AERODYNAMIC PERFORMANCE
OF LOW FORM FACTOR SPOILERS

A thesis submitted to the University of Manchester for the degree of
Doctor of Philosophy
in the Faculty of Engineering and Physical Sciences

2010

Christopher Donald Harley

School of Mechanical, Aerospace and Civil Engineering
# LIST OF CONTENTS

| LIST OF CONTENTS | ........................................................................................................ | 2 |
| LIST OF FIGURES | ........................................................................................................ | 4 |
| LIST OF TABLES | ........................................................................................................ | 8 |
| NOMENCLATURE | ........................................................................................................ | 9 |
| ABSTRACT | ........................................................................................................ | 13 |
| DECLARATION | ........................................................................................................ | 14 |
| COPYRIGHT STATEMENT | ........................................................................................................ | 14 |
| ACKNOWLEDGEMENTS | ........................................................................................................ | 16 |
| 1 INTRODUCTION | ........................................................................................................ | 17 |
| 1.1 LOW FORM FACTOR SPOILERS FOR FLIGHT CONTROL | ........................................................................................................ | 17 |
| 1.2 AIMS AND OBJECTIVES | ........................................................................................................ | 22 |
| 2 THEORY | ........................................................................................................ | 23 |
| 2.1 QUALITATIVE MODEL FOR THE EFFECT OF CAMBER | ........................................................................................................ | 23 |
| 2.2 COMPARISON OF FLOW TOPOLOGY MODELS FOR GEOMETRIC AND FLUIDIC SPOILERS | ........................................................................................................ | 24 |
| 2.3 GEOMETRIC SPOILER FLOWFIELD AND GLOBAL AERODYNAMIC COEFFICIENTS | ........................................................................................................ | 25 |
| 2.4 SPOILER PRESSURE DISTRIBUTION MODEL | ........................................................................................................ | 27 |
| 3 LITERATURE REVIEW | ........................................................................................................ | 29 |
| 3.1 GEOMETRIC SPOILERS FOR AIRCRAFT FLIGHT CONTROL | ........................................................................................................ | 29 |
| 3.1.1 DEVELOPMENT OF SPOILERS FOR AIRCRAFT CONTROL | ........................................................................................................ | 29 |
| 3.1.2 EFFECTIVENESS OF SPOILERS APPLIED TO SWEPT WINGS | ........................................................................................................ | 38 |
| 3.1.3 USE OF SPOILERS ON MODERN CIVIL TRANSPORT AIRCRAFT | ........................................................................................................ | 39 |
| 3.1.4 APPLICATION OF SPOILER TYPE DEVICES TO TAILLESS AIRCRAFT | ........................................................................................................ | 41 |
| 3.2 FLUIDIC SPOILER CONCEPTS FOR AIRCRAFT FLIGHT CONTROL | ........................................................................................................ | 45 |
| 3.2.1 INTRODUCTION | ........................................................................................................ | 45 |
| 3.2.2 ROLL CONTROL | ........................................................................................................ | 46 |
| 3.2.3 YAW CONTROL | ........................................................................................................ | 48 |
| 3.3 SUMMARY OF LITERATURE REVIEW AND CONCLUDING REMARKS | ........................................................................................................ | 50 |
| 4 EXPERIMENTAL METHODS | ........................................................................................................ | 52 |
| 4.1 EXPERIMENTAL APPARATUS | ........................................................................................................ | 52 |
| 4.1.1 PROJECT WIND TUNNEL | ........................................................................................................ | 52 |
| 4.1.2 WIND TUNNEL MODEL CONFIGURATIONS AND MANUFACTURE | ........................................................................................................ | 53 |
| 4.1.3 MiGS DESIGN AND MANUFACTURE | ........................................................................................................ | 55 |
| 4.1.4 CFFS DESIGN AND MANUFACTURE | ........................................................................................................ | 56 |
| 4.1.5 EXPERIMENTAL SET UP | ........................................................................................................ | 59 |
| 4.1.6 WAKE SURVEY | ........................................................................................................ | 60 |
| 4.1.7 DIRECT FORCE AND MOMENT MEASUREMENTS | ........................................................................................................ | 61 |
| 4.1.8 PRESSURE MEASUREMENTS | ........................................................................................................ | 62 |
| 4.1.9 DATA ACQUISITION SYSTEM | ........................................................................................................ | 63 |
| 4.2 EXPERIMENTAL PROCEDURE | ........................................................................................................ | 63 |
| 4.2.1 PHASE 1 – PRELIMINARY AND MODEL BASELINE CONFIGURATION TESTS | ........................................................................................................ | 63 |
| 4.2.2 PHASE 2 & 3 – BASELINE AND BULK TESTING | ........................................................................................................ | 64 |
| 4.2.3 WIND TUNNEL TEST PROCEDURE | ........................................................................................................ | 64 |
| 4.3 DATA REDUCTION | ........................................................................................................ | 65 |
| 4.3.1 BOUNDARY CORRECTIONS | ........................................................................................................ | 65 |
| 4.3.2 AEROFOIL SURFACE PRESSURE DISTRIBUTION CALCULATION | ........................................................................................................ | 66 |
4.3.3 Force and Moment Transfer ................................................................. 67
4.3.4 Aerodynamic Coefficient Calculated from the Force/Torque Balance ................................................................. 68
4.3.5 Lift Coefficient Calculated from the Surface Pressure Distribution .................................................................................................................. 68
4.3.6 Profile Drag Coefficient Calculated from the Wake Survey ...... 69
4.3.7 CFFS Jet Blowing Coefficient Calculation ........................................ 69
4.3.8 Aerofoil Profile Drag Calculation ......................................................... 70
4.3.9 Control Volume Analysis for CFFS Wind Tunnel Model Configuration .......................................................................................................................... 72
4.4 Measurement Validation Methods .............................................................. 74
4.4.1 CFFS End Effects Investigation ............................................................ 74
4.4.2 Sampling Period Investigation ............................................................... 75
4.4.3 Forced Boundary Layer Transition ....................................................... 76
4.4.4 CFFS Plenum Validation ...................................................................... 77
4.5 Measurement Uncertainty ...................................................................... 78
4.5.1 An Overview of Measurement Uncertainty ......................................... 78
4.5.2 Uncertainty in Lift Coefficient Measurement ...................................... 80
5 Computational Methods ........................................................................... 82
5.1 Computation Apparatus and Case Set-Up .............................................. 82
5.1.1 Numerical Solver .............................................................................. 82
5.1.2 Convergence Criteria ....................................................................... 82
5.1.3 Solution Monitoring and Convergence .............................................. 83
5.1.4 Turbulence Model .......................................................................... 83
5.1.5 Calculation of the Lift and Drag Coefficient ..................................... 84
5.1.6 Mesh Definition ............................................................................... 84
5.1.7 Mesh Independence Study ................................................................. 85
5.2 Computational Method Validation ......................................................... 86
5.2.1 Section Overview ........................................................................... 86
5.2.2 Geometric Spoiler Case .................................................................. 87
5.2.3 Normal Blowing Fluidic Spoiler Case ................................................ 89
6 Results & Discussion ................................................................................ 93
6.1 Comparison of Baseline Results ............................................................. 93
6.2 Micro Geometric Spoiler Results .......................................................... 94
6.3 Counter-Flow Fluidic Spoiler Results ..................................................... 100
6.4 Comparison of MiGS and CFFS ............................................................. 104
6.5 Upper and Lower Surface CFFS Blowing .............................................. 107
7 Conclusions & Future Research Opportunities .......................................... 110
7.1 Conclusions ......................................................................................... 110
7.2 Future Research Opportunities .............................................................. 114
8 References ............................................................................................... 119
9 Appendix .................................................................................................. 123
9.1 Change in Lift Due to the Deflection of a Plain Flap ............................. 123

Final word count: 25,874
LIST OF FIGURES

FIGURE 1.1: Photo of an Airbus A380 during landing phase with spoilers and high lift devices fully deployed. Taken from [6] .................................................. 18

FIGURE 1.2: Comparison of Macro geometric (MaGS), Micro geometric (MiGS) and counter-flow fluidic spoilers (CFFS). ................................................................. 19

FIGURE 2.1: Qualitative illustration of the effect of aerofoil camber on lift drag and pitching moment .......................................................... 24

FIGURE 2.2: Steady time-averaged two-dimensional streamline topology for three spoiler devices, A) geometric spoiler and B) i) normal blowing fluidic spoiler, and B) ii) counter-flow fluidic spoiler. ........................................... 25

FIGURE 2.3: Typical flowfield due to a rearward mounted macro geometric spoiler .......................................................... 26

FIGURE 2.4: Typical rearward mounted (apart from where stated) spoiler effect on lift, drag, pitching moment and change in surface pressure with deflection. A) and B) adapted from ESDU data sheet [2]. C) adapted from McLachlan et al [16] .................................................. 27

FIGURE 2.5: Effect of increasing spoiler deflection/height on the wing section surface pressure distribution ............................................... 28

FIGURE 3.1: One of the first applications of a forward mounted and forward hinged spoiler to aircraft wings taken from Weick and Wenzinger [21] .................................................................................................................. 30

FIGURE 3.2: Comparison of rolling and yawing moment coefficients obtained with ailerons and spoilers during wind tunnel testing, taken from Weick et al [24]. .......................................................... 32

FIGURE 3.3: Lateral control device configurations on the Fairchild 22 airplane during flight tests. Taken from Weick et al [23] ................. 33

FIGURE 3.4: Time history curves showing the lag characteristics of various control systems. Indicated air speed, 22m/s, full control deflection taken from Weick et al [23] .................................................. 34

FIGURE 3.5: Summary of a selection of spoiler and deflector combinations investigated by Wenzigner and Rogallo. [25] .............. 35

FIGURE 3.6: Three quarter rear-view of the test airplane as instrumented for flight showing deflected flap, drooped aileron, deflected spoiler and open slot. Taken from [26] .................................................. 36

FIGURE 3.7: Effect of chordwise location on pitching moments for flaps and spoilers, taken from Purser and McKinney [28] ...................... 38

FIGURE 3.8: Effect of angle of sweepback on rolling-moment coefficients produced by flap-type ailerons and spoilers. Flap deflection and spoiler projection measured in plane perpendicular to leading edge. Taken from Letko et al [30] ...................... 39
FIGURE 3.9: PHOTO TAKEN FROM INSIDE A BOEING 737 DURING THE APPROACH PHASE. 40
FIGURE 3.10: SPOILER DEPLOYED DURING A) CRUISE CONDITIONS OF AN AIRBUS A333
AND B) LANDING PHASE OF AN AIRBUS A380......................................................41
FIGURE 3.11: MICRO DRAG GENERATOR (MDG) SYSTEM SHOWING MDG’S ON THE UPPER
AND LOWER SURFACE ADAPTED FROM BAUER [37] ........................................43
FIGURE 3.12: EFFECT OF SPOILER SWEEP ON THE LATERAL CONTROL POWER OF A 60°
SWEPT TAILLESS AIRCRAFT CONFIGURATION. TAKEN FROM DORSET AND MEHL
[36].........................................................................................................................44
FIGURE 3.13: LATERAL CONTROL POWER OF A SPOILER AND SSD APPLIED TO A 60° SWEPT
TAILLESS AIRCRAFT CONFIGURATION. TAKEN FROM DORSET AND MEHL [36] ....45
FIGURE 3.14: NORMAL BLOWING FLUIDIC SPOILER CONCEPT TAKEN FROM LEOPOLD ET
AL [40] ....................................................................................................................46
FIGURE 3.15: PRESSURE DISTRIBUTION OF THE NORMAL BLOWING FLUIDIC SPOILER
TAKEN FROM LEOPOLD ET AL [40]. BLOWING COEFFICIENT, C_n = 0.48, ANGLE OF
ATTACK, α = 0°, REYNOLDS NUMBER, Re = 2x10^5 ............................................47
FIGURE 3.16: DIAGRAM OF A) CROSS-SECTIONAL VIEW OF THE MODEL AT THE SLOT MID-
SPAN AND B) TOP VIEW OF THE SEMI-SPAN WIND TUNNEL MODEL USED BY
TAVELLA ET AL [42] ..................................................................................................49
FIGURE 4.1: OVERVIEW OF THE PROJECT TUNNEL TEST SECTION SHOWING MAJOR
DIMENSIONS AND MODEL ORIENTATION .............................................................53
FIGURE 4.2: CROSS-SECTIONAL VIEWS OF THE A) MiGS, B) CFFS WIND TUNNEL MODEL
CONFIGURATIONS, AND C) WIND TUNNEL MODEL DETAILS ................................54
FIGURE 4.3: WIND TUNNEL MODEL INTERNAL STEEL ‘SKELETON’ LAYOUT .................55
FIGURE 4.4: PHOTO OF THE WIND TUNNEL MODEL IN CFFS CONFIGURATION ........56
FIGURE 4.5: CFFS DESIGN USED IN THE WIND TUNNEL MODEL ..............................57
FIGURE 4.6: PHOTO OF THE WIND TUNNEL SETUP WITH VERTICALLY MOUNTED MODEL, 6-
COMPONENT FORCE BALANCE, SCANIVALVES FOR SURFACE PRESSURE AND WAKE
RAKE FOR DRAG MEASUREMENT ........................................................................59
FIGURE 4.7: LAYOUT OF THE WIND TUNNEL SETUP. SOLID ARROWS SHOW THE DIRECTION
OF AIRFLOW FROM THE COMPRESSED SUPPLY, DASHED LINES INDICATE 0.35C
CFFS LOCATION AND PRESSURE SUPPLY PIPING ..............................................60
FIGURE 4.8: PHOTO OF THE WAKE RAKE USED DURING WIND TUNNEL EXPERIMENTS ....61
FIGURE 4.9: FLOW CHART OF EXPERIMENTAL PROCEDURE ......................................65
FIGURE 4.10: MiGS NORMAL TO A WALL PARTIALLY SUBMERGED BY A BOUNDARY LAYER
OF THICKNESS, δ_{0.99} ...............................................................................................71
FIGURE 4.11: CONTROL VOLUME FOR A CFFS AEROFOIL ...........................................74
FIGURE 4.12: THE EFFECT OF SAMPLING PERIOD ON WAKE SURVEY DRAG COEFFICIENT
MEASUREMENTS AT A SAMPLING RATE OF 1kHz ..................................................76
FIGURE 4.13: RESULTS OF A TRANSITION STUDY USING A SERIES OF TWO-DIMENSIONAL
TAPE (OR PINKED TAPE) OF INCREASING THICKNESS LOCATED AT 0.1C FROM THE
LEADING EDGE ON THE UPPER AND LOWER SURFACES OF THE MODEL. THE DRAG

C.D. HARLEY 5
COEFFICIENT VALUES SHOWN ARE MEASURED FROM THE FORCE BALANCE AND MOMENTUM LOSS IN THE WAKE.................................................................77

FIGURE 4.14: SPANWISE VARIATION OF STATIC PRESSURE IN THE CFFS PLENUM ........78

FIGURE 5.1: COMPUTATIONAL GRID USED FOR THE FLUIDIC SPOILER STUDY. CASE SHOWN IS FOR SPOILER AT 0.65x/c ........................................................................85

FIGURE 5.2: EFFECT OF GRID REFINEMENT ON AEROFOIL LIFT COEFFICIENT, FOR UPPER SURFACE CFFS AT 0.65C, WITH MAX. Cµ, AND α = 6° ........................................................................86

FIGURE 5.3: CLOSE-UP OF THE MESH AROUND THE RA16SC AEROFOIL WITH INLAY SPOILER ..................................................................................88

FIGURE 5.4: GEOMETRIC SPOILER VALIDATION CASE FOR A 16% SUPERCRITICAL WING WITH INLAY SPOILER, δ = 20°, α = 0°, AND Re = 1.9x10⁶ [61] ...........................................89

FIGURE 5.5: CLOSE-UP OF THE MESH AROUND THE NACA 0018 AEROFOIL WITH NORMAL BLOWING FLUIDIC SPOILER .................................................................90

FIGURE 5.6: FLUIDIC SPOILER VALIDATION CASE FOR A NACA 0018 AIRFOIL WITH JET ISSUING FROM THE LOWER SURFACE; α = 0°, AND Re = 2x10⁵, EXPERIMENTAL Cµ = 0.48, SIMULATED Cµ = 0.24.[40] ..................................................92

FIGURE 6.1: COMPARISON OF THE BASELINE EXPERIMENTAL AND NUMERICAL LIFT, DRAG AND SURFACE PRESSURE DISTRIBUTIONS ..................................................................94

FIGURE 6.2: EFFECT OF MICRO GEOMETRIC SPOILER (MiGS) LOCATION ON THE CHANGE IN EXPERIMENTAL LIFT, DRAG AND PITCHING MOMENT WITH SPOILER HEIGHT. ANGLE OF ATTACH RANGE: ○ = 0°, Δ = 3°, □ = 6°. CORRESPONDING SURFACE PRESSURE DISTRIBUTION PLOTS (IN FIG. 6.3) INDICATED IN VERTICAL AXIS (ACp, BCp, CCP, DCP) ........................................................................98

FIGURE 6.3: EFFECT OF 0.03C MiGS LOCATION ON CHANGE IN SURFACE PRESSURE DISTRIBUTIONS AT α = 6° .............................................................................99

FIGURE 6.4: GAIN (EFFECTIVENESS) OF MiGS FOR BOTH LOWER AND UPPER SURFACES, AND BOTH CHORDWISE LOCATIONS ..................................................99

FIGURE 6.5: EFFECT OF COUNTER-FLOW FLUIDIC SPOILER (CFFS) LOCATION ON THE CHANGE IN EXPERIMENTAL LIFT, DRAG AND PITCHING MOMENT WITH BLOWING COEFFICIENT. ANGLE OF ATTACH RANGE: ○ = 0°, Δ = 3°, □ = 6°. CORRESPONDING SURFACE PRESSURE DISTRIBUTION PLOTS (IN FIG. 6.6) INDICATED IN VERTICAL AXIS (ACp, BCp, CCP, DCP). (Z IS THE MOMENT ARM FOR THE JET) .......................102

FIGURE 6.6: EFFECT OF Cµ=0.05 BLOWN CFFS LOCATION ON CHANGE IN SURFACE PRESSURE DISTRIBUTIONS AT α = 6° ..................................................103

FIGURE 6.7: GAIN (EFFECTIVENESS) OF CFFS FOR BOTH LOWER AND UPPER SURFACES, AND BOTH CHORDWISE LOCATIONS ..................................................103

FIGURE 6.8: EXPERIMENTAL LIFT-DRAG POLAR PLOTS COMPARED WITH DATA SHEET (MiGS) AND CFD (CFFS) RESULTS .................................................................106

FIGURE 6.9: EXPERIMENTAL PITCHING MOMENT – LIFT POLAR PLOTS ..................................106

FIGURE 6.10: FORCE BALANCE DRAG COEFFICIENT FROM DUAL SURFACE BLOWING FOR YAW CONTROL. UPPER SURFACE CFFS = 0.35x/c. LOWER SURFACE CFFS = 0.65x/c. ZERO PITCHING MOMENT (CM) AND LIFT (CL) LOCI INDICATED ........109

C.D. HARLEY 6
FIGURE 9.1: GEOMETRIC DEFINITION OF AN AEROFOIL WITH A PLAIN FLAP CONTROL SURFACE FOR THE DETERMINATION OF THE THEORETICAL LIFT COEFFICIENT.... 124
LIST OF TABLES

**TABLE 4.1:** Wind tunnel balance force and torque nominal ranges, in force balance axes... 62

**TABLE 4.2:** Uncertainty calculation of the force balance lift coefficient. The expanded uncertainty at the 95% confidence level for $C_L = 0.6197$ is 9.15%. 81

**TABLE 5.1:** Overview of the important mesh parameters used in the mesh refinement study 86

**TABLE 5.2:** Validation cases conducted for the CFD process 87

**TABLE 9.1:** Parameter ranges for test data for plain trailing-edge flaps 124
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>Speed of sound</td>
<td>m/s</td>
</tr>
<tr>
<td>A</td>
<td>Axial force</td>
<td>N</td>
</tr>
<tr>
<td>c</td>
<td>Chord length</td>
<td>m</td>
</tr>
<tr>
<td>$c_i$</td>
<td>Functional relationship in uncertainty analysis</td>
<td>-</td>
</tr>
<tr>
<td>$C_L$</td>
<td>Lift coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$C_M$</td>
<td>Pitching moment coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$C_p$</td>
<td>Pressure coefficient</td>
<td>-</td>
</tr>
<tr>
<td>$C_\mu$</td>
<td>Blowing coefficient $= \frac{\rho_{\infty} h_j V_j^2}{q_{\infty} c}$</td>
<td>-</td>
</tr>
<tr>
<td>D</td>
<td>Drag force</td>
<td>N</td>
</tr>
<tr>
<td>F</td>
<td>Force</td>
<td>N</td>
</tr>
<tr>
<td>$F_x$</td>
<td>Balance drag force</td>
<td>N</td>
</tr>
<tr>
<td>$F_y$</td>
<td>Balance side force</td>
<td>N</td>
</tr>
<tr>
<td>$F_z$</td>
<td>Balance lift force</td>
<td>N</td>
</tr>
<tr>
<td>h</td>
<td>Slot height</td>
<td>m</td>
</tr>
<tr>
<td>$h_{eff}$</td>
<td>Effective height of spoiler in the freestream</td>
<td>m</td>
</tr>
<tr>
<td>$h_s$</td>
<td>MiGS height</td>
<td>m</td>
</tr>
<tr>
<td>k</td>
<td>Coverage factor</td>
<td>-</td>
</tr>
<tr>
<td>L</td>
<td>Lift force</td>
<td>N</td>
</tr>
<tr>
<td>M</td>
<td>Pitching moment</td>
<td>Nm</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>Mass flow rate</td>
<td>Kg/s</td>
</tr>
<tr>
<td>n</td>
<td>Flat plate velocity profile constant</td>
<td>-</td>
</tr>
<tr>
<td>N</td>
<td>Normal force</td>
<td>N</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
<td>Unit</td>
</tr>
<tr>
<td>--------</td>
<td>-------------------------------------</td>
<td>-------</td>
</tr>
<tr>
<td>P</td>
<td>Static pressure</td>
<td>Pa</td>
</tr>
<tr>
<td>$P_T$</td>
<td>Total pressure</td>
<td>Pa</td>
</tr>
<tr>
<td>q</td>
<td>Dynamic pressure</td>
<td>Pa</td>
</tr>
<tr>
<td>R</td>
<td>Slot radius of curvature</td>
<td>m</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
<td>-</td>
</tr>
<tr>
<td>$R_x$</td>
<td>Reactionary force</td>
<td>N</td>
</tr>
<tr>
<td>s(x)</td>
<td>Standard deviation of the sample</td>
<td>-</td>
</tr>
<tr>
<td>S</td>
<td>Area</td>
<td>$m^2$</td>
</tr>
<tr>
<td>t</td>
<td>Time</td>
<td>s</td>
</tr>
<tr>
<td>$T_x$</td>
<td>Balance rolling moment</td>
<td>Nm</td>
</tr>
<tr>
<td>$T_y$</td>
<td>Balance pitching moment</td>
<td>Nm</td>
</tr>
<tr>
<td>$T_z$</td>
<td>Balance yawing moment</td>
<td>Nm</td>
</tr>
<tr>
<td>$u(\bar{x})$</td>
<td>Standard uncertainty of the measurement</td>
<td>-</td>
</tr>
<tr>
<td>$u_{cal}(x)$</td>
<td>Standard uncertainty of the calibration file</td>
<td>-</td>
</tr>
<tr>
<td>$V_x$</td>
<td>X component of the velocity</td>
<td>m/s</td>
</tr>
<tr>
<td>U(x)</td>
<td>Expanded uncertainty</td>
<td>-</td>
</tr>
<tr>
<td>V</td>
<td>Velocity</td>
<td>m/s</td>
</tr>
<tr>
<td>$x/c$</td>
<td>Non-dimensionalised chord length</td>
<td>-</td>
</tr>
<tr>
<td>Y</td>
<td>Side force</td>
<td>N</td>
</tr>
<tr>
<td>$y+$</td>
<td>Non-dimensional wall distance</td>
<td>-</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>Angle of attack</td>
<td>Degrees</td>
</tr>
<tr>
<td>$\delta$</td>
<td>Spoiler deflection</td>
<td>Degrees</td>
</tr>
<tr>
<td>$\delta_{0.99}$</td>
<td>Laminar boundary layer thickness</td>
<td>m</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>Ratio of specific heats of air (1.4)</td>
<td>-</td>
</tr>
</tbody>
</table>
NOMENCLATURE

\( \theta \) Jet ext angle relative to mean chord line Degrees
\( \rho \) Air density Kg/m\(^3\)
\( \xi_{0.2c \text{FLAP}} \) 0.2c Flap deflection Degrees

Subscripts

1 Properties related to station 1 conditions
2 Properties related to station 2 conditions
\( \infty \) Properties related to freestream conditions
aerofoil Properties related to the aerofoil
BL Properties related to effective boundary layer conditions
CFFS Properties related to the counter-flow fluidic spoiler
CP Properties related to surface pressure distribution
j Properties related to the fluidic spoiler jet conditions
MiGS Properties related to Micro Geometric Spoiler plate Properties related to flat plate
s Properties related to a geometric spoiler
Wake Properties related to wake conditions
y Properties related to boundary layer conditions at y above the wall

Abbreviations

2D Two-Dimensional
3D Three-Dimensional
ACE Advanced Control Effector
CC Circulation Control
<table>
<thead>
<tr>
<th>Acronym</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD</td>
<td>Computation Fluid Dynamics</td>
</tr>
<tr>
<td>CFFS</td>
<td>Counter-Flow Fluidic Spoiler</td>
</tr>
<tr>
<td>DAQ</td>
<td>Data Acquisition</td>
</tr>
<tr>
<td>EPSRC</td>
<td>Engineering and Physical Sciences Research Council</td>
</tr>
<tr>
<td>ESDU</td>
<td>Engineering Sciences Data Unit</td>
</tr>
<tr>
<td>FLAVIIR</td>
<td>Flapless Air Vehicle Integrated Industrial Research</td>
</tr>
<tr>
<td>ICE</td>
<td>Innovative Control Effector</td>
</tr>
<tr>
<td>LE</td>
<td>Leading Edge</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>MaGS</td>
<td>Macro Geometric Spoiler</td>
</tr>
<tr>
<td>MDG</td>
<td>Micro Drag Generator</td>
</tr>
<tr>
<td>MiGS</td>
<td>Micro Geometric Spoiler</td>
</tr>
<tr>
<td>NACA</td>
<td>National Advisory Committee for Aeronautics</td>
</tr>
<tr>
<td>NBFS</td>
<td>Normal Blowing Fluidic Spoiler</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-Averaged Navier-Stokes</td>
</tr>
<tr>
<td>SIMPLE</td>
<td>Semi-Implicit Method for Pressure-Linked Equations</td>
</tr>
<tr>
<td>SSD</td>
<td>Spoiler-Slot-Deflector</td>
</tr>
<tr>
<td>SST</td>
<td>Shear Stress Transport</td>
</tr>
<tr>
<td>TE</td>
<td>Trailing Edge</td>
</tr>
<tr>
<td>UAV</td>
<td>Unmanned Aerial Vehicle</td>
</tr>
<tr>
<td>URANS</td>
<td>Unsteady Reynolds-Averaged Navier-Stokes</td>
</tr>
</tbody>
</table>
ABSTRACT

ABSTRACT OF THESIS submitted by Christopher Donald Harley for the Degree of Doctor of Philosophy and entitled ‘Aerodynamic Performance of Low Form Factor Spoilers’. Submitted September 2010

The development of low form factor flight controls is driven by the benefits of reducing the installed volume of the control device and/or minimising the change in external geometry, with particular application to flight control of low observable aircraft. For this work, the term ‘low form factor’ does not refer to the aspect ratio of the control device rather the overall installed volume. This thesis compares the use of low form factor geometric and fluid devices on a NACA 0015 aerofoil section through two-dimensional numerical analysis and low speed wind tunnel experiments. The geometric spoiler is implemented as a small (boundary layer scale) variable height tab oriented normal to the local surface, referred to as a Micro Geometric Spoiler (MiGS). The fluidic spoiler is implemented as an air jet tangential to the local surface acting in the forward direction, referred to as a Counter-Flow Fluidic Spoiler (CFFS). Two chordwise spoiler locations were considered: 0.35c and 0.65c. Numerical analysis was undertaken using a commercial CFD code using an unsteady solver and k-omega shear-stress-transport turbulence model. Experimental forces and moments were measured via an overhead force balance, integrated surface pressures and pressure wake survey. Device performance is assessed against the magnitude of control achievable compared to macro scale spoilers and trailing edge controls (effectiveness), the ratio of aerodynamic output to control input (efficiency or gain), the shape of control response curve (linearity), and the degree of control cross coupling.

Results show that the MiG and CFF spoilers work by a similar mechanism based on inducing flow separation that increases the pressure ahead of the spoiler and reduces the pressure downstream. Increasing control input increases drag and reduces lift, however the change in pitching moment is dependent on chordwise location. Chordwise location has a significant effect on effectiveness, efficiency, linearity and separability. Forward MiGS location gives the largest drag gain however the control response is strongly nonlinear with angle of attack and there is a significant undesirable coupling of drag with pitching moment. Aft MiGS location significantly improves control linearity and reduces pitching moment coupling however the drag gain is much reduced. For the CFFS, the control linearity with respect to control input and angle of attack is good for both forward and aft locations, with the aft location giving the largest gain for lift and drag. The control response trends predicted from numerical analysis are good, however a calibration factor of around ½ has to be applied to the control input momentum to match the experimentally observed gains. Furthermore numerical control drag polars under predict the change in lift with change in drag at low blowing rates. Through the use of a CFFS device on both the upper and lower surfaces of a wing section it is possible to generate control drag inputs fully decoupled from both lift and pitching moment, thus potentially simplifying device control law implementation within an integrated yaw control system.
DECLARATION

No portion of the work referred to in the thesis has been submitted in support of an application for another degree or qualification of this or any other university or other institute of learning.

Christopher Harley, September 2010

COPYRIGHT STATEMENT

The author of this thesis (including any appendices and/or schedules to this thesis) owns certain copyright or related rights in it (the “Copyright”) and s/he has given The University of Manchester certain rights to use such Copyright, including for administrative purposes.

Copies of this thesis, either in full or in extracts and whether in hard or electronic copy, may be made only in accordance with the Copyright, Designs and Patents Act 1988 (as amended) and regulations issued under it or, where appropriate, in accordance with licensing agreements which the University has from time to time. This page must form part of any such copies made.

The ownership of certain Copyright, patents, designs, trademarks and other intellectual property (the “Intellectual Property”) and any reproductions of copyright works in the thesis, for example graphs and tables (“Reproductions”), which may be described in this thesis, may not be owned by the author and may be owned by third parties. Such Intellectual Property and Reproductions cannot and must not be made available for use without the prior written permission of the owner(s) of the relevant Intellectual Property and/or Reproductions.
Further information on the conditions under which disclosure, publication and commercialisation of this thesis, the Copyright and any Intellectual Property and/or Reproductions described in it may take place is available in the University IP Policy (see http://documents.manchester.ac.uk/DocuInfo.aspx?DocID=487), in any relevant Thesis restriction declarations deposited in the University Library, The University Library’s regulations (see http://www.manchester.ac.uk/library/aboutus/regulations) and in The University’s policy on Presentation of Theses.
ACKNOWLEDGEMENTS

I would like to start by thanking Dr Bill Crowther, firstly for offering me the opportunity to do this PhD, secondly for his supervision throughout and finally for his brutally honest critique where necessary. I would like to thank Dr Alistair Revell for his CFD knowledge and support, and Dr Rodger Edwards for his guidance and support. I must also thank the BAE Systems/EPSRC sponsored FLAVIIR project for funding this work.

I have been fortunate enough to spend part of the PhD career at the now closed offsite facility, Barton. I would like to thank the long serving lab technicians, Mike, Ken and Dave who gave a wealth of engineering knowledge and general advice. This was a true research facility where I learnt the art of research with the help of my fellow PhD students: Russell, Ken, Steve Michie, Chip, Phil, Matt, Steve Liddle, and Paul Wilde. A group tea break solved many problems.

The second part of my PhD has been spent at the George Begg Building, where my University career began. I would like to thank the technicians Mike, Dave and Lee who collectively helped me put the wind tunnel experiment together, and where there was always an open ear willing to listen. I would also like to thank John Ashley, Ben, Johnny Potts, Phil and Matt who were always willing to discuss research over a brew.

I would like to thank my closest family and friends and especially Kate, who has supported me throughout this process, but who I have undoubtedly neglected over the past few years, especially since I have been writing and working. Finally, to my Mum and Dad, thank you being there every step of the way over the past 28 years.
1 INTRODUCTION

The aim of this chapter is to introduce the “low form-factor” spoiler class and set it in context within the broader field of aircraft spoilers for flight control. The low form factor devices investigated during this study are introduced and a description of the motivation behind the work is given. Finally, the aim and objectives for this thesis are defined.

1.1 Low form factor spoilers for flight control

Whilst much of aerodynamic design is concerned with maximising the efficiency with which attached air flows around a solid surface, there are specific operational incidences where the flow is required to depart from or 'separate' from the surface. Typically a device used to cause flow separation is referred to as a 'spoiler', in the sense that it 'spoils' the smooth flow around an aerodynamically contoured body. An aircraft spoiler is a type of geometric flight control typically mounted on the upper surface of a wing that when deflected causes the flow to separate. This results in an increase in drag, a loss in lift and a change in pitching moment, with the sign of the pitching moment dependent on the chord wise location of the spoiler.

Aircraft require control about three axis for flight control, which can be grouped under longitudinal control (changes to rate of pitch), and lateral control (changes to rate of roll and rate of yaw). Spoiler devices are suitable for lateral aircraft control due to their ability to rapidly deploy, produce favourable yawing moments and typically cause lower changes in pitching moment compared to ailerons. Spoilers deployed independently on either aircraft wing, asymmetric operation, can be used to provide the majority of the lateral control authority required during flight [1]. Spoilers deployed
simultaneously on either wing, symmetric operation, can provide a method of lift dumping, and air braking, typically used during landing [2]. Spoilers can also provide active control for flutter suppression [3], direct lift control [4] and gust load alleviation [5]. Figure 1.1 shows a photo of an Airbus A380 during the landing phase with all eight spoilers fully (per wing) deployed for lift dumping and air braking.

Figure 1.1: Photo of an Airbus A380 during landing phase with spoilers and high lift devices fully deployed. Taken from [6]

The motivation for the work in this thesis is based on the development of low 'form factor' spoiler-like flight controls on lifting surfaces located in between of the leading and trailing edges. Low form factor devices are defined by utilising minimum wing volume for installation of the control devices and systems, and are compatible with best practice for low observable design [69]. For the purposes of this work ‘low form factor’ should not be confused with ‘low aspect ratio’ controls. Two types of low form factor spoiler are considered in the work: Micro Geometric Spoilers (MiGS) and Counter-
Flow Fluidic Spoilers (CFFS). A schematic comparing the main flow field features of each of the spoiler types compared with a macro geometric spoiler is shown in Figure 1.2.

![Comparison of macro geometric spoiler (MaGS), micro geometric spoiler (MiGS) and counter-flow fluidic spoiler (CFFS).](image)

**Figure 1.2: Comparison of macro geometric spoiler (MaGS), micro geometric spoiler (MiGS) and counter-flow fluidic spoiler (CFFS).**

For the purposes of this work a micro geometric spoiler is defined as a device whose deployed length scale is of a similar order to the local boundary layer thickness at the point of operation. This contrasts with conventional 'macro geometric spoilers' where the deployed length scale is much larger than the local boundary layer thickness. The Micro Geometric Spoilers considered here are similar in function to what are referred to as micro geometric tabs, which have been variously used for load alleviation on aircraft wings [7], load alleviation on helicopter rotor blades [8], and load alleviation on wind turbine blades [9]. The work in this thesis is distinct in that it considers the use of MiGS and CFFS as flight control devices directly in comparison with macro geometric spoilers, and trailing edge controls.

A Counter-flow Fluidic Spoiler is a device that produces a thin tangential jet of air on the surface of a wing in a direction opposing the local flow direction. Tangential blowing in general for flow control has been widely studied with applications mainly
focussing on the area of Circulation Control (CC) which is based on control of separation location on a curved trailing edge [10] or, less commonly leading edge blowing which is based on vortex modification [11]. CC applications are historically concerned with increasing lift for high lift applications, however there have been recent applications where CC has been used as control effectors [12], [13]. In this work a trailing edge dual upper/lower surface co-flow slot device is used to provide lift modulation in a positive and negative sense. The present work is distinct to the foregoing in that a counter-flow tangential jet is used and the aim is primarily to produce drag through flow separation, with change in lift a secondary (but still important) consideration. In the same way that flow topology generated by suction is structurally different to the topology of blowing, co-flow and counter-flow topology are also structurally different, with counter-flow introducing distinct separation structures in the flow whereas no additional structures are produced by co-flow.

Whilst MiGS and CFFS are very different from a practical implementation point of view, there are strong similarities in the nature of the flow control input they provide, in that they both introduce a forcing in a counter-flow tangential direction. A MiGS does this by effective reduction of momentum in the co-flow direction, whereas a CFFS achieves this by addition of momentum in the counter-flow direction. As such, it is instructive to compare these devices in the same study since this sheds light on the fundamental fluid mechanism common to both.

An aim of this work is to show that the mechanism of flow control of MiGS and CFFS devices is similar to that of macro geometric spoilers, based on aerodynamic investigations of MiGS and CFFS. Two-dimensional wind tunnel investigations of MiGS and CFFS and two-dimensional numerical analyses of CFFS have been
performed and are presented in later chapters of this thesis. The independent variables used in the investigations are the actuation input (height for MiGS, and blowing coefficient for CFFS), angle of attack, and chordwise geometric location of the device. The dependent variables are lift, drag and pitching moment. Aerodynamic performance is discussed in terms of effectiveness, efficiency, linearity and cross coupling. Effectiveness considers the relative magnitude of the control output for a particular configuration, whereas efficiency (or gain) describes the ratio of aerodynamic output to control input (where the control input for MiGS is the spoiler height nondimensionalised with wing chord, $h_s/c$, and for CFFS is blowing coefficient, $C_\mu$). Effectiveness and efficiency are linked in the absence of control saturation. Effectiveness is typically determined by the control gain at the test condition. Control linearity refers to the degree to which control outputs are simply proportional to control inputs. Whilst some degree of nonlinear control is acceptable, increasing nonlinearity increases the complexity of the control system implementation and may limit the ultimate authority of the control. Control cross coupling refers to the degree to which changes in lift, drag and pitching moment due to control inputs are correlated. Ideally, from an implementation point of view, the control outputs should be fully uncorrelated. For a single control device this is not possible, however by the use of an upper and lower surface device it is possible to remove the correlation between drag and lift, and drag and pitching moment.
1.2 Aims and objectives

The aim of this thesis is:

To develop and demonstrate an understanding of the aerodynamic performance of Micro Geometric and Counter-Flow Fluidic Spoilers for flight control applications.

The objectives of this thesis are to:

- Set the work in context within the broader field of aircraft flight control devices and establish the motivation of the work (Chapter 1)

- Describe the aerodynamics of aircraft control devices, in particular spoilers and introduce the qualitative aerodynamic models used in this work (Chapter 2)

- To provide a relevant literature review, focusing on the historical development of aircraft geometric and fluidic spoiler technology (Chapter 3)

- Describe the experimental and computational research methods used in this work (Chapter 4 and 5)

- Present and discuss the computational and experimental results (Chapter 6)

- Present conclusions and propose areas for future research (Chapter 7)
2 THEORY

The aim of this chapter is to build understanding of the aerodynamic characteristics of spoiler devices. A qualitative model which collects conventional trailing edge and spoiler devices into equivalent modifications of camber to predict their aerodynamic response is presented. The similarities and differences of the flow topology of geometric and fluidic spoilers are then discussed, followed by a description of a geometric spoiler flow field. Finally the effect of a geometric spoiler on the surface pressure distribution is described.

2.1 Qualitative model for the effect of camber

A very simple but useful model for predicting the qualitative performance of flight controls on lifting surfaces can be derived by considering the effect of the control on the camber of the local aerofoil section. For a two-dimensional symmetrical aerofoil section at a fixed low angle of attack (below the stall angle), a positive increment in camber produces a positive increment in lift, positive increment in drag and a negative increment in pitching moment (taken about the aerodynamic centre). A negative increment in camber at the same conditions produces a negative increment in lift, a positive increment in drag and a positive increment in pitching moment. This behaviour is summarised in Figure 2.1. Note that for camber change through rearward mounted spoiler deflection, the model is qualitatively correct for lift and drag, however the sign of the pitching moment will in general depend on the chordwise location of the spoiler (forward spoiler locations will tend to produce a pitching moment change in the opposite sense to aft located devices). The drag considered in this case is the profile
drag, however one could also consider the pressure drag which does not take into account the effect of the skin friction.

![Diagram showing lift and drag effects](image)

**Figure 2.1**: Qualitative illustration of the effect of aerofoil camber on lift drag and pitching moment

### 2.2 Comparison of flow topology models for geometric and fluidic spoilers

Figure 2.2 shows the steady time-averaged two-dimensional streamline topology for three spoiler devices, a) a micro geometric spoiler (MiGS), b) i) a normal blowing fluidic spoiler (NBFS, a jet exhausts from a slot normal to the local surface) and b) ii) a counter-flow fluidic spoiler (CFFS). These diagrams are based on streamline data from CFD analyses; refer to Chapter 5 for details on CFD methodology used. It can be seen in Figure 2.2 that the flow field from a geometric and fluidic spoiler is broadly similar, but with a number of detailed differences regarding the number and location of the
separation and attachment points. Upstream of all spoiler locations there is a separation point, however the fluidic spoiler cases also have an attachment point due to the jet momentum causing a recirculating region upstream of the jet. Downstream of the spoiler locations a large recirculating region is formed, however unlike the geometric and normal blowing spoilers the CFF spoiler does not have a small recirculating region.

From the above observations it is expected that the control response of micro geometric and fluidic spoiler devices is similar. However, for macro geometric devices at large deflection angles the control drag response is dominated by the increase in projected area of the spoiler, so it is likely that there will be significant differences with fluidic devices in this case.

![Figure 2.2: Steady time-averaged two-dimensional streamline topology for three spoiler devices, a) geometric spoiler and b) i) normal blowing fluidic spoiler, and b) ii) counter-flow fluidic spoiler.](image)

### 2.3 Geometric spoiler flowfield and global aerodynamic coefficients

This section discusses the flow field around a two-dimensional aerofoil with deflected geometric spoiler and presents the effect of spoiler deflection on the global aerodynamic coefficients of a lifting surface. It is based mainly on information from the
ESDU data sheet item on spoiler aerodynamics [2] and with some additional material from standard aerodynamics texts [14].

A deflected spoiler will generally cause the flow over a lifting surface to separate. The extent and detailed characteristics of the separation will in general depend on the effective height of the spoiler (projected frontal height), the chordwise position, the aerofoil section, the aerofoil incidence, and the free stream Mach number and Reynolds number. Figure 2.3 shows the typical steady flow field features around an aerofoil with deflected rearward mounted (macro) geometric spoiler at a low positive angle of attack, adapted from Lee and Bodapati [15]. There are two main regions of separation identified in the figure. One termed the hinge bubble is located just ahead of the spoiler hinge position and encompasses the lower region of the spoiler. The second region of separation originates from the spoiler tip and emanates aft of the spoiler as a free shear layer. If this separation stays completely detached from the aerofoil surface an approximately constant base pressure is formed. At low spoiler deflections and/or a forward located spoiler the separated flow from the spoiler tip can reattach ahead of the trailing edge. This can cause a non-linear change in lift with spoiler deflection, as shown in Figure 2.4 a).

![Large separation behind spoiler from tip to trailing edge](image)

**Figure 2.3: Typical flowfield due to a rearward mounted macro geometric spoiler**

Figure 2.4 shows the effect of a deflected spoiler on the global aerodynamic coefficients of a wing. A rearward mounted deflected spoiler causes a loss in lift, consistent with the qualitative model of Figure 2.1 a), an increase in drag, consistent with the qualitative model of Figure 2.1 b), and an increase in pitching moment,
consistent with the qualitative model of Figure 2.1 c). Chordwise location is an interesting variable, such that a forward mounted upper surface spoiler at low deflections can cause an increase in lift, due to flow reattaching downstream of the spoiler, and generally causes a reduction in pitching moment.

![Diagram showing lift change with spoiler deflection](image1)

**Figure 2.4**: Typical rearward mounted (apart from where stated) spoiler effect on lift, drag, pitching moment and change in surface pressure with deflection. a) and b) adapted from ESDU data sheet [2]. c) adapted from McLachlan et al [16]

### 2.4 Spoiler pressure distribution model

Separation of the flow over a lifting surface causes a modification of the surface pressure distribution and therefore changes the overall forces and moments. A qualitative model for the change in surface pressure distribution for an aerofoil with deflected upper surface rearward mounted spoiler is shown in Figure 2.5. There is an increase in pressure ahead of the spoiler and a decrease in pressure downstream of the spoiler. The pressure on the lower surface decreases with spoiler deflection. This decrease is relatively small compared to the changes on the upper surface and is associated with the reduction in circulation around the aerofoil, due to a change in the effective camber of the aerofoil.
Figure 2.5: Effect of increasing spoiler deflection/height on the wing section surface pressure distribution
3 LITERATURE REVIEW

The first part of this section will review the early development of geometric spoilers for lateral aircraft control. This will be followed by the application of spoilers for lateral control of tailless aircraft. The section will then conclude with a review of fluidic spoiler type devices for lateral aircraft control.

3.1 Geometric spoilers for aircraft flight control

3.1.1 Development of spoilers for aircraft control

Hinged flight controls on the trailing edges of aerodynamic surfaces are relatively simple to implement, are effective in producing the control moment magnitudes needed for flight control, and the control response characteristics are generally linear over a useful range of angle of attack and control surface deflection. As such, most aircraft in the early part of the 20th Century used combinations of aileron, elevator and rudder for flight control. One issue with ailerons for roll control is the drag on the down going aileron generates a yawing moment in the adverse sense, in that it generates a yaw rate of opposite sense to that required for a coordinated turn [17]. A further issue with use of ailerons is that control deflection generates significant pitching moment, which tends to twist the wing in the opposite sense to the deflection of the control. In the first instance this leads to a reduction in aileron effectiveness, moreover with slender wings and high speed flight this may lead to control reversal [18]. In light of these issues, a number of studies were initiated in the 1920’s looking at the use of alternative flight controls for lateral control, in particular, spoiler type controls for which the yawing
moment is pro turn and there is significantly less pitching moment for a given change in lift compared to trailing edge controls.

Some of the earliest spoiler type devices included “Baffle flaps” [19] and “Projecting flaps” [20]. Baffle flaps were leading edge spoilers on the upper surface of a wing, aimed at decreasing lift, increasing drag and thereby causing a rolling moment and a pro turn yawing moment. Projecting flaps were very similar to baffle flaps, however, were not limited to only forward chordwise location, but various locations across the wing surface. Figure 3.1 shows a typical forward mounted spoiler from the early 1930’s. References for both forms of spoiler terminology can be traced back to the early 1920’s. Around the early 1930’s these terms were quickly dropped for the more common “spoiler” terminology, but it is unclear who first used the term.

![Image of Plain Clark Y wing. Spoilers and ailerons deflected individually.](image)

**Figure 3.1: One of the first applications of a forward mounted and forward hinged spoiler to aircraft wings taken from Weick and Wenzinger [21]**

Eliminating adverse yaw was an important driving factor in the first spoiler investigations of the 1930’s and 1940’s. A number of investigations were performed by NACA during this period aiming to gain a full understanding of spoiler devices for lateral aircraft control[17], [22], [23], [19-21], [24-33], the main findings of which were reported to the state of congress in NACA’s 18th annual report [22]. The report states the overall aim of the investigations is to obtain “Satisfactory stability and
controllability...attained throughout the entire range of speed or of angle of attack”.

An actual definition of satisfactory stability and control was not provided in the report and was not attempted until flight investigations a few years later [23]. The flight investigations tested control devices for their lateral aircraft control potential, in particular ailerons and spoilers. The required flight characteristics for lateral aircraft control were defined as:

- The production of a rolling moment that corresponds to a lateral movement of the centre of pressure by 7.5 percent of the span.

- The maximum rolling rate is aircraft and pilot sensitive but was not given a definitive limit.

- The response of the aircraft to any movement of the lateral control surface should be immediate, any noticeable delay or hesitation in the action is objectionable.

- Finally, the action should be graduated so that the acceleration and maximum rate of roll increase smoothly and regularly as the stick deflection is increased.

Wind tunnel tests performed by Weick et al [24] showed forward mounted spoilers to be effective, especially at high angles of attack where the effectiveness of ailerons reduces significantly. Figure 3.2 compares the rolling and yawing moments of a spoiler deflected on the upper surface of a starboard wing, with a positively deflected aileron on the starboard wing, as the angle of attack is increased. The aileron rolling moment is independent of angle of attack up to 12 degrees, while the spoiler rolling moment increases with angle of attack up to 16 degrees. The opposite is observed in the yawing moment response, where the spoiler yawing moment is independent of angle of attack up to 12 degrees and the aileron yawing moment decreases with angle of attack. The
constant positive yawing moment of the spoiler is a favourable (pro turn) yawing moment. However, this report lacks any comparison of the effect of pitching moment due to control deflection.

![Comparison of rolling and yawing moment coefficients obtained with ailerons and spoilers during wind tunnel testing, taken from Weick et al [24].](image)

**Figure 3.2:** Comparison of rolling and yawing moment coefficients obtained with ailerons and spoilers during wind tunnel testing, taken from Weick et al [24].

An investigation into the in-flight characteristics of lateral control devices is presented by Weick et al [23]. The Fairchild 22 aircraft was used with the configurations shown in Figure 3.3, consisting of three types of forward mounted spoiler; a rearward hinged spoiler, a forward hinged spoiler and a retractable spoiler.
Figure 3.3: Lateral control device configurations on the Fairchild 22 airplane during flight tests. Taken from Weick et al [23].

The aircraft time response due to inputs from these spoiler configurations and an aileron control input is shown in Figure 3.4. Pilots reported a delay in aircraft response to a control input from these forward mounted spoilers. The figure shows the aircraft control response due to an aileron deflection has a lag of 0.1s and a spoiler deflection has a lag of between 0.4s and 0.6s, based on a rate of roll of 0.1rad/s. The time lag can be transformed into a time constant dependent on the aircraft velocity and chord length to be, \( tV/c = 5.9 \) to 8.8. The spoiler lag was only observed in the rate of roll, whereas
no lag was observed in the rate of yaw. Whilst the authors don’t suggest a reason for this, it could be due to the change of local wing section surface pressure caused by the separated flow downstream of the spoiler, whereas the main component of the drag is probably caused by the pressure drag on the front face of the spoiler which would adjust more quickly than the local wing section surface pressure. Subsequent research focused on the lateral control performance of spoilers in a more rearward chordwise location.

![Figure 3.4](image_url)

**Figure 3.4:** Time history curves showing the lag characteristics of various control systems. Indicated air speed, 22m/s, full control deflection Taken from Weick et al [23].

Wenzigner and Rogallo [25] showed the lag time of the rolling moment coefficient of a spoiler deflection reduced by a half when a spoiler was moved from 0.29c to 0.55c, and the maximum rolling moment reduced by ~20%. Wenzigner and Rogallo also performed investigations on a number of other types of spoiler devices such as lower
surface spoilers or deflectors, upper surface spoilers and deflector combinations, and upper surface spoilers and deflector combinations with a slot allowing airflow in between, more commonly known as spoiler-slot-deflectors (SSD). Figure 3.5 shows some of the investigated spoilers in combination with ailerons and split flaps. The following provides a summary of the main conclusions found is the study.

![Summary of a selection of spoiler and deflector combinations investigated by Wenzinger and Rogallo.](image)

**a) Conventional upper surface spoiler combinations**

1. Retractable deflector, in combination with split flap
2. Rearward hinged deflector, in combination with split flap
3. Rearward hinged deflector, in combination with aileron
4. Forward hinged deflector at 0.11c, in combination with aileron

**b) Deflector combinations**

1. Spoiler and deflector, in combination with split flap
2. Spoiler and deflector, in combination with split flap
3. Retractable spoiler and deflector, in combination with split flap

**c) Spoiler Deflector combinations**

In terms of effect on lift, it was observed that spoilers or deflectors alone were not as effective as the combination of a spoiler and deflector. This effectiveness was increased.
further by adding a slot in between the spoiler and deflector such that airflow from the lower surface is fed through the aerofoil to the upper surface behind the spoiler. This causes a loss of pressure on the lower surface of the aerofoil, and an increase in pressure on the upper surface, and therefore an overall reduction in lift.

Following the improvement in lag time for a rearward mounted spoiler, a series of flight investigations were performed, aiming to test a number of lateral control configurations. One such lateral control configuration, shown in Figure 3.6, is the use of a spoiler and aileron at the same spanwise station. This configuration in the form of high lift flap and spoiler went on to become the standard configuration for transport aircraft.

![Figure 3.6: Three quarter rear-view of the test airplane as instrumented for flight showing deflected flap, drooped aileron, deflected spoiler and open slot. Taken from [26]](image)

With the advent of the Second World War, aircraft technology advanced dramatically and with it the requirements for lateral control changed. Investigations of lateral control performance of a 0.75c spoiler on a tapered wing at high speed showed an increase in
effectiveness with speed. The overall performance of which was stated “suitable to replace ailerons”, however, with the caveat that a time response investigation were required to show a negligible lag [34].

As aircraft speeds increased, wing twist due to aileron deflection became large enough to cause an appreciable loss in rolling effectiveness, known as control reversal. This high speed effect once again made spoilers attractive as lateral control devices due to their lower associated pitching moments. An investigation on the effect of wing twist caused by a spoiler deflection was performed by Fitzpatrick and Furlong [27]. Figure 3.7 shows a comparison of the effect of flaps and spoilers on the pitching moment with the hinge point of the control surface at various chord-wise locations. The change in pitching moment from the flap and spoiler deflection is of the same orientation; in this case a negative flap deflection and upper surface spoiler deflection cause a nose up or positive pitching moment. The figure shows that the negative flap deflection causes a positive pitching moment, nose up, at all hinge locations, whereas a spoiler deflected and located ahead of 0.4c causes a negative or nose down pitching moment. The overall effect of a spoiler deflection on the aircraft pitching moment will depend on the spoiler longitudinal location with respect to the aircraft centre of gravity. However, a spoiler deflection produces half of the pitching moment change compared to a flap at approximately 0.75c, which is advantageous for a reduced wing twist compared to ailerons and is therefore capable of a higher control reversal speed.
3.1.2 Effectiveness of spoilers applied to swept wings

This thesis reports and discusses the two dimensional aerodynamic performance of a MiGS and CFFS device. For future research directions it is important to highlight the effect of wing sweep on the control performance of a spoiler device.

Figure 3.8 shows the effect of wing sweep on the lateral control power of spoilers and flaps taken from Letko [30]. The spoilers and flaps span the same amount of semi-span, with the spoilers hinged at a constant 0.7c, and the flaps hinged at a constant 0.75c. There is a loss in rolling-moment with angle of sweepback at a fixed Mach number and angle of attack for both the flaps and spoilers. The rate of loss in rolling moment with increasing sweep angle is similar for both flap and spoiler. Both the flap and spoiler have an inherent directional stability mechanism, whereby the windward wing is more effective at producing an increase in drag with control deflection, therefore a control deflection increases directional stability.
Figure 3.8: Effect of angle of sweepback on rolling-moment coefficients produced by flap-type ailerons and spoilers. Flap deflection and spoiler projection measured in plane perpendicular to leading edge. Taken from Letko et al [30].

3.1.3 Use of spoilers on modern civil transport aircraft

Conventional wing/tail transport aircraft design has converged to the use of upper surface spoilers typically mounted on the wing rear spar. Civil transport aircraft today, such as the Boeing B737, use asymmetric aileron deflection and or spoiler deflection to minimise any adverse yawing moment for roll control. Figure 3.9 shows a control surface deployment comparison between cruise, roll manoeuvre and landing on
approach. Conventional spoilers are suitable lateral aircraft control devices particularly during the approach phase where a reduction in lift and an increase in drag are required. Figure 3.9 b) shows the spoilers deflected for the production of roll rate modification, while Figure 3.9 c) shows all spoilers fully deployed for lift dumping and drag increase on landing. What is not apparent from these pictures is that spoilers are deflected asymmetrically for a roll manoeuvre, and deflected symmetrically for lift dumping and air braking.

![Figure 3.9: Photo taken from inside a Boeing 737 during the approach phase](image)

Figure 3.10 shows spoilers deployed during a) cruise conditions for roll manoeuvre and b) the landing phase for lift dumping and air braking. The Airbus A380 deploys a total of 8 spoiler control surfaces on each wing covering 60% of the wing span during the landing phase.
3.1.4 Application of spoiler type devices to tailless aircraft

For tailless/finless aircraft where spoilers may be the only means of yaw control, implementation is more varied compared to aircraft with vertical lifting surfaces for yaw control. Tailless/finless aircraft such as the B2-spirit and diamond planform Pegasus UCAV use split flaps, and upper and lower surface spoiler devices respectively for yaw control.

There have been a number of studies investigating innovative control effectors for tailless aircraft [35], [36]. These studies aimed to implement control effectors that could reduce weight, improve reliability, reduce radar signature, improve aerodynamic...
efficiency and improve aircraft manoeuvrability. These control aims have moved on since the requirement for “satisfactory stability and controllability” in the 1930’s.

Wood and Bauer in 1998 performed an investigation on a number of control concepts that focused on “micro flow management” for aircraft control [35]. Micro flow management refers to the use of micro geometric/fluidic devices for flow control that can provide sufficient changes to the global aerodynamic coefficients for aircraft flight control. Micro geometric flow devices include micro drag bumps or micro drag generators (MDG), spoilers and splitter plates, which due to having heights of the order of the local boundary layer, are attractive for low observable military applications. Alone, micro flow devices may not provide comparable effectiveness compared to conventional controls, however, surface contouring technologies that alter the pressure field over a wing such that only a small disturbance is required to cause a control response may provide an indirect method of improving the effectiveness of micro controls devices.

The micro drag generator (MDG) concept consists of a number of micro tabs with heights similar to the local boundary layer, distributed at regular chordwise intervals across a spanwise station of a wing [37]. These MDG’s extrude from the wing surface to cause a local surface flow separation, which in turn causes an increase in wing drag. A high profile drag modulating system such as the MDG’s would also lend itself to a steep-descent manoeuvre for transport aircraft where high profile drag is very beneficial as discussed by Filippone [38]. The MDG concept is shown in Figure 3.11.
Wood and Bauer suggest the performance of a MDG system would be equivalent to a single spoiler device with the same overall projected area. A simple analysis is performed in this thesis that calculates the effect or the drag due to the projected face of the MiGS (similar to the MDG concept). It was found that 20% to 40% of the total drag was due to the frontal area of the MiGS, but with high dependence on chord-wise location. With a system of MiGS devices any amplification similar as that observed for the MiGS would probably be reduced. However, a study investigating performance with increasing numbers of MiGS devices would be worthwhile, assuming the cost of increasing the number of surface discontinuities was justified.

An MDG system may be attractive for its low observable characteristics; however the wing skin is often part of the load bearing structure which may be compromised by the large number of discontinuities in the wing. The need for additional strengthening may lead to an increase in aircraft weight. A single MiGS or CFFS device would therefore cause less of an impact on the wing structural load.

The Innovative Control Effectors (ICE) investigations from NASA, reported in 1996 on the lateral control effectiveness of control devices for tailless aircraft [36]. Spoiler type devices included spoiler-slot-deflectors (SSD) and lower surface spoilers (LSP). Initially an overview of spoiler type devices applied to a tailless aircraft configuration is given, and is highlighted here.
The effect of spoiler sweep angle on a diamond wing planform is shown in Figure 3.12. At high angles of attack (>20°), the figure shows that aircraft response from a spoiler deflection is dependent on spoiler sweep angle. It is not clear whether this result is planform specific, however due to the lack of literature on effectiveness due to spoiler sweep angle no general conclusions on spoiler performance can be made.

A spoiler-slot-deflector (SSD) can improve the overall effectiveness of a simple upper surface spoiler configuration for tailless aircraft lateral control. The lateral effectiveness of a spoiler and SSD applied to the same 60° swept tailless aircraft configuration is shown in Figure 3.13. The figure shows the SSD provides nearly 50% improvement in lateral control power compared to conventional spoilers at 20° angle of attack. SSD’s were also shown to remove the non-linear control response of an upper surface spoiler at small deflections.
The SSD device benefits from greater effectiveness than a conventional spoiler, however the implications with airflow movement between the upper and lower wing surfaces provides other engineering problems that have hindered further development of SSD’s.

![Graph showingSpoiler-Slot-Deflector Low-Speed Data](image)

*Figure 3.13: Lateral control power of a spoiler and SSD applied to a 60° swept tailless aircraft configuration. Taken from Dorset and Mehl [36]*

### 3.2 Fluidic spoiler concepts for aircraft flight control

#### 3.2.1 Introduction

This section reviews a number of studies investigating fluidic spoilers for lateral aircraft control. Jets of air issuing from the lower wing surfaces were initially investigated for VSTOL applications during the 1970’s [39]. Particular focus was on hover and the transition phases (from forward flight to hover, or from hover to forward flight). Both favourable and unfavourable effects can be identified during these phases.
of the VSTOL process. For example, during hover, a loss of lift due to jet thrust can be caused by the jet entraining the freestream air, and during the transition phase an increase in lift can be caused by effects of the jet-cross flow interaction on the aerodynamics of the wing. The favourable effect during the transition phase has motivated a number of studies investigating the potential use of jets issuing from the wing surfaces to cause a change to the aerodynamic characteristics and ultimately lateral aircraft control.

3.2.2 Roll control

Leopold et al [40] presents the data from a two-dimensional wind tunnel investigation of a normal blowing fluidic spoiler (NBFS) concept, previously mentioned in the theory section when comparing the flow topology of geometric and fluidic spoilers. Figure 3.14 shows a jet issuing from the lower surface of the wing with the aim of causing an increase in lift and therefore a roll control.

![Figure 3.14: Normal blowing fluidic spoiler concept taken from Leopold et al [40].](image)

The model consisted of a NACA 0018 aerofoil with a slot running 90% of the span to minimise three-dimensional effects, and located at a 0.5c. A uniform flow along the slot was obtained by the use of a plenum and internal vanes, for which there are no schematics.

The results show that at zero angle of attack and for a blowing coefficient, $C_\mu = 0.48$, the sectional lift coefficient increases by $\sim 1.0$ (this does not include the lift due to the
jet thrust). A sectional lift coefficient of 1.0 is approximately equal to an angle of attack of 10° for a NACA 0018 aerofoil. The lift was calculated from the aerofoil surface pressure distribution, which with a low density of pressure tapping’s can be a source of error, however there appears to be sufficient data to capture the changes in surface pressure. Figure 3.15 shows the model pressure distribution due to a blowing coefficient of 0.48. The effect of a NBFS is very similar to that of a geometric spoiler, with an increase in pressure ahead of the jet, a reduction in pressure downstream of the jet, and a reduction in pressure on the aerofoil surface opposite to the jet.

Leopold et al presents a wake profiles at various distances downstream of the model, but does not calculate the associated drag from these results. There are also no force balance measurements and therefore no reference to jet blowing effect on the model pitching moment coefficient. Although achieving impressive lift gains with the use of a jet, without understanding the effect of normal blowing on drag and pitching moment there cannot be a valid conclusion on the potential of normal blowing for lateral aircraft control from this work.

![Graph showing pressure distribution](image)

*Figure 3.15: Pressure distribution of the normal blowing fluidic spoiler taken from Leopold et al [40]. Blowing coefficient, $C_\mu = 0.48$, angle of attack, $\alpha = 0^\circ$, Reynolds number, $Re = 2 \times 10^5$.***
Walchli and Langan [41] applied the NBFS concept to a highly swept semi-span high aspect ratio wing in high subsonic and transonic regimes. The normal blowing spoiler configuration consisted of a row of holes just ahead of the flap hinge line. In all cases tested, increasing the blowing coefficient reduced the lift with the exception at a Mach number of 0.9, where the lift increased. This change in lift is due to influencing the shock location. The effectiveness of blowing appears to plateau after a small amount of blowing. This is the only experimental investigation of fluidic spoiler devices applied to a realistic wing planform in the transonic regime, known to the author.

### 3.2.3 Yaw control

Yaw control using a NBFS was investigated by Tavella et al [42]. The concept is very similar to that of Leopold et al and appears to be from the same research group. Leopold et al showed that a lower surface NBFS can cause a substantial increase in aerofoil lift. Tavella et al targets an increase in drag to cause a yawing moment, with minimal changes in lift. A NBFS was applied to the outboard region of a rectangular semi-span wing of constant NACA 0018 cross-section. The blowing concept is shown in Figure 3.16 a) and the wind tunnel model is shown in b).
Results show that the overall effect on lift is negligible over the angle of attack range of 
-4° to +4° up to a blowing coefficient of 0.0375. The change in drag due to a blowing
coefficient was constant over the angle of attack range of -4° to +4°. It appears that
the NBFS is sensitive to wing tip effects, however this cannot be confirmed due to the lack
of lift and drag data from a comparable NBFS study. This study also does not present
the effect normal blowing on the pitching moment of this model, therefore a conclusion
on the relative effectiveness of a NBFS for aircraft lateral control cannot be made.
For aircraft control systems it is important to understand the effect on the lift, drag and pitching moment due to actuation. Currently there is no literature known to the author that combines this information for a fluidic spoiler device, and also compares the aerodynamic performance of a fluidic and geometric spoiler.

3.3 Summary of literature review and concluding remarks

- Spoilers were initially investigated for their production of a favourable (pro turn) yawing moment with deflection, and relatively higher control reversal speeds compared to trailing edge devices.
- Early wind tunnel testing showed that forward mounted spoilers were capable of replacing ailerons for lateral control. However, flight testing of forward mounted spoilers showed a substantial lag in aircraft response compared with ailerons. Moving the spoiler further aft reduced the aircraft response time, but also reduced the effectiveness of the spoiler.
- Micro flow management devices such as micro drag generators may be attractive for low observable military applications, however implementation for flight control may require further understanding on the potential cost to the wing structural design.
- The effectiveness of macro geometric spoilers and ailerons as lateral control devices reduces with increasing wing sweep angle.
- A two-dimensional wind tunnel study of a NBFS located at the mid-chord on the lower surface of a NACA0018 aerofoil section can increase the lift by a $C_L = 1.0$ for a blowing coefficient, $C_\mu = 0.48$, at constant angle of attack.
- A NBFS located in near the wing tip of a semi span wing can provide increases in drag while producing negligible changes in lift at a blowing coefficient, $C_\mu =$
0.0375.

- A general observation of most of the papers reviewed here is the lack of lift, drag and pitching moment (for two-dimensional studies, or rolling moment, yawing moment and pitching moment for three-dimensional studies) results from a single control input. Therefore a full picture of the control characteristics cannot be performed.
4 EXPERIMENTAL METHODS

This chapter presents the experimental research methods employed to obtain the two-dimensional aerodynamic performance of Micro Geometric Spoilers (MiGS) and Counter-Flow Fluidic Spoilers (CFFS). The wind tunnel model configurations are described, along with the measurement techniques, data reduction performed and the experimental validation and uncertainties.

4.1 Experimental apparatus

4.1.1 Project wind tunnel

Wind tunnel tests were performed in the open circuit Project Wind Tunnel at the University of Manchester. The tunnel has an octagonal test section with maximum dimensions of 1m x 1.1m and a length of 2m, Figure 4.1. The maximum test section velocity is ~50m/s. The test section velocity was measured using calibrated static pressure tappings in the tunnel settling chamber ahead of the contraction cone and just ahead of the test section. The tunnel overhead six-component force/torque balance was used to measure forces and moments. The tests were conducted at a velocity of 24.5m/s corresponding to a Reynolds number based on model chord length of 6x10^5. Whilst this Reynolds number is clearly lower than that expected for full scale application, it is sufficiently high to avoid gross low Reynolds number effects within the angle of attack range tested [43].
4.1.2 Wind tunnel model configurations and manufacture

Details of the wind tunnel model geometry are shown in Figure 4.2. The wind tunnel model has a NACA 0015 aerofoil section, constant chord of 350mm and a span of 730mm. Use of a symmetrical aerofoil section and upper and lower surface spoilers locations removed the need to test at negative angles of attack. A relatively thick (15%) section was used to increase the likely changes in drag obtained from the spoiler and hence improve measurement accuracy.

The model core was manufactured from 'blue' polyurethane foam using an outsourced CNC hot wire cutting service. Recesses were cut out of the surface of the foam for 28 thin tube pipes for surface pressure measurement. The model skin was made from obechi veneer skins bonded with epoxy to the foam core and finished with carbon fibre composite sheet for surface finish and strength.
The model was mounted using the steel internal structure or ‘skeleton’ shown in Figure 4.3. Two 10mm steel rods running the length of the model allowed 5mm thick steel plates to be clamped to the model at either end. These plates were recessed into the model profile. This steel structure created extra stiffness and was the mounting point for a 25mm steel rod that connected to the force balance via a 5mm steel plate. Two CFFS plenums were recessed on opposite sides of the model so that the slots were located at 0.35c and 0.65c. This allowed a combined upper and lower surface CFFS blowing investigation to be performed. Practical constraints meant the fluidic spoiler only spanned the middle 55% of the model, hence the model is only partially two-dimensional. A photo of the model in its CFFS configuration is shown in Figure 4.4. Due to the partial span length of the CFFS slot, appropriate scaling of the balance data for the CFFS results is required to obtain equivalent full span (two-dimensional) data suitable for comparison with the surface pressure measurements, momentum loss in the wake and numerical results. The required scaling factor is the ratio of the slot length to...
span length and has been applied to all CFFS force balance measurements presented in this thesis. This is explained in section Error! Reference source not found..

![Figure 4.3: Wind tunnel model internal steel ‘skeleton’ layout](image)

### 4.1.3 MiGS design and manufacture

The MiGS test articles are implemented as variable height tabs oriented normal to the local surface. Tabs were machined from aluminium box section to give heights of 0.01c, 0.02c and 0.03c. The MiGS span was set as the same as that for the CFFS, i.e. 55% of the model span so that the same correction factor was applied to both sets of results.
4.1.4 CFFS design and manufacture

The design objective for the CFFS system is to provide maximum efficiency from a given pneumatic supply with minimum installed volume. Crowther et al [44] provides a useful discussion of the design variables for two fluidic control devices. The paper identifies the important design features of fluidic devices, which provided the starting point for the design of an efficient CFFS system.
The CFFS device produces a thin uniform velocity jet sheet from a slot within a pressurised plenum. The slot is recessed within the aerodynamic mould line of the model in keeping with the practical aim of minimising the installed drag penalty when the device is not operating. Once leaving the slot the jet sheet remains attached to the curved surface that starts as tangent to the slot exit and ends as tangent to the wing surface Figure 4.5 c). The jet sheet is encouraged to stay attached to the wing surface downstream of the slot exit through the so called Coanda effect [45-47]. A number of flight control applications make use of the Coanda effect including circulation control [48], [49] and fluidic thrust vectoring [50]. The reader may refer to these references for a detailed discussion of the Coanda effect and its applications.
Figure 4.5 shows the design of the CFFS plenum for the wind tunnel test. The key features of the CFFS design are:

1. The top plate is located in position using the housing knuckle and top plate groove. Application of a small amount of silicone in the top plate knuckle groove provided an airtight seal. The top plate was secured by two rows of 2.5mm countersunk screws.

2. The support pillars were located just behind the point at which the top plate begins to taper from 2mm to 0.5mm. This helps the formation of a uniform exit velocity along the entire slot length.

3. Good practice suggests that a contraction ratio defined by the slot height to internal plenum height of at least 10 should be used to obtain good slot flow quality. At the location where the pillars meet the top plate the contraction ratio is ~25. A contraction ratio of at least 10 also implies that the plenum static pressure will be 99% of the total pressure in the flow, which allows a method of jet velocity calculation from the plenum static pressure [44].

4. The slot lip height was defined by the limit of manufacturability as 0.2mm ± 0.05mm. Due to manufacturing tolerances and the potential for deformation, the plenum top plate was manufactured from steel sheet, while the plenum housing was manufactured from aluminium.

5. The angular range over which a coanda jet will remain attached to a curved surface is strongly dependent on the slot curvature (h/R), with smaller values improving attachment. However from an implementation point of view it is desirable to minimise R hence a compromise is necessary. Based on practical considerations and the investigation by S. Frith [51], it was decided to use a slot curvature of 5 per cent for the CFFS slots in this study.
4.1.5 Experimental set up

Recognising the challenge of obtaining accurate measurement of drag in wind tunnel experiments, both force balance and wake pressure techniques were used to improve measurement robustness by providing independent measurements. An annotated photo of the wind tunnel set-up is shown in Figure 4.6 and a diagram of the model installation shown in Figure 4.7.

![Figure 4.6: Photo of the wind tunnel setup with vertically mounted model, 6-component force balance, scanivalves for surface pressure and wake rake for drag measurement.](image)
Figure 4.7: Layout of the wind tunnel setup. Solid arrows show the direction of airflow from the compressed supply, dashed lines indicate 0.35c CFFS location and pressure supply piping.

4.1.6 Wake survey

The drag coefficient is calculated from the momentum loss in the wake caused by the model drag. The wake rake technique uses integrated total pressure measurements in the wake of a body to determine the momentum loss in the wake. For the present experiments, the survey location was one chord length downstream of the trailing edge of the model. This is far enough downstream that the static pressure variation across the wake can be assumed to be negligible [52]. The wake rake consisted of 40 total pressure tubes (single tube shown on fig 21) with a centre distance of 3mm. Two static pressure tubes were located at the extremities of the total pressure tube array. The static pressure holes were located eight tube diameters downstream of the tube tip as described as best practice in Krause et al [53]. To reduce measurement errors caused by
closely adjacent tubes the tips of the total pressure tubes were flattened perpendicular to the tube array axis, as suggested in [54].

The wake rake was mounted to a vertical rod that was attached to a manual traverse underneath the test section. Once the Scanivalve had stepped through each total and static pressure probe, the manual traverse was used to move the mounting rod to the next location, such that each wake survey overlapped the previous by at least 20% of the wake rake measurement length (120mm). The number of movements of the wake rake was defined by the size of the wake, a total of 2-3 movements was sufficient for the baseline aerofoil at all angles of attack measured. Figure 4.8 shows a close up of the wake rake Pitot and static tube array mounted in the wind tunnel.

![Image of wake rake with labels for static pressure tubes, total pressure tubes, and flexible pressure tubing.]

*Figure 4.8: Photo of the wake rake used during wind tunnel experiments*

### 4.1.7 Direct force and moment measurements

Direct force and moment measurements were taken using the overhead 6-component force/torque balance. The load ranges of the force balance are shown in Table 4.1.
Assuming a $2\pi$ lift curve slope and a maximum angle of attack of $6^\circ$, the maximum lift produced by the baseline model is $\sim 61\,\text{N}$, which is in the range of the balance side force load limit. For all components the nominal force/moment range is equivalent to $1\,\text{V}$ or $\pm 1\,\text{V}$ signal from the displacement transducer. A balance calibration matrix was obtained by loading each displacement transducer individually through the balance load centre, located at the centre of the tunnel test section, and correlating the voltage output to the load input. This balance calibration matrix was used to transform all balance raw voltage data to the equivalent force/moment data.

<table>
<thead>
<tr>
<th>Load/Moment</th>
<th>Drag, $F_x$</th>
<th>Side, $F_y$</th>
<th>Lift, $F_z$</th>
<th>Roll, $T_x$</th>
<th>Pitch, $T_y$</th>
<th>Yaw, $T_z$</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Range</strong> ($\text{N, Nm}$)</td>
<td>67</td>
<td>$\pm 135$</td>
<td>$\pm 220$</td>
<td>$\pm 3.5$</td>
<td>$\pm 11$</td>
<td>$\pm 3.5$</td>
</tr>
</tbody>
</table>

*Table 4.1: Wind tunnel balance force and torque nominal ranges, in force balance axes.*

### 4.1.8 Pressure measurements

Static and total pressure data was recorded from the following sources:

1. Wind tunnel throat – static pressure
2. Ahead of the wind tunnel contraction – static pressure
3. Model surface pressure tappings – static pressure
4. CFFS plenum internal pressure – static pressure
5. Wake rake – static and total pressure
6. Wind tunnel test section – total pressure

All pressure measurements except for the CFFS plenums and wing surface pressure measurement were measured using Sensortechnics HCXPM005D6H fully signal conditioned pressure transducers. These pressure transducers are limited to a maximum of $5\,\text{mbar}$. The CFFS plenum pressure was measured using the Sensortechnics
HCXM350D6 pressure transducer limited to a maximum range of 350mbar. The wing surface static pressure was measured using the HCXM020D6 pressure transducer limited to a maximum range of ±20mbar. All pressure transducers were powered by a locally regulated 5V supply.

4.1.9 Data acquisition system

Data was sampled using the National Instruments PCI-6229 card which is capable of sampling up to 80 analogue channels at a sampling rate of 250kS/s, as well as driving up to 4 analogue output channels. The PCI-6229 card was controlled through the National Instruments LabVIEW software. A series of simple programmes driven from the same user interface read all channels required. Scanivalves were driven by a 5V signal from the PCI card that switched the Scanivalve to the next pressure port after which a pressure measurement was taken before switching to the next pressure port. A delay of 1 second was left between switching ports and taking readings to allow for a settling time.

4.2 Experimental procedure

4.2.1 Phase 1 – preliminary and model baseline configuration tests

The following tests were performed with the aim of providing measurement calibration and validation:

1. Wind tunnel force balance calibration
2. Wind tunnel test-section velocity calibration
3. Wake rake calibration
4. Model baseline configuration measurements without trip strip
5. Comparison of surface pressure distribution and force balance at zero
incidence. The outcome of this test was that the model surface pressure
distribution was not symmetric at the model zero incidence. This was caused
by a slightly non-symmetric model due to manufacturing tolerances. For
results comparisons it was decided to use a zero incidence as the location that
gave a symmetric surface pressure distribution. The offset angle in this case
was ~0.6° nose up.

6. A trip strip analysis was performed to find the trip strip height that would be
just sufficient to cause a turbulent boundary layer over the model surface. This
is discussed in section 4.4.3.

4.2.2 Phase 2 & 3 – baseline and bulk testing

These phases were split into two due to time requirements for initial validation of test
measurements. The first measurements included the model at zero lift incidence in
MiGS and CFFS configurations. This allowed measurement validation before the bulk
of the testing was completed in phase 3.

4.2.3 Wind tunnel test procedure

A flow chart of the experimental procedure (shown in Figure 4.9) is described below.

1. The model was set to the required configuration and incidence. The first zero
reading (1) was taken. For the CFFS cases the pneumatic supply was turned to
the required pressure.

2. Once the required pressure had settled, a second zero measurement was taken.
This zero was taken so that the thrust of the jet could be calculated if required.
(For the baseline and MiGS tests no extra zero is required was taken). The
tunnel was then set to a test-section velocity of 24.5m/s.

3. Once the velocity was stable, the scanivalves were started at the current wake
rake location.

4. Once the Scanivalve results were obtained the wake rake was moved to the next position, and the wake rake Scanivalve was started again (3). This loop was performed until the entire model wake was recorded. The wind tunnel velocity was then turned off and allowed to settle.

5. A tunnel off zero reading was taken. For the CFFS cases then pneumatic supply was then turned off and allowed to settle.

6. A final zero reading was then taken.

![Flow chart of experimental procedure](image)

**Figure 4.9: Flow chart of experimental procedure**

For the results, the zero readings of 2) and 5) were average and subtracted from the baseline and MiGS readings to allow for bias in measurements over the testing time. For the CFFS cases the zero readings of 1) and 6) were used.

### 4.3 Data Reduction

#### 4.3.1 Boundary corrections

The flow conditions in a wind tunnel are not the same as real flight conditions because the air is bounded by walls. However, a wind tunnel aims to simulate actual flight conditions and therefore requires boundary corrections that take the effect of the walls into account. Boundary corrections for two-dimensional cases can be split into horizontal buoyancy, solid blockage, wake blockage, streamline curvature [54].
Horizontal buoyancy is the static pressure variation in the wind tunnel test section in the streamwise direction due to the thickening of the boundary layer at the walls. The horizontal buoyancy is usually insignificant for wing models, however, this variation is minimised even further in the Project tunnel due to a test-section that expands in the longitudinal axis.

The solid blockage is the ratio of the frontal area of the model to the test section area. Using the maximum model angle of attack and assuming a constant test section area the solid blockage comes out to 0.1. This is at the higher end of a typical solid blockage but still within the typical range.

The wake blockage is similar to the solid blockage but in terms of the effect of the wake of a body in the test section. Pope et al [54] provide a two-dimensional wind tunnel test example to show the impact of the solid and wake blockages. Applying the blockage corrections to a representative case was found to cause <1% change in the drag measured. Since this is small compared to other measurement uncertainties basis, corrections due to blockage effects have not been applied to wind tunnel results.

The streamline curvature refers to the alteration of the streamline curvature due to the flow around a body in the wind tunnel. If the wing chord is less than 0.7 times the tunnel height this effect on the distribution of lift may be neglected, the value in this experiment was 0.35 hence it has been neglected.

4.3.2 Aerofoil surface pressure distribution calculation

The static pressure was measured at 28 locations around the surface of the wind tunnel model at a constant spanwise location (distribution shown in Figure 4.2 c). The spanwise location was offset 1/8c from the model centre line to avoid potential
symmetry effects caused by three-dimensional structures in the wind tunnel. The pressure coefficient at each location on the wing surface was calculated \([14]\) using:

\[
C_p = \frac{P - P_\infty}{q_\infty}
\]

(4.1)

Where: \(q_\infty\), is the dynamic pressure in the test section, \(P_\infty\), is the freestream static pressure, and \(P\) is the static pressure measured on the wing surface.

### 4.3.3 Force and moment transfer

The wind tunnel force/torque balance default set up is to measure forces and moments with respect to a horizontally mounted model. The current investigation uses a vertically mounted wing which means an initial reordering of forces and moments is required:

\[
\begin{bmatrix}
A \\
Y \\
N \\
T_x \\
T_y \\
T_z
\end{bmatrix}_{\text{model axes}} =
\begin{bmatrix}
F_x \\
F_y \\
F_z \\
T_x \\
T_y \\
T_z
\end{bmatrix}_{\text{balance axes}}
\]

The drag, lift and pitching moment can then be transformed from the model axes to the wind axes by performing a rotation of alpha about the model y axis:

\[
\begin{bmatrix}
D \\
L \\
M
\end{bmatrix}_{\text{wind axes}} =
\begin{bmatrix}
N \sin \alpha + A \cos \alpha \\
N \cos \alpha - A \sin \alpha \\
T_y
\end{bmatrix}
\]

(4.2)

The balance centre is located at \(c/2\) on the model chord line. Moments at \(c/4\) are thus obtained by:
\[ M_{c/4} = M - L \frac{c}{2} \quad (4.3) \]

### 4.3.4 Aerodynamic coefficient calculated from the force/torque balance

The forces and moments in wind axes calculated from equations (4.2) and (4.3) are nondimensionalised using the wind tunnel dynamic pressure and appropriate reference length/area:

\[
C_L = \frac{L}{q_\infty c} \\
C_D = \frac{D}{q_\infty c} \\
C_{M_{c/4}} = \frac{M_{c/4}}{q_\infty c^2} \quad (4.4)
\]

### 4.3.5 Lift coefficient calculated from the surface pressure distribution

The following equation is used to calculate the lift due to model surface pressure distribution [14]:

\[
C_{NCP} = \int_{LE}^{TE} C_{Plower} \frac{dX}{C} - \int_{LE}^{TE} C_{Pupper} \frac{dX}{C} \quad (4.5)
\]

Where, lower and upper refer to the lower and upper surface of the model, and LE and TE refer to the leading and trailing edges of the model respectively.

Note that calculation of the lift coefficient from the surface pressure distribution does not take into account any lift due to shear stress (which is very small compared to the lift due to pressure).
4.3.6 Profile drag coefficient calculated from the wake survey

For a configuration without any blowing from the model the wake drag is calculated using [54]:

\[
C_{DWake} = 2 \int \left( \sqrt{\frac{P_T - P}{P_T - P_{\infty}} - \frac{P_T - P}{P_T - P_{\infty}}} \right) \frac{\gamma}{c} d\frac{y}{c} \tag{4.6}
\]

4.3.7 CFFS jet blowing coefficient calculation

The CFFS jet velocity is calculated from the plenum pressure using isentropic flow relations:

\[
V_j = a \sqrt{\frac{P_\infty}{P_{CFFS}} \left( \frac{\gamma - 1}{\gamma} - 1 \right)} \frac{2}{\gamma - 1} \tag{4.7}
\]

Where \(a\) is the speed of sound, \(P_\infty\) is the free stream static pressure, \(P_{CFFS}\) is the mean pressure in the CFFS plenum and \(\gamma\) is the ratio of specific heats of air (=1.4). The two-dimensional blowing coefficient can then be calculated using:

\[
C_\mu = \frac{\rho_\infty h_j V_j^2}{q_\infty c} \tag{4.8}
\]

Where \(\rho_\infty\) is the freestream air density, \(h_j\) is the slot height, \(V_j\) is the velocity of the jet calculated using equation (4.7), and \(c\) is the wing local chord length. This calculation does not allow for total pressure losses in the contraction and assumes a top hat velocity profile at the slot exit. In practice there will be some pressure losses and the exit profile will include a boundary layer at each side. Both these effects will mean that the calculated blowing coefficient is slightly higher than the actual delivered blowing coefficient, however the effect is likely to be small and the level of uncertainty from
ignoring these effects is considered to be acceptable. Another potential error source is the impact the wing surface pressure has on the jet velocity and therefore the blowing coefficient. For an upper surface CFFS, as the wing angle of attack increases, the pressure on the surface would decrease, causing the jet expansion and therefore an increase in jet velocity. This complex correlation between plenum pressure, surface pressure and jet velocity has been studied in a number of articles in [71]. Due to the small angle of attack range over which measurements were performed in this thesis, the impact of the surface pressure modification on the jet velocity has been assumed negligible.

### 4.3.8 Aerofoil profile drag calculation

As part of the experimental analysis it is of interest to separate the aerofoil profile drag from the drag directly due to the projected area of the spoiler. To achieve this, the MiG spoiler drag was estimated using a combination of empirical [55] and theoretical methods [56]. The empirical method is used to calculate the drag on a plate normal to a wall submerged in a boundary layer, thickness of the boundary layer thickness is calculated from laminar boundary layer theory. This approach assumes a laminar boundary layer, however the flow is turbulent. Therefore this approach is not accurate, but does allow one to estimate the impact of the MiGS drag due to the frontal area, and therefore provide a method of calculating the aerofoil profile drag. The total drag of the MiGS is equal to the drag on the MiGS within the boundary layer and the drag on the MiGS within the freestream. Figure 4.10 shows a partially submerged MiGS by a shear layer and the effective height in the freestream.
Figure 4.10: MiGS normal to a wall partially submerged by a boundary layer of thickness, $\delta_{0.99}$.

From empirical methods the effective velocity in the boundary layer is:

$$V_{BL}^2 = \frac{n}{n + 2} V_\infty^2$$  \hspace{1cm} (4.9)

Where $n = 6$, for a flat plate velocity profile constant, $V_\infty$ is the velocity in the freestream.

