Aerodynamics of battle damaged finite aspect ratio wings

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

Additional Information:

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

Metadata Record: https://dspace.lboro.ac.uk/2134/10736

Publisher: © M.Samad-Subaeb

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

For the full text of this licence, please go to: http://creativecommons.org/licenses/by-nc-nd/2.5/
Please note that fines are charged on ALL overdue items.
Aerodynamics of Battle Damaged
Finite Aspect Ratio Wings

by
Ir. Mujahid Samad-Suhaeb, M.Sc.

A Doctoral Thesis
Submitted in partial fulfilment of the requirements for the award of
Doctor of Philosophy of Loughborough University
– 2005

© by M.Samad-Suhaeb, 2005
Abstract

When an aircraft is aerodynamically or structurally damaged in battle, it may not be able to complete the mission and the damage may cause its loss. The subject of aircraft battle survivability is one of critical concern to many disciplines, whether military or civil.

This thesis considered and focused on Computational Fluid Dynamics (CFD) predictions and experimental investigations into the effects of simulated battle damage on the low-speed aerodynamics of a finite aspect ratio wing.

Results showed that in two-dimensional (2D) and three-dimensional (3D) CFD simulations, Fluent’s® models work reasonably well in predicting jets flow structures, pressure distributions, and pressure-coefficient Cp’s contours but not for aerodynamic coefficients. The consequences were therefore that CFD prediction was poor on aerodynamic-coefficients increments. The prediction of Cp’s achieved good agreement upstream and near the damage hole, but showed poor agreement at downstream of the hole. For the flow structure visualisation, at both weak and strong jet incidences, the solver always predicted pressure-distribution-coefficient lower at upstream and higher at downstream. The results showed relatively good agreement for the case of transitional and strong jet incidences but slightly poor for weak jet incidences.

From the experimental results of Finite Wing, the increments for Aspect-ratio, AR6, AR8 and AR10 showed that as damage moves out towards the tip, aerodynamic-coefficients increments i.e. lift-loss and drag-rise decreased, and pitching-moment-coefficient increment indicated a more positive value at all incidence ranges and at all aspect ratios. Increasing the incidence resulted in greater magnitudes of lift-loss and drag-rise for all damage locations and aspect ratios. At the weak jet incidence 4° for AR8 and in all of the three damage locations, the main characteristics of the weak-jet were illustrated clearly. The increments were relatively small. Whilst at 8°, the flow structure was characterised as transitional to stronger-jet. In Finite Wing tests and for all damage locations, there was always a flow structure asymmetry. This was believed to be due to gravity, surface imperfection, and or genuine feature. An ‘early strong jet’ that indicated in Finite Wing-AR8 at ‘transitional’ incidence of 8°, also indicated in two-dimensional results but at the weak-jet incidence of 4°.

For the application of 2d data to AR6, AR8, and AR10, an assessment of 2d force results led to the analysis that the tests in the AAE’s Low Turbulence Tunnel for 2d were under-predicting the damage effects at low incidence, and over-predicting at high incidences. This suggested therefore that Irwin’s 2d results could not be used immediately to predict three-dimensional.

Keywords: Battle Damage, Survivability, Jets-in-crossflow, Cavity, Weak-jet, Strong-jet, Low Speed Wind Tunnel, Force Measurement, Surface Pressure Measurements, Flow Visualisation, Finite Aspect Ratio Wing, CFD, Fluent®.
Acknowledgement

Many thanks are presented to my supervisor Dr Peter M.Render for his support, advice and guidance on my experimental work as well as for encouraging me to finish my PhD thesis during the most difficult moments of the research. In addition, I am deeply grateful to Dr Zhiyin Yang, my CFD work supervisor for his careful reading, judgement, and helpful comments. These two supervisors, to whom, apart from dedicating an important part of their time to the supervision of my thesis, has provided me with their excellent vision to conduct investigations and concentrate on the important matters.

This full-time research would not have been possible without generous financial support by Aeronautical and Automotive Engineering Department of Loughborough University during my PhD studies. Thanks to Dr Gary J. Page – Director of Research.

Distinctive gratitude is also paid to Rob Hunter - Shop/Wind Tunnel Technician and Peter Stinchcomb – Wood/Composite Shop Technician, whom participated in the process of the model design and manufacturing. The contribution by Grenville, Pradeep and Jeff are very much appreciated, thanks to their technical support during part of the experimental works, as well as their personal attention.

The support by all my colleagues from the Aerodynamics/CFD group of the Aeronautical and Automotive Engineering is highly appreciated. I would like to thank Indi H. Tristanto and Guru S.Johl for their personal and technical support.

I owe a special distinction to Ida, who decided to follow me to Loughborough, staying by my side, giving me all her love, taking care of me and my daughter Ainun. She also gifted me a baby boy Yusuf. I am infinitely thankful to my father Puang who passed away during my study. This thesis is dedicated to him. Also, to my stepmother Ummy, my son Naldy, and all my brothers and sisters to whom I have been in contact practically every fortnight since I studied in Loughborough, and who have given me their unconditional love, advice, care and emotional support in the most difficult moments.

I would also like to express my gratitude to the rest of my family and my friends in UK/Indonesia who have never forgotten me and have made me feel closer to home -- thanks to their phone calls, letters, emails and visits to Loughborough.

Mujahid Samad-Suhaeb, 2005
# Contents

Abstract  
Acknowledgement  

## Chapter-1: Introduction

1.1 Background  
1.2 Survivability, Susceptibility and Vulnerability of Military Aircraft  
1.3 Survivability of Military Aircraft Design  

## Chapter-2: Literature Survey

2.1 Studies Related to Aerodynamics of Battle Damage: Modelling, Simulation, Methods and Testing Techniques  
2.2 General Definition of ‘Weak-jet’ and ