zero to the free-stream value.
6 Wind Tunnel Results and Comparison with CFD

This chapter describes surface flow features observed during experiments on wing and wind tunnel floor junction region flow interactions, both with and without localised suction. Experiments were carried out at chord Reynolds numbers of $0.89 \times 10^6$ and $0.44 \times 10^6$, although suction was only investigated at the lower Reynolds number, this was due to the limited capabilities of the suction system. The influence of suction on wing surface pressure measurements at two spanwise stations, 5 mm (0.5 % span) and 75 mm (7.9 % span) from the floor, are also discussed at the lower Reynolds number. Results from experiments, both with and without suction, are compared with CFD predictions. Most of the tests were carried out at $10.0^0$ since this incidence was chosen as the design case described in Chapter 4; unless specified both experimental and CFD results are for this incidence.

6.1 Half Wing Flow Patterns Without Suction, $Re=0.89 \times 10^6$

Fig. 6.1 shows an overall view of the junction flow viewed from upstream of the wing leading edge. The gap between the plate and tunnel floor filled with playdoh appears as straight white lines in the photograph. The existence of the junction horseshoe vortex is confirmed by the presence of a separation line around the wing and a saddle point in the leading edge region on the floor. The figure shows the location of a nodal attachment point in the leading edge region close to the pressure surface.

![Fig. 6.1: Overall view of junction flow features, $Re=0.89x10^6$](image)
Fig. 6.2 shows flow patterns on the wing suction surface and the wind tunnel floor. Heights of 5 mm and 75 mm, which correspond to the locations of pressure tappings, are marked on the surface to give an idea of the scale. The vertical line aft of the wing leading edge at higher spanwise positions indicates a laminar separation bubble and hence the approximate location of boundary layer transition. The line ceases to exist approximately 75 mm from the floor indicating fully turbulent flow on the wing below this height as a result of the turbulent floor boundary layer/wing interaction. The fact that there is no transition on the wing close to the junction justifies the decision not to include transition location modelling in the CFD predictions of the lifting half wing (Chapter 3).

A zoomed-in view of the flow features on the wing suction surface and floor is shown in Fig. 6.3. A secondary vortex is seen emerging from the leading edge; near the leading edge the height of the vortex is approximately 5 mm. On the wing surface the vortex expands as it moves towards the trailing edge. Mid-chord separation on both the wing and the floor originates at about 50 % chord and increases in size as it travels downstream. The mid-chord separation is blocked by the trailing edge separation, which is due to flow coming around the trailing edge from the pressure surface. The secondary vortex is diverted around the mid-chord separation on the wing surface and the floor. All these features were seen in the CFD predictions at 10° incidence but were significantly enhanced in scale compared to the experimental observations.
Fig. 6.3: Wing suction surface (zoomed-in views), Re=0.89x10^6

Fig. 6.4 shows the flow patterns on the wing pressure surface and the wind tunnel floor in the junction region. Boundary layer transition on the wing surface is seen close to the trailing edge, and again in the junction flow affected region near the floor no evidence of transition is seen on the wing surface. No significant separation regions are seen on the wing surface, although horseshoe vortex separation line and saddle point are seen on the floor.

Fig. 6.4: Wing pressure surface, Re=0.89x10^6
The saddle point is seen more clearly in Fig. 6.5, which shows a view of the flow features on the floor and the wing pressure surface. A small separated region is now visible on the floor near the trailing edge of the wing. A secondary vortex is seen near the leading edge within close proximity of the nodal point. A further zoomed-in view with saddle and nodal points marked is shown in Fig. 6.6.

Fig. 6.5: Wing pressure surface (zoomed-in views), Re=0.89x10^6

Fig. 6.6: Zoomed-in view of saddle and nodal point, Re=0.89x10^6
6.2 **Half wing flow features, Re=0.44 x 10⁶**

To check the effect of lowering the Reynolds number to the level at which pressure tapping data could be gathered, flow visualisation at Re = 0.44 x 10⁶ was carried out. An overall view of the junction flow viewed from upstream of the wing leading edge is shown in Fig. 6.7. All the flow features identified previously (Fig. 6.1), i.e. saddle point, nodal point and separation line, may be observed as essentially unchanged at this lower Reynolds number. A close-up view of the main flow features on the suction surface is shown in Fig. 6.8. These are the secondary vortex, the mid-chord separation and the trailing edge separation. Their locations and behaviour are comparable to those seen at the higher Reynolds number (Fig. 6.3). The unchanged nature of these flow features at the lower Reynolds number, gave confidence that the effects of localised suction could be investigated at the lower Reynolds number which formed the upper limit of the operational capability of the suction system.

![Image of flow features](image_url)

**Fig. 6.7: Overall view of junction flow features, Re=0.44 x 10⁶**
The experimental pressure coefficients at the two tapping locations are shown in Fig. 6.9. These measurements are compared with 2D pressure distributions available from Ref. 31, although the 2D measurements were carried out at the higher Re = 2.0 x 10⁶ and with a transition strip on the wing. On an ideal half wing the pressure distribution at the reflection plane should be comparable although not identical to 2D values. Even allowing for small Reynolds number effects, the measured pressure distributions of half-wing in the junction region show large deviations from the 2D value. Pressures on the suction surface are more severely affected by the junction flow. At a height of 75 mm, which is outside the junction flow region, the leading edge peak drops to about -2.0. The Cp distribution on the pressure surface shows a better match. Bernstein & Hamid⁴ and Applin¹¹ have also observed that pressure measurements are affected by the junction flow at heights significantly greater than the onset flow floor boundary layer thickness. Hence, it is not surprising that the pressure distributions at 75 mm diverge from the 2D measurements.

The mid-chord and trailing edge separations seen in the junction region during flow visualisation (Fig. 6.13) are not apparent as any pressure plateau in the Cp curves at 5 mm. Bippes²¹ also observed in his experiments on 3D junction flows that small separated regions seen in the visualisations may not necessarily be represented by the classic 2D representation of a constant pressure plateau, perhaps due to the low velocity values.

Fig. 6.8: Wing suction surface (zoomed-in views), Re=0.44 x 10⁶
Well inside the floor boundary layer, i.e. at 5mm, generally the pressure coefficients are even further from the 2D curves, particularly on the pressure surface, where loss of total pressure prevents the stagnation value of ‘1.0’ being seen.

![Graph of Cp distribution](image)

**Fig. 6.9:** Measured Cp distribution, Re=0.44 x 10^6

### 6.3 CFD Predictions at lower Re=0.44 x 10^6

As in the wind tunnel measurements, CFD predictions of the half-wing at 10^0 incidence were also repeated for the lower Reynolds number to compare with the results at Re = 0.89 x 10^6 presented in Chapter 3. The predicted flow patterns on the floor and wing surfaces at a Re = 0.44 x 10^6 are shown in Fig. 6.10 and Fig. 6.11, respectively. These flow patterns have all the flow features, with similar dimensions, predicted at the higher Reynolds number (Fig. 3.21 to Fig. 3.24).

![Predicted flow patterns](image)

**Fig. 6.10:** Predicted flow patterns on floor, Re=0.44 x 10^6
6.4 **Comparison of Experiment and CFD, No-Suction, Re=0.44x10^6**

The existence of the junction horseshoe vortex predicted by CFD has been confirmed in the flow visualisation experiments but the wing flow changes caused by this in the wing/floor junction region have shown significant differences. A comparison of predicted and measured separation line locations is shown in Fig. 6.12. On the pressure side of the wing there is good agreement between experiment and prediction. The saddle point from predictions was also close to the measured position. On the suction surface, however, the comparison deteriorates significantly with the predicted separation line much further from the wing. This is due to the large separation regions predicted on the suction surface (Fig. 6.11a) which are much larger than seen in the experiment (Fig. 6.13).

![Fig. 6.11: Predicted flow patterns on wing surface, Re=0.44 x 10^6](image)

![Fig. 6.12: Separation line location on floor, Re=0.44 x 10^6](image)
Fig. 6.13: Wing suction surface, Re=0.44 x 10^6
On the pressure surface the predicted flow patterns (Fig. 6.11b) are comparable to the experimental flow visualisation (Fig. 6.14). However, other than the secondary vortex no noticeable flow features are present on this surface.

Fig. 6.14: Wing pressure surface, Re=0.44 x 10^6
Differences in the predicted and experimental flow features are the cause of differences in predicted and measured wing surface pressure measurements at 5 mm (0.5 % span) and 75 mm (7.9 % span); these are shown in Fig. 6.15 and
Fig. 6.16, respectively. At both heights, predicted and experimental pressure distributions on the pressure surface are comparable both in trend and absolute values. Note in particular that predicted and measured pressure surface show good agreement in the $C_p$ peak value, which is much less than 1.0 at the lower (5 mm) height but reaches very close to 1.0 at the 75 mm height showing that the latter is outside the boundary layer loss region. On the suction surface, at 5 mm, the trends are fundamentally different as CFD predicts large scale separation originating at 10 % chord and extending to the trailing edge, whilst the experiment shows no evidence of such large separation. At 75 mm, the predicted pressure distribution shows separation from about 40 % of chord, whilst again experiment shows no evidence of this separation.

**Fig. 6.15: $C_p$ distribution at 5 mm, Re=0.44 x 10$^6$**

**Fig. 6.16: $C_p$ distribution at 75 mm, Re=0.44 x 10$^6$**
In an attempt to explore the onset of large scale separation in the experiments further an additional test was carried at increasing incidence with the pressure curves showed evidence of what was predicted in CFD. Fig. 6.17 shows the measured pressure distribution at a height of 5 mm and at 22° incidence. These measurements were made at an even lower Re = 0.22 x 10^6 to avoid undue loading of the balance resulting from vibrations when the wing was close to stall. At this incidence the measured pressure distribution is similar to the profile predicted by CFD at 10° incidence since both indicated (compare Fig. 6.17 and predicted suction side Cp in Fig. 6.15) large regions of pressure plateau on the suction surface. It seems that the predicted flow behaviour with the current CFD methodology may be reasonable, but it is predicted to occur at much too lower incidence. Again, this supports the findings of Apsley & Leschziner that demonstrated a better qualitative comparison as compared to a quantitative comparison.

![Cp distribution at 5 mm, 22.0° incidence, Re=0.22 x 10^6](image)

At 10.0° incidence, comparison of the Spalart-Allmaras predictions with experiment was not good but it was decided to continue with the turbulence model as is explained earlier in Chapter 3 (Section 3.2).

### 6.5 CFD Predictions with suction at Re=0.44 x 10^6

Initially the effect of suction at a Re = 0.44 x 10^6 was investigated using the same suction rate employed as at Re = 0.89 x 10^6, i.e. 4Cq. Further
investigations showed that a suction rate of $3.3C_q$ was in fact more effective in eliminating the junction region flow separation. Hence, all predicted results presented here are for the value of $3.3C_q$. The effect of applying suction on the flow features is similar to that seen earlier at the higher Reynolds number (Fig. 4.4 and Fig. 4.9). This was the elimination of the large separation region visible in the floor pathlines on the suction side and on the wing suction surface itself, with no marked effect on the pressure side both on the wing and floor. Since this has been discussed in detail in Chapter 4, the predicted flow patterns with suction for a $Re = 0.44 \times 10^6$ are shown here for reference and to allow comparison with experiment (Fig. 6.18 to Fig. 6.21).

![predicted flow patterns](image)

**Fig. 6.18:** Predicted flow patterns on floor with suction, $Re=0.44 \times 10^6$

![predicted flow patterns](image)

**Fig. 6.19:** Predicted flow patterns on wing surface with suction, $Re=0.44 \times 10^6$
The effects of suction on the junction flow were studied by pressure measurements on the wing surfaces and surface flow visualisation. Pressure measurements were carried out for a range of suction rates whilst flow visualisation was carried out at a suction rate of 2.9Cq. This rate was chosen from preliminary flow visualisation using wool tufts to provide a quick and simple method to identify flow separation. Wool tufts were taped onto the floor and the root region of the wing suction surface. The behaviour of the tufts was then monitored to identify how separated regions changed with increasing suction.
rates. Once the suction rate was beyond 2.9Cq the tufts indicated no further improvement in the junction flow and hence this rate was selected for surface flow visualisation. Fig. 6.22 shows the effects of suction in an overall view of the junction flow from a location upstream of the wing leading edge. This figure can be compared with the no-suction case in Fig. 6.7. A separation line exists on the pressure surface side, but on the suction side the separation line is limited only to a region close to the suction hole.