The force on a plate normal to a wall is equal to the force due to the boundary layer, and the force due to the freestream:

$$F_{\text{plate}} = C_{D\text{plate}} \frac{1}{2} \rho V_\infty^2 h_{\text{eff}} + C_{D\text{plate}} \frac{1}{2} \rho V_{BL}^2 \delta_{0.99}$$  \hspace{1cm} (4.10)

Where $C_{D\text{plate}}$ is the drag on the flat plate, taken from boundary layer theory, and $h_{\text{eff}}$ is the effective height of the MiGS outside of the boundary layer thickness, shown in Figure 4.10. Rearranging in terms of drag coefficient and dynamic pressure by substituting in (4.9) gives:

$$F_{\text{plate}} = C_{D\text{plate}} \frac{1}{2} \rho V_\infty^2 \left( h_{\text{eff}} + \delta_{0.99} \frac{n}{n+2} \right)$$  \hspace{1cm} (4.11)
Using the force on the plate, the drag coefficient due to the plate can be calculated:

\[ C_{D_{MiGS}} = \frac{F_{plate}}{\frac{1}{2}\rho V_{\infty}^2 c} = \frac{C_{D_{plate}} \frac{1}{2} \rho V_{\infty}^2 (h_{eff} + \delta_{0.99\frac{n}{1+n}})}{\frac{1}{2} \rho V_{\infty}^2 c} \]  

to give the aerofoil profile drag.

\[ C_{D_{MiGS}} = \frac{C_{D_{plate}} \left( h_{eff} + \delta_{0.99\frac{n}{1+n}} \right)}{c} \]  

4.3.9 Control volume analysis for CFFS wind tunnel model configuration

A control volume analysis is required to define the impact of the added momentum and mass flow from the jet of the CFFS on the total drag. The method is limited by the following assumptions:

- The freestream flow enters the control volume from the inlet on the left and leaves the control volume through the exit on the right.
- The freestream flow at the inlet and outlet is perpendicular to the inlet and outlet boundaries respectively.
- The upper and lower boundaries are parallel to the freestream flow at the upper and lower boundaries.
- The pressures at control volume boundaries are equal to the pressures in the freestream at the boundaries.
- The flow is of steady state form inside the control volume.
- Air is injected from the CFFS slot.

Following a method presented in [57], [58], Figure 4.11 depicts a control volume surrounding the aerofoil with CFFS device shown by the dotted line from \( a \) to \( i \).
The resultant force acting on the control volume is equal to the momentum variation across the control volume boundary:

\[ \sum F = \int_S \rho V \cdot dS \cdot V \tag{4.14} \]

Applying this equation in the \( x \)-direction of Figure 4.11 gives:

\[ -p_1 S_1 + p_2 S_2 + (p_j S_j)_x + R_x = - \int_i^a \rho V_1 dy V_1 + \int_h^b \rho V_2 dy V_2 - \dot{m}_j V_x j \tag{4.15} \]

The total drag of the aerofoil can be defined as the \( x \)-component of the total force acting on the aerofoil. The total force is made up of the pressure and shear stress acting on the aerofoil surface, the thrust of the jet acting on the aerofoil, and the pressure at the slot acting on the aerofoil. Rearranging equation (4.15) with respect to the drag give:

\[ D = -(R_x + \dot{m}_j V_x j + (p_j S_j)_x) = \int_i^a \rho V_1 dy V_1 - \int_h^b \rho V_2 dy V_2 - p_1 S_1 + p_2 S_2 \tag{4.16} \]

The conservation of mass gives:

\[ \int_i^a \rho V_1 dy = \int_h^b \rho V_2 dy - \dot{m}_j \tag{4.17} \]

Assuming \( p_1 = p_2 \), and using equations (4.16) and (4.17), the drag on the aerofoil is:

\[ D = \int_h^b \rho V_2 (V_1 - V_2) dy - \dot{m}_j V_1 \tag{4.18} \]

From the general assumptions of control volume analysis, the velocity at station 1 is equal to the freestream velocity, giving:

\[ D = \int_h^b \rho V_2 (V_\infty - V_2) dy - \dot{m}_j V_j \tag{4.19} \]
Converting to coefficient form by dividing through by dynamic pressure and wing area gives:

\[
C_D = C_{D\text{Wake}} - C_\mu \frac{V_\infty}{V_j}
\]  

(4.20)

Therefore the total drag is the drag from the wake survey minus a correction factor due to conservation of mass. This means the thrust of the jet is a part of the wake survey measurement. The same result is reached for circulation control aerofoils [57], [58] and injection only aerofoils [59] whose jet is in the positive drag direction.

---

**Figure 4.11: Control volume for a CFFS aerofoil**

### 4.4 Measurement validation methods

#### 4.4.1 CFFS end effects investigation

An investigation of the three-dimensional effects of the CFFS device was performed. The slot length was varied from 100mm to 400mm in 100mm intervals and measurements were taken using the force balance. The effect of slot length with change in lift is shown in Figure 4.12. For the purposes of this work the gradients of the results have been assumed within a usable tolerance. The plot shows that there are negligible slot end effects. Therefore extending the result to a full span slot requires a scaling factor based on the slot length and wing span.
4.4.2 Sampling period investigation

An investigation into the effect of the sampling period on the drag coefficient measured from the wake survey method was performed. The time period per sample taken at each total pressure port in the wake rake was increased from 3 seconds upwards until the drag calculated converged to a constant result. This investigation helps to reduce the errors in the wake survey method from potential fluctuations of the tunnel speed. The sampling period investigation was performed with the wind tunnel model in its baseline configuration at zero incidence.

Figure 4.13 shows the change in drag coefficient with increasing sampling period from 3 to 14 seconds. At sampling periods between 3 and 6 seconds the drag coefficient varies by 10%. Only above 6 seconds does the variation of the drag measurement begin to settle. Based on these results a sampling period of 10 seconds was chosen to be used for the rest of the wind tunnel testing. The same sampling period was also used for all measurements including the surface pressure distribution.
Figure 4.13: The effect of sampling period on wake survey drag coefficient measurements at a sampling rate of 1kHz.

4.4.3 Forced boundary layer transition

“Scale effects” refer to differences between the experimental and actual flight operation flow conditions. A trip strip analysis is one method of reducing the differences between these flow conditions, by tripping the surface flow from laminar to turbulent at 0.1c for all angles of attack tested. The location of transition is not fixed, but moves with angle of attack. It is assumed that a transition fixed at 0.1c is suitable for this investigation [54]. Figure 4.14 shows the results of a transition study on the model using two-dimensional or pinked tape as the transition mechanism. The process followed is as presented in Pope et al [54].

At trip strip heights of less than 0.3mm the drag coefficient increases rapidly. Beyond a height of 0.3mm any added height leads to a constant increase in drag. This indicates that boundary layer transition has been reached with a pinked tape of thickness 0.3mm. The trip strip drag correction is the delta between the (fully established transition or) chosen trip strip height to that extrapolated back to zero trip strip height. This delta corrects for the pressure drag caused by the trip strip, while the trip strip ensures
laminar to turbulent transition occurs at the proper location. Due to the discrepancy with the force balance data, a trip strip height of 0.5mm was used in all following tests.

![Graph](image.png)

**Figure 4.14**: Results of a transition study using a series of two-dimensional tape (or pinked tape) of increasing thickness located at 0.1c from the leading edge on the upper and lower surfaces of the model. The drag coefficient values shown are measured from the force balance and momentum loss in the wake.

### 4.4.4 CFFS plenum validation

Based on [44], the design objectives for the CFFS plenum were:

1. To minimise the total pressure distortion at the slot exit
2. To minimise the static pressure drop from plenum entry to plenum exit
3. To minimise the installed volume

The design was tested by measuring the static pressure distribution across the span on the internal lower surface of the plenum, shown by Figure 4.15. The four pressure tappings are equally spaced along the span of the plenum. All pressure readings are within 3% of the mean value, and therefore shows good uniformity.
4.5 Measurement uncertainty

4.5.1 An overview of measurement uncertainty

“The purpose of uncertainty evaluation is to define the result of the measurement in terms of three parameters: the mean value, the expanded uncertainty and the confidence level or coverage factor.”[60]

The method of uncertainty calculation in this thesis is based on that defined by the general metrology for measurement uncertainty provided by the British Standards Institute [60]. The following process was used to calculate the measurement uncertainty of the lift coefficient from the force balance in the following sections:

1. Calculate the standard uncertainty of a measurement based on the number of samples and standard deviation of the sample set.

\[
u(x) = \frac{s(x)}{\sqrt{n}}
\]  

\[(4.21)\]
Where, \( s(x) \), is the standard deviation of the sample and, \( n \), is the number of samples taken.

2. Obtain the expanded uncertainty of the measurement device (usually provided in a calibration file for the device) used to then calculate the calibration standard uncertainty.

\[
u_{\text{cal}}(x) = \frac{U(x)}{k}
\]  \hspace{1cm} (4.22)

Where, \( U(x) \), is the expanded uncertainty and, \( k \), is the coverage factor (1.96 for a confidence level of 95%).

3. The sample and calibration standard uncertainties are then combined to give the total standard uncertainty of the measurement.

\[
u = \sqrt{(u(x))^2 + (u_{\text{cal}}(\bar{x}))^2}
\]  \hspace{1cm} (4.23)

4. The effect of the measurement on the output quantity (e.g. lift coefficient) is based on a sensitivity coefficient. This is defined as the partial derivative of the functional relationship (the output quantity written in terms of the input quantities) between the input quantities and the output quantity.

\[
c_i = \frac{\partial f}{\partial x_i}
\]  \hspace{1cm} (4.24)

5. The combination of uncertainty due to each input quantity can then be combined, if uncorrelated using:

\[
u_c^2(Y) = c_{x_1}^2 u^2(x_1) + c_{x_2}^2 u^2(x_2) + \cdots
\]  \hspace{1cm} (4.25)
4.5.2 Uncertainty in lift coefficient measurement

The lift coefficient from equation (4.4) can be written in terms of its input quantities from equation (4.2) as:

\[ C_L = \frac{N \cos \alpha - A \sin \alpha}{q_\infty c} \tag{4.26} \]

The input quantities are the axial force, \( A \), the normal force, \( N \), and the dynamic pressure, \( q_\infty \). The following table sets out the calculation of the uncertainty associated with the force balance lift coefficient of the baseline model configuration at an angle of attack of 6 degrees. The nominal lift coefficient is 0.6197, and the calculated expanded uncertainty at the 95% confidence level is 9.15%. This is an acceptable uncertainty for the purposes of this study.
<table>
<thead>
<tr>
<th>$x_i$</th>
<th>A</th>
<th>N</th>
<th>Q</th>
<th>Combined</th>
<th>Nominal value</th>
<th>% uncertainty</th>
</tr>
</thead>
<tbody>
<tr>
<td>cu</td>
<td>2.7</td>
<td>0.00075</td>
<td>-0.000113</td>
<td>0.028919</td>
<td>-0.00075</td>
<td>$5.698e^{-7}$</td>
</tr>
<tr>
<td>c</td>
<td>0.4418</td>
<td>0.000836</td>
<td>0.01071</td>
<td>0.028919</td>
<td>-0.00075</td>
<td>$5.677e^{-7}$</td>
</tr>
</tbody>
</table>

Table 4.2: Uncertainty calculation of the force balance lift coefficient. The expanded uncertainty at the 95% confidence level for $C_L = 0.6197$ is 9.15%.
5 COMPUTATIONAL METHODS

This chapter presents the computational methods applied to obtain predictions of the two-dimensional aerodynamic performance of counterflow fluidic spoilers. The numerical solver is described along with the mesh, the case set-up and convergence metrics. Results from two validation cases are then presented, one for a macro geometric spoiler [61] and one for a normal blowing fluidic spoiler [40], where the aim is to match the surface pressure distribution due to a control input.

5.1 Computation apparatus and case set-up

5.1.1 Numerical solver

The computational fluid dynamics (CFD) code Fluent (version 6.1) was used for the numerical study. An Unsteady Reynolds-Averaged Navier-Stokes (URANS) approach was employed using the fully implicit unsteady solver and a second-order-discretisation scheme in time and space for all variables. The Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) [62] algorithm is suitable for incompressible flows [70] and was used for the pressure and velocity coupling.

5.1.2 Convergence criteria

Unsteady solution convergence was measured by monitoring the velocity at a number of locations in the unsteady flow field region, as well as the global aerodynamic force coefficients. The maximum number of inner-loop iterations (per time step) was set to 20, which was sufficient to ensure inner-loop convergence (inner step convergence criteria was set to $10^{-5}$); the computation required approximately 6,000 to 8,000 outer-loop iterations to reach a periodically converged solution.
5.1.3 Solution monitoring and convergence

To ensure efficient convergence, computations were initially obtained for steady state flow up to 1000 iterations, before then being restarted using the unsteady solver. A constant time step size of $\Delta t = 1 \times 10^{-4}$ was used, which ensures that the Courant number is less than 1 in the majority of the domain, and a maximum Courant number throughout the entire domain of around 10.

5.1.4 Turbulence model

The k-\(\omega\) Shear Stress Transport (SST) turbulence model of Menter [63] was used in all cases presented. This selection was based on numerical studies performed by Choi et al [64] who tested a range of turbulence models for flow past an inlay spoiler; the k-\(\omega\) SST provided the best agreement with experimental data.

The SST model was developed based on observations that the k-e model [65] had difficulties at the surface of a wall due to the non-zero value of epsilon (the rate of turbulence dissipation), whilst the standard k-omega model [66] was known to have an unstable dependence upon freestream turbulence levels. The SST effectively employs the k-omega model in the near-wall region, whilst switching to the k-epsilon equation away from the wall. As such, the SST model is able to be used in the near-wall viscous region without the need for additional modelling and without adversely influencing the calculation performance or stability.

The numerical meshes used have a near wall resolution sufficient to ensure that the non-dimensional wall distance, $y+$, attains a value of less than unity at all times.
5.1.5 Calculation of the lift and drag coefficient

The lift and drag forces for all cases were calculated from the surface pressure and shear-stress distribution around the aerofoil. These forces are then summed and converted to coefficient form by dividing by the freestream dynamic pressure and the chord length, as is shown in equation (4.4). For the CFFS case the momentum of the jet is taken into account when calculating the lift and drag coefficients.

5.1.6 Mesh definition

An unstructured hybrid mesh (a combination of both triangular and rectangular cells) was used in all cases in order to minimise the number of cells and therefore the computational time per iteration. A structured mesh block of regular rectangular cells was used in the immediate vicinity of the aerofoil, from the surface up to a distance of 0.3c away from the aerofoil surface, so as to provide an accurate resolution of the boundary layer, and to ensure a y+ value of less than unity at the first node away from the wall. The boundary layer refinement region was projected downstream by 1 chord in order to provide detailed resolution of the wake profile. Beyond this region, at a suitable distance downstream, an unstructured mesh was employed until the end of the domain, approximately 10 chord lengths away from the aerofoil, where the mesh ended as a circular far-field boundary split into an inlet on the upstream side and an outlet on the downstream side. These details are shown in Figure 5.1. Mesh refinement is also concentrated around the jet slot, to capture the interaction of the jet with the oncoming freestream flow.
5.1.7 Mesh independence study

In order to ensure mesh independence of the results, three different mesh resolutions were examined, as summarised in Table 5.1.

The increase in mesh density at each state was split in the perpendicular and parallel directions from the aerofoil surface in the structured mesh region. The increase in cell numbers in the unstructured mesh was aimed to be kept constant.

Figure 5.2 displays the evolution of predicted lift coefficient with numerical time, and highlights the impact of mesh refinement on the results. Both the coarse and medium mesh sizes are completely unable to capture any periodic oscillation of the separation in the wake. However, this was captured by the fine mesh, which was therefore selected to provide a reference mesh resolution for all further cases.
### Table 5.1: Overview of the important mesh parameters used in the mesh refinement study

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Number of Cells</th>
<th>Max $y+$</th>
<th>Average Courant number</th>
</tr>
</thead>
<tbody>
<tr>
<td>1: Coarse</td>
<td>100,000</td>
<td>1.6</td>
<td>~10</td>
</tr>
<tr>
<td>2: Medium</td>
<td>117,000</td>
<td>1.4</td>
<td>~10</td>
</tr>
<tr>
<td>3: Fine</td>
<td>132,000</td>
<td>1</td>
<td>~10</td>
</tr>
</tbody>
</table>

**Figure 5.2: Effect of grid refinement on aerofoil lift coefficient, for upper surface CFFS at 0.65c, with max. $C_{\mu}$ and $\alpha = 6^\circ$.**

### 5.2 Computational method validation

#### 5.2.1 Section overview

The computational procedure described above was validated against experimental data for two cases; a geometric and fluidic spoiler case as summarised in Table 5.2. Results are presented in the form of the surface pressure distribution comparison between experiment and computation. The flow was deemed to have converged when both the velocity probes downstream of the spoiler and the total lift coefficient reached periodic solutions.
Table 5.2: Validation cases conducted for the CFD process

5.2.2 Geometric spoiler case

The flow around an aerofoil with an inlay spoiler was studied experimentally by Consigny et al [61]. This was selected as a validation case since another relevant computational study also investigated this case, and so it was possible to directly compare the computational prediction [67]. The aerofoil is at zero incidence, the spoiler is deflected to 20° and the Reynolds number with respect to chord length is 1.9x10⁶.

A close up of the mesh used is shown in Figure 5.3. Note that the mesh refinement increases towards the aerofoil and spoiler surfaces in order to accurately resolve the boundary layer and flow structures in these regions. The mesh refinement downstream of the spoiler is designed to capture the unsteady oscillation of the wake resulting from the separation. The Courant number is less than unity for the majority of the flow field.
The aerofoil profile used by Consigny et al was the experimental aerofoil RA16SC, which is a rare geometry and was not obtainable during this work. As such the profile was extrapolated digitally from an electronic journal paper; for this reason it was not possible to capture the geometry of the leading edge to a satisfactorily high level of accuracy and so minor discrepancies were expected in this region (see Figure 5.4). However, since the primary aim of this validation case was to evaluate the ability of the present computational method in the prediction the separated flow resulting from the spoiler, it was deemed that this loss of accuracy in the vicinity of the leading edge was acceptable.

As shown in Figure 5.4, the computation provides a good approximation of the surface pressure distribution in the region of separated flow, a feature of the flow which has substantial impact on the overall prediction accuracy. However, there is a slight discrepancy at the spoiler hinge point (0.5x/c) on the upper surface, which was also
observed by Filippone [67]. This is most likely due to geometrical differences between the computational geometry and experimental geometry.

\[ C_p \]

\[ \delta = 20^\circ, \alpha = 0^\circ, \text{ and } Re = 1.9 \times 10^6 [61] \]

**Figure 5.4: Geometric spoiler validation case for a 16% supercritical wing with inlay spoiler, \( \delta = 20^\circ, \alpha = 0^\circ, \) and \( Re = 1.9 \times 10^6 \) [61].**

### 5.2.3 Normal blowing fluidic spoiler case

The flow around a normal blowing fluidic spoiler case at \( Re = 2 \times 10^5 \) was reported by Leopold et al [40]. With respect to the spoiler jet, normal blowing refers to blowing from an orifice/slot such that the jet is normal to the local aerofoil surface. The experimental data is fairly sparse, yet it provides sufficient information to enable a useful validation of the application of CFD to a case with a fluidic spoiler. The aerofoil section is a NACA 0018, the jet slot width is 0.0067c and located at the mid chord with a blowing coefficient, \( C_\mu = 0.48 \).

The mesh used in this validation case is shown in Figure 5.5. Notice the refinement around the aerofoil surface and particularly in the region of the jet plume. The Courant number is less than unity for the majority of the flow field except for the refinement
region around the jet orifice, where some larger values could not be efficiently avoided, reaching a maximum value of \( \sim 10 \) throughout the entire domain.

![Figure 5.5: Close-up of the mesh around the NACA 0018 aerofoil with normal blowing fluidic spoiler](image)

Figure 5.6 shows a comparison of the predicted \( C_p \) and that reported from the experiment for the normal blowing case. Initial results indicated that when the jet momentum coefficient was the same as that reported in the experiment, the numerical prediction was poor in the vicinity of the spoiler. This could be due to differences in the dimensionality between the experimental and numerical flow; the current numerical simulation assumes a fully two-dimensional jet which might be expected to be considerably more effective than that in experimental (three-dimensional) conditions. Given that the aim of this validation is to evaluate the accuracy of the current numerical approach in the application to the flow around a fluidic spoiler, it was decided to adjust the jet momentum coefficient to a value that was more representative of the flow reported experimentally. A factor of one half of the simulated jet momentum coefficient was found to provide the best agreement, and was
therefore used to obtain the predictions plotted in Figure 5.6. The figure shows good agreement upstream of the jet location; though the agreement is not quite so good downstream of the jet. In particular, the predicted flow on the upper surface of the aerofoil, downstream of the jet, indicates regions of pressure which are lower than those in the experiment. This is a fairly common occurrence in the unsteady modelling of 2D separated turbulent flow, since the additional instabilities that would be generated in the spanwise direction of a 3D flow are not able to be captured, and so a region of rotating flow is erroneously allowed to persist. So as to reduce all other possible sources of error, a large number of iterations were performed to ensure that these results are sufficiently time-averaged, and grid resolution was improved as much as possible. When comparing the experimental and numerical surface pressure distributions for the CFFS cases, in the results section of this thesis, the numerical blowing coefficient will be half of that of the experimental value (and stated in the legends of the plots).

While it has not been possible to achieve a perfect agreement with the case of a fluidic spoiler, a sufficiently accurate prediction is obtained, and the jet momentum coefficient has been calibrated for this type of flow. If one were aiming to improve the accuracy of this prediction further, one would most likely need to consider the simulation of a 3D domain, together with a more advanced turbulence simulation technique such as Large Eddy Simulation (LES). Both these measures increase computational cost considerably and have therefore not been selected for this work.
Figure 5.6: Fluidic spoiler validation case for a NACA 0018 airfoil with jet issuing from the lower surface; \( \alpha = 0^\circ \), and \( Re = 2 \times 10^5 \), experimental \( C_\mu = 0.48 \), simulated \( C_\mu = 0.24 \) [40].
6 RESULTS & DISCUSSION

The aim of this chapter is to present and discuss the results of the Micro Geometric spoiler and Counter-Flow Fluidic spoiler investigations, comparing results with conventional macro geometric spoilers and trailing edge controls where appropriate.

6.1 Comparison of baseline results

A comparison of baseline lift, drag and surface pressure between different measurement techniques, CFD and published data for the baseline model is shown in Figure 6.1. The experimental lift curve slope, Figure 6.1 a), is consistent between force balance data and integrated surface pressure data, and these measurements are also consistent with CFD data and data from the literature, providing evidence that the experimental and computational methods are suitable for the evaluation of lift. A similar comparison for drag is shown in Figure 6.1 b). In this case there is significant discrepancy between drag measurement from the force balance and the other sources. It is believe that this is due to inaccuracies in accounting for support interference and tare effects, and lack of two-dimensionality flow towards the wing tips. As a result of this discrepancy all experimental drag data presented in the following is based on wake survey measurements, apart from for the control coupling tests presented at the end of the results section where balance data had to be used for experimental expediency. The baseline surface pressure data, Figure 6.1 c) and d), show reasonably good agreement between experiment and computation, providing further evidence that the computational tools used are fit for purpose at least for the benign baseline cases.
6.2 Micro geometric spoiler results

Attention will now be focussed on presentation and discussion of the experimental force, moment and pressure results for the Micro Geometric Spoilers (MiGS). Lift, Drag and Moment data for varying spoiler deflection and chordwise location is shown in Figure 6.2. The broad conclusion from these data is that the MiGS is behaving in a similar manner to that expected from a macro scale spoiler from simple base area and camber considerations, i.e. spoiler deflection on either the upper or lower surface...
increases drag, whereas upper surface spoiler deflection reduces lift and lower surface deflection increases lift.

Focussing on the lift data, Figure 6.2 a) and b), the spoiler chordwise location makes a relatively significant difference to the control response. For the forward location the lift characteristic is nonlinear with respect to spoiler deflection, with evidence of control reversal at low spoiler deflections. This is similar to the effect by small deflection of forward located macro geometric spoilers (as seen in Figure 2.4 a). At the aft location the response is monotonic. Considering now the effect of upper or lower surface spoiler location, it can be seen that at $\alpha = 0^\circ$ (circles), the response is symmetric, as required by the symmetric geometry. For positive angle of attack the change in lift from the upper surface is always greater than the change in lift from the lower surface. However, for the forward spoiler location on the upper surface there is significant change in control response with incidence whereas for the aft position there is very little change. In summary, for use of a MiGS for lift control, the aft position has better linearity compared to forward, however the forward location is more efficient (larger gain).

Considering now the drag data, Figure 6.2 c) and d), it can be seen that deflection of a spoiler at either the upper or lower, or fore or aft location generates an increase in drag, (consistent with increasing base area and reduced downstream pressure recovery), however the magnitude of the response is much larger for the forward location. Furthermore, the magnitude of control response is proportional to alpha for the forward location, whereas at the aft location the response is approximately independent of alpha. In terms of upper/lower surface location, for the forward station the control drag gain is significantly higher for the upper surface compared to the lower surface. On the other hand, for the aft station the spoiler drag response is symmetric upper/lower. The
dotted lines in Figure 6.2 b) and c) show the total drag minus the calculated base drag of the spoilers (as detailed in the method section) as a way of illustrating the relative contribution of the aerofoil section and the spoiler itself to the production of drag. For the forward located spoiler the majority of the drag comes from the aerofoil component whereas for the aft location the majority of drag comes from the calculated base drag component. As mentioned in the research method section, the base drag estimate will be an over estimate since the reference velocity was based on the free stream rather than the local boundary layer velocity, however even with this uncertainty it can be seen that if drag is required then it may be advantageous to place MiGSs in a forward location on an aerofoil, however tests of the time response due to a control input are required to make sure the lag is below a minimum limit. In summary, for use of a MiGS for drag control, the aft location has relatively poor efficiency with the drag increment mainly due to the spoiler base drag increment. The forward location has much higher efficiency, however the magnitude of control response is strongly coupled with angle of attack (more control at higher alpha).

The MiGS generated pitching moments about the quarter chord are shown in Figure 6.2 e) and f). Of particular note is the change in sign of the pitching moment response with spoiler deflection between the fore and aft spoiler locations, which is observed in macro geometric spoilers in Figure 3.7. For the fore location, upper surface spoiler deflection produces a nose down (negative) pitching moment in association with a decrease in lift (and an increase in drag). This implies that the centre of pressure of the loading increment is aft of the quarter chord. For the aft spoiler location at low incidence the pitching moment trend with spoiler deflection is opposite to the fore location, even though the drag and lift response has the same sign as for the fore location.
Change in surface pressure distribution generated from MiGS deflection relative to the baseline is shown in Figure 6.3 at 6° angle of attack. These results demonstrate that the fundamental spoiling mechanism is based on the generation of an increase (positive increment) in pressure ahead of the spoiler and a decrease (negative increment) in pressure behind the spoiler, consistent with the established mechanism for macro geometric spoilers. Furthermore it can be seen that the magnitude of pressure change is largest for the forward located spoiler on the upper surface, consistent with observed changes in forces shown in Figure 6.2.

The efficiency of the MiGS defined in section 1.1 as the change in aerodynamic coefficient divided by the change in nondimensional spoiler height at 6° angle of attack is shown in Figure 6.4. Due to the nonlinear nature of a number of the control responses, the gain is defined by the gradient of the line at the maximum spoiler height. Comparing the efficiency of the forward and aft spoiler locations, it can be seen that the upper surface forward location provides the maximum efficiency for all aerodynamic coefficients show. The aft spoiler location provides greater consistency between the upper and lower surface.
a) Lift, MiGS at 0.35x/c. Showing $\Delta C_L$ of 20% chord flap deflected to 14°.

b) Lift, MiGS at 0.65x/c. Showing $\Delta C_L$ of 20% chord flap deflected to 14°.

c) Drag, MiGS at 0.35x/c.

d) Drag, MiGS at 0.65x/c.

e) Pitching moment, MiGS at 0.35x/c.

d) Pitching moment, MiGS at 0.65x/c.

Figure 6.2: Effect of Micro Geometric Spoiler (MiGS) location on the change in experimental lift, drag and pitching moment with spoiler height. Angle of attach range: $\circ = 0^\circ$, $\Delta = 3^\circ$, $\square = 6^\circ$. Corresponding surface pressure distribution plots (in Fig. 6.3) indicated in vertical axis ($A_{CP}$, $B_{CP}$, $C_{CP}$, $D_{CP}$).
Figure 6.3: Effect of 0.03c MiGS location on change in surface pressure distributions at $\alpha = 6^\circ$. (□) = upper surface, (Δ) = lower surface, (- -) = Spoiler chordwise location.

Figure 6.4: Gain (efficiency) of MiGS for both lower and upper surfaces, and both chordwise locations.
6.3 Counter-flow fluidic spoiler results

Force and moment data for CFFS plotted in the same format as for the MiGS is shown in Figure 6.5. The first observation is that the overall form of the lift and drag plots are broadly similar for the MiGS and the CFFS, confirming that whilst the implementation of these two types of control is very different, the fundamental control response for the different control locations is similar. In terms of differences, the forward upper surface CFFS has a monotonic lift control response with blowing coefficient, unlike the MiGS in the same location. Also, for drag, the control gain for the aft CFFS is greater than for the fore location, which is the opposite way round to the MiGS case.

In order to understand the level of ‘amplification’ generated by the CFFS the overall measured drag with the calculated jet thrust component subtracted is shown in Figure 6.5 c) and d) as dotted lines. For the aft location at higher blowing rates the fluidic gain of the system is around 2 (one unit of momentum gives two units of drag). At the forward location the fluidic gain is reduced, with a gain of less than two at higher blowing rates, and a gain approaching unity at lower blowing rates.

The moment data for the CFFS is shown in Figure 6.5 e) and f). The change in moment magnitude for a given change in lift is roughly similar between the MiGS and CFFS, however for drag, the moment coupling is significantly reduced for the aft CFFS position compared to the best case moment coupling for the MiGS. The effect of amplification is carried over from the drag plots into the pitching moment plots by the dotted line, which shows the pitching moment caused by the jet. It can be seen that the effect of the jet on the pitching moment is almost negligible compared to the pitching moment from the aerofoil surface pressure.
Pressure distributions for the CFFS are shown in Figure 6.6. Comparison with the pressure distributions for the MiGS shows that the basic pressure signature of the CFFS is similar to that of the MiGS, i.e. actuation generates an increase in pressure ahead of the device and a decrease behind. Notice, however, that the device pressure signature is projected further ahead of the device location for the CFFS compared to the MiGS. This is consistent with the difference in providing actuation through tangential momentum injection and local momentum ‘removal’ by the MiGS.

The CFD data is plotted against the upper surface blowing CFFS in plots Figure 6.6 a) and b). The simulated $C_{\mu}$ value is half of the experiment as discussed in the computational chapter. There appears to be a larger difference between the experimental and simulated CFFS case than observed in the normal blowing case study. The main difference is observed ahead of the spoiler, where surface flow separation occurs. The experimental results show a further forward projected pressure signature than the simulation. This is expected to be caused by additional instabilities generated in the spanwise direction of a 3D flow are not able to be captured in the simulated case.

The efficiency of the CFFS is shown in Figure 6.7. As with the MiGS efficiency plot the gain is defined by the gradient of the line at the maximum control input. Unlike for the MiGS, the forward located CFFS provides less overall efficiency across most of the aerodynamic coefficients than the aft location, on both the upper and lower surfaces. This difference is expected to be caused by the lack of increase in base observed in the MiGS case, such that the change aerodynamic coefficients are mainly caused by change in surface pressure distribution.
CHAPTER 6 – RESULTS & DISCUSSION

a) Lift, CFFS at 0.35x/c. Showing ΔCL of 20% chord flap deflected to 14°.

b) Lift, CFFS at 0.65x/c. Showing ΔCL of 20% chord flap deflected to 14°.

c) Drag, CFFS at 0.35x/c.

d) Drag, CFFS at 0.65x/c.

Figure 6.5: Effect of Counter-Flow Fluidic Spoiler (CFFS) location on the change in experimental lift, drag and pitching moment with blowing coefficient. Angle of attach range: ○ = 0°, △ = 3°, □ = 6°. Corresponding surface pressure distribution plots (in Fig. 6.6) indicated in vertical axis \( A_{CP}, B_{CP}, C_{CP}, D_{CP} \). \( z \) is the moment arm for the jet.
Figure 6.6: Effect of $C_{\mu}=0.05$ blown CFFS location on change in surface pressure distributions at $\alpha = 6^\circ$. (☐) = upper surface, (Δ) = lower surface, (---) = Spoiler chordwise location.

Figure 6.7: Gain (efficiency) of CFFS for both lower and upper surfaces, and both chordwise locations.
6.4 Comparison of MiGS and CFFS

In order to aid visualisation of the coupling between lift, drag and pitching moments generated by the MiGS and CFFS, Figure 6.8 and Figure 6.9 show the lift, drag and pitching moment data from Figure 6.2 and Figure 6.5 plotted as control response polars. The data presented in Figure 6.8 and Figure 6.9 is interpreted as follows. In each plot the three open circles joined by solid lines is the baseline data for zero control input at zero, three and six degrees angle of attack. The other open symbols joined by a dashed line are for increasing values of control deflection, with values as identified in the legend for each figure. The angle of attack for each data point is identified by a dotted line joining data points at constant angle of attack to a baseline angle of attack. The Lift/Drag polars in Figure 6.8 also show comparative data from data sheets for the MiGS and from CFD for the CFFS. These data are identified by filled symbols, linked to control input values in the legend.

Consider first the Drag/Lift polars, Figure 6.8. Linking with the discussion from the results presented on control response as a function of control input, it can be seen that for both MiGS and CFFS control input always produces a positive increment in drag (point on lines of constant alpha always move upwards with increasing control deflection), however the sign of the lift change depends on control upper/lower location, with upper surface controls reducing lift (points on lines of constant alpha move left) and lower surface controls increasing lift (points on lines of constant alpha move right). Comparison with extrapolated data sheet [2] values for small deflections of a ‘macro’ geometric spoiler at 0.35c and 0.65c with the MiGS are shown in Figure 6.8 a ii) and Figure 6.8 a iv). For the forward case (0.35c) the agreement between the present experimental data and the data sheet is reasonably good. At the aft location (0.65c) the agreement is less good, with the data sheet values considerably over
predicting the drag. This is not unexpected since the data sheet methods do not take into account the loss in spoiler effectiveness due to boundary layer emersion at small deflections. Comparison of upper surface CFFS CFD results with experimental data are given in Figure 6.8 b ii) and Figure 6.8 b iv). The CFD trends with increasing blowing are consistent with the experimental data, i.e. blowing reduces lift and increases drag. However, for the forward spoiler locations CFD over predicts both the reduction in lift and the increase in drag for a given blowing coefficient, whereas for the aft spoiler the CFD over predicts lift and under predicts drag. Comparing the gradients of the L/D CFD and experimental data, shows very similar results for the aft location but not for the fore location. This discrepancy in CFD and experimental data at the fore location was investigated and initially thought due to improper resolution of the separation location ahead of the jet. A number of increasingly dense meshes were used to capture this, during which the monitoring of velocity at a number of points showed no period excitation was captured. The forward mounted CFFS device causes the leading edge stagnation and stagnation point ahead of the CFFS to be much closer than in the aft case. This is thought to cause a higher frequency separation relative to the aft CFFS and beyond the ability of the URANS approach.
Figure 6.8: Experimental lift-drag polar plots compared with data sheet (MiGS) and CFD (CFFS) results

Figure 6.9: Experimental pitching moment – lift polar plots
Consider now the moment/Lift polars shown in Figure 6.9. The data presented in this form confirms that the pitching moment response of a forward mounted MiGS and CFFS are similar, however there are a noticeable few differences. A lower surface mounted CFFS produces very little pitching moment for a given lift compared to the MiGS. Also, unlike the MiGS the upper surface forward mounted CFFS of Figure 6.9 b ii) produces a negative pitching moment of a similar order to the aft location, at all lift coefficients tested. The pitching moment response from both lower surface locations are equivalent, implying that the change of pitching moment due to CFFS actuation is much less sensitive to chordwise location than the MiGS.