Two saddle points are seen on either side of the suction hole. These saddle points are dividing points between flow being sucked into the hole and flow following the free-stream. This is indicated by the arrows ‘local flow directions around the hole’ in Fig. 6.23. The saddle point on the suction side lies 45 mm upstream of the leading edge and approximately 57 mm offset from the chord line. The saddle point on the pressure side is offset approximately 98 mm from the chord line but is closer to the wing leading edge than on the suction side.
Flow patterns on the wing suction surface are shown in Fig. 6.24 with a zoomed-in view in Fig. 6.25. These figures can be compared with the no-suction case shown in Fig. 6.13 and Fig. 6.28.

Applying suction reduces the mid-chord and trailing edge separation regions. The height of the separation region on the wing surface is reduced to approximately 15 mm (originally 20 mm) and on the floor surface the width reduces to approximately 22 mm (originally measured as approximately 25 mm). The chord-wise extent of the separation also reduced from 145 mm to 90
mm. The secondary vortex remains largely unaffected by suction. Hence, the net result of suction may be summarised as a reduction of the mid-chord separated region.

Fig. 6.25: Wing suction surface with suction (zoomed-in views), Re=0.44 x 10^6

Fig. 6.26: Wing suction surface (zoomed-in views), Re=0.44 x 10^6
Flow patterns on the wing pressure surface are shown in Fig. 6.27 with a zoomed-in view in Fig. 6.28. Generally the flow patterns are the same as without suction, mainly because there were no prominent flow features on the pressure surface even without suction (Fig. 6.14). A small region of trailing edge separation on the floor seen in the no-suction case (Fig. 6.29) is largely eliminated by applying suction (Fig. 6.28). The main difference is that applying suction moves the saddle point and the separation line away from the wing.

Fig. 6.27: Wing pressure surface with suction, Re=0.44 x 10^6

Fig. 6.28: Wing pressure surface (zoomed-in views) with suction, Re=0.44 x 10^6
Wind Tunnel Results and Comparison with CFD

Fig. 6.29: Wing pressure surface (zoomed-in views), no-suction case, Re=0.44 \times 10^6

The location of separation lines with and without suction is shown in Fig. 6.30 and was obtained by superposition of Fig. 6.14 and Fig. 6.27. With suction the separation lines and the saddle point move away from the wing by approximately 40 mm, however on the suction side the horseshoe vortex was largely eliminated.

Fig. 6.30: Wing suction surface, Re=0.44 \times 10^6

The effect of suction on the pressure distribution at a height of 75 mm is shown in Fig. 6.31 for a range of suction rates. The effect of suction predicted by CFD, i.e. an increase in the suction peak (see Fig. 6.21) is not observed in these measurements. When suction is introduced the magnitude of the leading edge
peak reduces slightly along with pressures downstream of the leading edge up to about mid-chord. Increasing the suction rate resulted in a further decrease of the pressure magnitude. On the pressure surface applying suction moves the pressure slightly in the direction observed in the CFD, i.e. increased pressure values.

Fig. 6.31: Cp distribution at 75 mm, Re=0.44 x 10^6

The trends due to suction measured at the lower height of 5 mm are similar (Fig. 6.32) to those seen at 75 mm. On the suction surface the pressure peak deteriorates on application of suction. However, on the pressure surface applying increased levels of suction does tend to move the Cp values closer to the stagnation value of 1.0, indicating the suction system has been partially successful in removing the boundary layer effects near the leading edge on the pressure side.

Fig. 6.32: Cp distribution at 5 mm, Re=0.44 x 10^6
The effect of suction at higher incidence remained the same. The 5 mm height at 22° incidence is shown as an example in Fig. 6.33. Applying suction does not increase the peak negative \( \text{C}_p \) value neither does it eliminate the effect of the large scale separation leading to the expected pressure plateau. Again, an improvement in the pressure coefficients is seen on the pressure surface.

![Fig. 6.33: Cp distribution at 5 mm (22.0° incidence), Re= 0.44 x 10^6](image)