6.5 Upper and lower surface CFFS blowing

The final set of data explores the simultaneous use of both an upper surface and lower surface CFFS, with the aim of understanding the potential for use of dual CFFS devices for control of drag at constant lift and pitching moment. It was decided to use an upper surface CFFS at the forward location and a lower surface CFFS at the aft station based on the control authority demonstrated in these positions when used exclusively (only one control operating at a time). Since there is a fair degree of decoupling between the pressure distribution on the controlled side and the pressure distribution on the opposite (uncontrolled) side, it is hypothesized that the control effect from non exclusive actuation (both controls working at the same time) will be similar to the sum of the exclusive effects. Measurement of drag, lift, and moment were obtained from the wind tunnel force balance for varying blowing through both the upper and lower CFFS devices for an angle of attack of zero and six degrees.

Figure 6.10 shows contour maps of the drag data plotted using the lower surface blowing as the x axis and upper surface blowing as the y axis. Overlaid on top of this
are loci of upper and lower surface blowing for zero change in lift and zero change in pitching moment, labelled $\Delta C_M=0$ and $\Delta C_L=0$, respectively. Thus by choosing blowing control input pairs that correlate with either of the two loci it is possible to generate a finite drag with either zero change in pitching moment or zero change in lift, which is the desired result from a control independence point of view. As it happens, the pitch and lift loci are approximately overlaid for both zero and six degrees alpha, so following either loci means that drag can be obtained with zero change in pitching moment and zero change in lift. Whilst this is a fortuitous result, it arises because for the present configuration there is proportionality between control lift and control pitching moment; hence if either is driven to zero then the other will be zero also. Finally, in comparing the achievable drag control at zero and 6 degrees alpha, it can be seen that the magnitude of drag obtainable is significantly greater at the higher angle of attack; however, these high values of drag cannot be obtained along the zero change in pitching moment and lift loci. Indeed, the maximum achievable drag with no pitch and lift coupling is less for the six degrees angle of attack case compared to the zero angle of attack.
Figure 6.10: Force balance drag coefficient from dual surface blowing for yaw control. Upper surface CFFS = 0.35x/c. Lower surface CFFS = 0.65x/c. Zero pitching moment ($C_M$) and lift ($C_L$) loci indicated.
7 CONCLUSIONS & FUTURE RESEARCH OPPORTUNITIES

7.1 Conclusions

Micro geometric spoilers (MiGS) and Counter-flow fluidic spoilers (CFFS) can be placed in a similar class of spoiler, ‘low form factor spoiler’, due to their similar installed volume and influence on the surrounding flow field.

- A MiGS is defined as a device whose deployed length scale is of a similar order to the local boundary layer thickness at the point of operation. This is contrast to conventional 'macro geometric spoilers' where the deployed length scale is much larger than the local boundary layer thickness. The MiGS acts as a barrier to the smooth near wall flow effectively reducing the momentum in the co-flow direction.

- A CFFS is distinct from other fluidic control implementations that use tangential blowing in that the blowing direction is perpendicular to the leading edge and in opposition to the local freestream direction and that the control is placed on the upper or lower surface away from the leading and trailing edges. This is significant in that leading and trailing edges are typically already highly constrained areas of real estate on a lifting surface in terms of geometry and systems placement and hence the capability to implement new flight controls outside these areas is advantageous. The CFFS acts as a barrier to the smooth near wall flow by addition of momentum in the counter-flow direction.
A simple qualitative comparison has shown that a typical rearward mounted spoiler can produce similar changes in the aerodynamic coefficients as camber modifications. Geometric and fluidic spoilers have also been shown to have broadly similar flow topology features apart from a number of detailed differences.

- An upper surface rearward mounted spoiler or negative camber modification cause a reduction in lift, an increase in drag and an increase pitching moment. A lower surface rearward mounted spoiler or positive camber modification cause an increase in lift and increase in drag and a reduction pitching moment.

- Both geometric and fluidic spoilers cause a large recirculating region that reattaches to the surface or stays separated depending on the lifting surface configuration and flow field conditions. Both geometric and fluidic spoilers have small recirculating regions similar to those termed “hinge bubbles” just ahead and behind the spoiler base or slot, apart from the CFFS, whose large recirculating region is entrained by the jet such that there is no minor aft recirculation.

Relatively small scale, low speed two dimensional wind tunnel experiment has been shown suitable to obtain the two-dimensional qualitative aerodynamic performance of MiGS and CFFS at low angles of attack.

- Measurements were taken from an overhead mounted force balance, surface pressure tapping’s and wake survey. The force balance drag measurements captured three-dimensional flow characteristics for the baseline aerofoil
configuration therefore the wake survey drag was used for subsequent drag data except where stated.

- A control volume analysis of the wind tunnel test section containing CFFS model configuration has shown that a correction related to the additional mass flow of the jet is required to obtain the actual drag coefficient, similar to that observed in other fluid injection device experiments.

CFD can provide realistic results for aerofoils at low angles of attack with deployed geometric spoilers. However, accurate simulation of fluidic spoiler devices using URANS is limited.

- CFD analyses of geometric surface configurations, such as baseline wind tunnel model and macro geometric spoiler configurations from literature have shown good agreement of both surface pressure distributions and global aerodynamic coefficients.

- Accurate correlation between CFD and experimental results for the CFFS cases was only found when the calibration factor used in the fluidic spoiler validation case was used. The use of a calibration factor is considered acceptable practice on the basis that the effective discharge coefficient of the blowing slot is unknown and hence there needs to be some correction applied to the blowing coefficient used in the CFD and the actual blowing coefficient delivered by the experiment when the experimental blowing coefficient is derived from measurement of plenum pressure.

- It was also found that for the forward CFFS location using the methods described; the CFD methodology used could not qualitatively predict the global
aerodynamic coefficients. Comparing the CFD results to the experimental results, it is clear that the good qualitative agreement is found for the aft CFFS case; however there are differences in the forward location results. A mesh refinement study failed to capture the periodic oscillation in the flow field, therefore it is perceived this discrepancy is most likely due to a limitation of URANS in capturing the high frequency oscillations that may dominate the flow field.

Experimental results combined with understanding of flow field topology from CFD shows that the fundamental fluid mechanism for both MiGS and CFFS is similar, however there are some differences.

- Both the MiGS and CFFS generate an adverse pressure gradient ahead of the actuation mechanism which causes an upstream flow separation. The location of this separation is approximately proportional to the control input, that is, larger MiGS deflection or increased blowing from the CFFS shifts the separation point forward. The static pressure downstream of the spoiler generally decreases (becomes more negative) with increasing spoiler input, however, the magnitude of the change is generally smaller than pressure change ahead of the device. This overall mechanism is similar to the way in which macro spoiler devices work at small deflection angles.
- For an aerofoil at a positive lift coefficient, spoiler input on the upper surface reduces the lift and increases the drag, whereas a spoiler on the lower surface decreases lift and increases drag.
- The fore/aft location of the spoiler devices has important effects on the device
control response and these effects are different for the MiGS and CFFS. For the MiGS, the greatest lift and drag control gain (efficiency) is for a forward located device on the upper surface, however the control response varies considerably with angle of attack. This coupled with the potential lag in control response observed in the early development of spoilers makes control implementation problematic. For the CFFS, the greatest lift and drag gain is for the aft location and the control response is reasonably independent of angle of attack. The pitching moment generated by the MiGS is nose down for the forward location and nose up for the aft location on both upper surfaces. For the CFFS, the pitching moment is generally nose down for all locations.

- By placement of a CFFS device on the upper and lower surfaces and use of simultaneous blowing from both devices it is possible to generate finite changes in drag with zero change in both lift and pitching moment. This potentially simplifies the implementation of spoiler devices into a yaw control scheme based on lateral differential drag control.

### 7.2 Future research opportunities

The work presented in this thesis combined the micro geometric spoiler and counter-flow fluidic spoiler in to the low form factor spoiler class. The two-dimensional aerodynamic performance of these two devices has been evaluated and shown to provide similar effectiveness to small deflections of macro geometric spoilers and trailing edge devices. Following on from this thesis, a number of areas may benefit from further work.

- The URANS methodology applied in this thesis has shown good agreement with experiment for aerofoils with deflected geometric spoiler, however, the
fluidic spoiler has proven more difficult to simulate. In particular two areas require further study, 1) the blowing coefficient calibration factor required to match simulated surface pressure distribution with experimental results, and 2) the ability to capture the period excitation within the flowfield due to the fluidic spoiler in the fore (0.35x/c) location separating the flow near the leading edge. 3D URANS and LES simulations investigating these issues could lead to a greater understanding of the limitations of 2D URANS simulations in predicting these complex flow structures.

- This thesis has evaluated low form factor devices in two-dimensions, however the flow over an aircraft wing is rarely two-dimensional. Therefore an understanding of the three-dimensional performance of low form factor devices is necessary before application to a flight vehicle. This could be achieved by applying the devices to a swept wing with and without end plates, to distinguish between wing sweep effects and end effects.

- Both the MiGS and CFFS are susceptible to interaction with a wide range of radar frequencies due to the discontinuity in the aerodynamic mould line of the wing. The MiGS discontinuity consists of a high aspect ratio physical spoiler, whereas the CFFS discontinuity consists of a high aspect ratio slot in the wing surface. The micro geometric spoiler requires an increase in surface area to increase effectiveness, which in turn will increase radar signature. However, the counter-flow fluidic spoiler has a fixed discontinuity and therefore a fixed radar signature at all actuation conditions. Therefore the CFFS may lend itself to low observable applications. Potential areas of further work are:
  - The radar signature of the device could be reduced by using a series of shorter slots, or holes, instead of a single high aspect ratio slot.
However, this may cause unwanted effects, such as a reduction of efficiency or effectiveness.

- The radar signature of the device could be minimised, when it is inactivate, by having a slot opening/closing mechanism. Potential solutions are a mechanical slot lip actuator and a flexible slot lip. A mechanical actuator could act as a valve or actuation mechanism for the CFFS, allowing the pneumatic system to be fully pressurised and therefore minimal lag in the device activation. A flexible slot lip could be designed to flex under certain pressure difference conditions between the local freestream and the plenum.

- The macro geometric spoiler located towards the leading edge has been shown to increase the lag of the aircraft response due to spoiler deployment. Moving the spoiler further towards the trailing edge reduced the lag but also reduced the effectiveness. A thorough understanding of the lag of the aircraft response to a control input is beneficial before flight investigations take place. An investigation of the lag due to a MiGS and CFFS is suggested through the use of a dynamic load recording wind tunnel investigation.

- Flapless flight refers to the ability of a aircraft to perform a full flight operation without the use of geometric control surfaces. This can be performed by the use of fluidic devices for aircraft control such as circulation control and fluidic thrust vectoring. If the pneumatic system of these fluidic devices relies on engine bleed, high control authority but low throttle phases of flight, such as the approach and landing phase, may cause issues. A CFFS system could provide a method which allows maximum throttle and therefore high control authority at these flight phases.
Technology transition to industry:

There is a known lack of transition or follow through of research from academia to industry. This is due to academic studies residing in a much lower technology readiness level (TRL) than that of industry. To bridge this gap a number of institutions and private companies are creating collaborative projects, such as the FLAVIIR program, driven by BAE Systems. With this in mind, and to provide an additional perspective to the further research opportunities section, the following table presents the relevant steps required to take low form factor technologies, in particular the CFFS device, from its current TRL of 2/3 to a TRL of 6. Each level has been targeted separately, presenting the performed or required capability for a CFFS device at that level:

<table>
<thead>
<tr>
<th>TRL</th>
<th>Level Description</th>
<th>Capability shown/required</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Basic principles identified</td>
<td>A novel spoiler type fluidic device consisting of a low installed volume plenum that when pressurised exhausts a thin counter-flow wall jet from a slot within the external boundaries of an aerodynamically contoured body. The thin counter-flow wall causes surface flow separation and therefore changes to the global aerodynamic coefficients. The device is termed a counter-flow fluidic spoiler (CFFS).</td>
</tr>
<tr>
<td>2</td>
<td>Technology capabilities understood</td>
<td>A two-dimensional experimental (wind tunnel test) and numerical (URANS CFD) investigation has been performed to identify the aerodynamic performance of the CFFS. A micro geometric spoiler has been tested to compare effectiveness and control response. Where appropriate comparisons with trailing edge devices have been made.</td>
</tr>
<tr>
<td>3</td>
<td>Element feasibility demonstrated</td>
<td>The CFFS system was designed with application to a low cost UAV in mind. The plenum housing is manufactured from aluminium for low weight, the plenum top plate from steel for accurate jet slot definition. A single pneumatic supply inlet is used at the tip of the plenum. Blowing coefficients used in the wind tunnel testing are comparable with those used on a Circulation Control system designed for a low cost UAV.</td>
</tr>
<tr>
<td>4</td>
<td>Component feasibility</td>
<td>3 main steps are required to gain a broader understanding of the aerodynamic performance of a CFFS device for</td>
</tr>
<tr>
<td>Demonstrated</td>
<td>Flight control, this includes:</td>
<td></td>
</tr>
<tr>
<td>--------------</td>
<td>--------------------------------</td>
<td></td>
</tr>
<tr>
<td></td>
<td>1. Effectiveness near stall angles</td>
<td></td>
</tr>
<tr>
<td></td>
<td>2. Wing tip effects</td>
<td></td>
</tr>
<tr>
<td></td>
<td>3. Effect of increasing wing sweep</td>
<td></td>
</tr>
<tr>
<td></td>
<td>4. Effect of yaw</td>
<td></td>
</tr>
<tr>
<td></td>
<td>5. Speed</td>
<td></td>
</tr>
<tr>
<td></td>
<td>6. Altitude</td>
<td></td>
</tr>
<tr>
<td></td>
<td>7. Scale effects</td>
<td></td>
</tr>
<tr>
<td></td>
<td>8. Environmental effects</td>
<td></td>
</tr>
<tr>
<td></td>
<td>9. Safety</td>
<td></td>
</tr>
<tr>
<td></td>
<td>10. Qualification</td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>5</th>
<th>Major component capability demonstrated</th>
<th>This would include:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Application of an optimised CFFS device to a representative flight ready pneumatic system, either a pressurised cylinder or engine bleed.</td>
<td>1. Application of an optimised CFFS device to a representative flight ready pneumatic system, either a pressurised cylinder or engine bleed.</td>
</tr>
<tr>
<td></td>
<td>Using a wind tunnel, investigate the aerodynamic performance of a CFFS device applied to a low cost UAV system to meet specific flight control performance requirements.</td>
<td>2. Using a wind tunnel, investigate the aerodynamic performance of a CFFS device applied to a low cost UAV system to meet specific flight control performance requirements.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>6</th>
<th>Integrated component capability demonstrated</th>
<th>Flight testing of a CFFS device integrated into a low cost UAV system.</th>
</tr>
</thead>
</table>
8 REFERENCES


REFERENCES


[56] Schlichting, H., Gersten, K., Boundary Layer Theory, 8th ed. Springer.


[68] *Increments in aerofoil lift coefficient at zero angle of attack and in maximum lift coefficient due to deployment of a plain trailing-edge flap, with or without a leading-edge high-lift device, at low speeds*. ESDU, data sheet 94028, 1994.


[70] *FLUENT 6.3. users guide*.

9 Appendix

9.1 Change in lift due to the deflection of a plain flap

The theoretical lift due to the deflection of a plain flap has been compared with the change in lift caused by the micro geometric spoiler and counter-flow fluidic spoiler in the results section. The theoretical and empirical working used to obtain the change in lift due to a flap deflection presented here is based on that presented in ESDU [68]. Figure 9.1 shows a schematic of a two-dimensional aerofoil and plain trailing edge flap to which the theory refers. This theory extends into three-dimensional applications, however, only the two-dimensional case is required in this study. The increment in lift due to a flap deflection at zero lift angle of attack is:

\[ \Delta C_{L0t} = 2J_p \xi_{flap} a_t \] (27)

Where \( J_p \) is an empirical constant based on aerofoil geometry, \( \xi_{flap} \) is the flap deflection and \( a_{flap} \) is:

\[ a_{flap} = \pi - \cos^{-1}\left(\frac{2c_{flap}}{c'} - 1\right) + \left[1 - \left(\frac{2c_{flap}}{c'} - 1\right)^2\right]^{1/2} \] (28)

Where \( c_{flap} \) is the flap chord length, and \( c' \) is the mean aerodynamic chord length.

The limiting parameters of the theory are shown in the following table. The predicted data is given to within ±20% of experimental data. Therefore the application as a comparison with micro geometric and fluidic spoilers is justified for this study.
<table>
<thead>
<tr>
<th>Parameter</th>
<th>Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thickness to chord ratio, t/c</td>
<td>0.06 to 0.18</td>
</tr>
<tr>
<td>Flap chord to chord ratio, c_{flap}/c</td>
<td>0.2 to 0.5</td>
</tr>
<tr>
<td>Reynolds number based on chord length</td>
<td>2.17 to 6 \times 10^6</td>
</tr>
<tr>
<td>Mach number</td>
<td>0.09 to 0.15</td>
</tr>
</tbody>
</table>

*Table 9.1: Parameter ranges for test data for plain trailing-edge flaps*

*Figure 9.1: Geometric definition of an aerofoil with a plain flap control surface for the determination of the theoretical lift coefficient*