### 6.7 Effect of Suction Hole Location

As observed in the experiments, suction affected the pressure distribution on the wing surface but not as expected from CFD predictions. One possible cause for this may have been the close proximity of the suction hole to the wing leading edge. This may have produced a slow recovery of the flow downstream of the suction hole which could not then remain attached to the suction surface at this incidence. To investigate this, the suction hole was relocated further upstream of the wing leading edge by moving it approximately one hole diameter (60 mm) along the chord-line. Investigations of this configuration were limited to pressure measurements. The effect of suction at a height of 5 mm for the relocated hole is shown in Fig. 6.34. The effect of suction is the same as seen with the original hole location (Fig. 6.32), i.e. a reduction in peak suction values with increasing suction rate. However, the reduction observed with the relocated hole was smaller than that observed for the original hole. In contrast to this the beneficial effect of suction seen on the pressure surface is much weaker with the new hole location.
A comparison between the two suction hole locations is shown in Fig. 6.35 for the 5 mm pressure tapping location and a suction rate of 2.9Cq. The trend observed on the suction side of the wing is similar for both hole locations, i.e. a reduction in the magnitude of peak, but the reduction is significantly less for the relocated hole. On the pressure side, the original hole location is the most beneficial in terms of allowing Cp values near the leading edge to approach free-stream stagnation levels (Cp = 1).
6.8 **Comparison of Experiment and CFD with suction, Re=0.44 x 10⁶**

In experiments it is clear that the effect of suction is significantly more limited than predicted by CFD. This is primarily because CFD predictions without suction show significant separated regions which were not present in the experiments. In CFD eliminating these separated regions through the addition of suction resulted in significant improvements to the predicted pressure distributions. In the experiments the ‘baseline’ no-suction flow already showed only small effects of the junction flow separations thus applying suction had little to influence in terms of flow patterns on the wing. Therefore the effects of suction were even seen as detrimental. Comparison of the predicted pressure distribution, for 3.3Cq, at 5 mm and 75 mm is shown in Fig. 6.36 and Fig. 6.37, respectively. Similar trends are seen at both the heights with the predictions producing increased magnitude of Cp and hence greater enclosed area, i.e. increased lift, than was evident from experiments.

![Fig. 6.36: Cp distribution at 5 mm with suction at 3.3Cq, Re=0.44 x 10⁶](image_url)
However, the predicted effects of suction do show some similarities with the experiment in terms of flow features on the floor. CFD predicted that applying suction moves the separation line away from the wing and also creates a second saddle point upstream of the wing leading edge on the suction surface (Fig. 6.38).

Predicted locations of the saddle points may be observed in Fig. 6.39 and are comparable with the locations observed in the experiment (Fig. 6.23). The predicted location of the suction side saddle point is approximately 40 mm...
upstream of the wing leading edge compared to 45 mm seen in the experiment. The saddle point is predicted to be located 44 mm from the wing chord-line as compared to 57 mm from the experiment. The predicted location of the saddle point on the pressure surface side is approximately 4 mm upstream of the wing leading edge compared to 13 mm seen in experiment. The saddle point is located 98 mm from the chord-line compared to a predicted value of 92 mm.

Fig. 6.39: Predicted saddle point location with suction, $Re=0.44 \times 10^6$

6.9 Summary
Experiments at $10^0$ incidence showed limited separation regions in the junction area of the half wing. Applying suction marginally affected the separation region but generally worsened the pressure measurements. Experiments to examine the influence of suction hole location showed some sensitivity that needs further investigation. Applying suction at the original hole location reduced the magnitude of the pressure coefficients on the wing suction surface but showed improvement in the pressures values on the pressure surface. Relocating the hole reduced the disadvantage on the suction surface pressures but also eliminated the beneficial effects seen with the original hole on the pressure surface. CFD greatly over-predicted junction region separation on the suction side in the no-suction flow and hence applying suction was observed to be effective in improving junction flow patterns. The use of an eddy viscosity RANS turbulence model may have contributed to the poor predictions of the no-suction case and hence overestimated the influence of suction.
7  Summary, Conclusions and Suggestions for Future Work

In this study the benefit of increased Reynolds number was not achieved but the following conclusions are drawn to help understand the benefit or otherwise of utilising leading edge suction in half model testing.

7.1  Summary and Conclusions

Wind tunnel tests on a lifting half wing confirmed the existence of aerodynamic losses in the wing/floor junction region. These losses were caused by regions of flow separation on the wing surfaces in the root region, with the suction surface separations larger than on the pressure surface.

Performance of the Spalart-Allmaras turbulence model, at lower incidence, was satisfactory, as it was able to capture basic flow features of a junction flow, although the comparison with experiment was not perfect. This was in line with the performance of other eddy viscosity models as reported by Ölçmen & Simpson\textsuperscript{43}. The performance of the turbulence model significantly degraded as incidence was increased. At $10^0$ incidence the extent of the separated flow and associated losses seen in the experiments were, however, significantly less than those predicted by CFD. In essence, the predicted large scale flow separations in the junction region at $10^0$ incidence were not observed in measurements until much higher ($\sim22^0$) incidence. Hence the computational study concluded that Spalart-Allmaras turbulence model is not suitable and hence may not be employed for junction flow study at high incidence.

In the experiments, localised suction modified the position of the pressure leg of the horseshoe vortex whereas the suction leg was largely removed. However, the influence of localised suction (as currently introduced) on the flow features in the immediate vicinity of the junction was seen to be generally small, even showing a detrimental rather than a positive effect. Localised suction was shown to influence the pressure distribution on the wing in the junction vicinity. On the pressure surface, suction resulted in pressures near the leading edge moving closer to free-stream stagnation values.
Applying suction through a relocated hole position further away from the wing leading edge did modify the wing pressure distributions. This shows that suction hole location is a parameter that needs to be optimised since the single alternate investigated so far, somewhat improved suction side flow but reduced the pressure side improvement.

The study was conducted to investigate if localised suction can be employed as a standard practice for half model testing in any wind tunnel. It was concluded that localised suction as employed in this study, i.e. a single hole located just upstream of the wing leading edge, have rather detrimental effects on half wing testing, seen as a reduction in leading edge suction peaks, and hence may not be used.

7.2 Suggestions for Future Work

In terms of future work, both experimental and computational investigation should be taken further:

**Experimental Work:**

i. Measurements of the velocity and turbulence field in the junction flow, both with and without suction, would be invaluable in implying understanding of the horseshoe vortex flow interaction with a lifting wing flow, and also flow induced by a suction hole.

ii. The above velocity field measurements are also necessary to provide validation data for CFD modelling to improve on the methodology adopted in the present thesis.

iii. The significance of an optimum location for suction, applied over an area which make its effective use on both suction and pressure surfaces and over a wide range of incidences, needs to be investigated. Results from relocating the hole showed that a suction setup, possibly with individual controls for suction and pressure surface, may be achieved that may bring the junction flow pressure distribution closer to the ideal values. If the improvement hence seen is short of ideal values it needs to be evaluated if the new uncertainties outweigh the existing uncertainties.
iv. Pressure measurements provide valuable but limited information of flow state on a model. Balance measurements are acknowledged to be difficult in half model with suction but in order for the technique to go forward the issue needs to be resolved.

Computational Work:

i. A turbulence modelling approach that is capable of capturing the baseline, no-suction, junction region flow patterns at large incidences ($>10^0$) is needed. This may come from more advanced RANS statistical models than used in the present thesis (e.g. Reynolds Stress Transport Modelling) but if large scale separation is involved, with its inevitable unsteady characteristics than the more sophisticated Large Eddy Simulations (LES) approach may be necessary.
8 References


References


33 Fluent-Commercial CFD software package based on finite Volume Method. Fluent Inc., Centerra Resource Park, 10 Cavendish court, Lebanon, NH 03766, USA.


60 ESDU 66027, (March 2007) “Friction losses for fully-developed flow in straight pipes”, ESDU.

61 ESDU 83037, (January 2007) “Pressure losses in curved ducts: single bends”, ESDU.


9 Appendix A - Modified LS(1)-0413 Properties

Characteristics of modified LS(1)-0413 at a Reynolds number of $1 \times 10^6$ and Mach number 0.1 are listed as follows:

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Zero lift angle of attack</td>
<td>-3.84 degrees</td>
</tr>
<tr>
<td>Stall angle of attack</td>
<td>12.4 degrees</td>
</tr>
<tr>
<td>Location of aerodynamic centre</td>
<td>25.5 % of chord</td>
</tr>
<tr>
<td>Pitching moment about aerodynamic centre</td>
<td>-0.09</td>
</tr>
<tr>
<td>Trailing edge thickness to chord ratio</td>
<td>0.0051</td>
</tr>
<tr>
<td>Thickness to chord ratio</td>
<td>0.13</td>
</tr>
</tbody>
</table>

Fig.A.1: Profile for LS(1)-0413 Aerofoil
Appendix B – Engineering Drawings

10 Appendix B - Engineering Drawings
Appendix B – Engineering Drawings

Airfoil Section would be manufactured in-house at AAE dept.

Spars would be glued to wooden wing

Spars would be glued (loctite) and screwed to aluminium block

Loughborough University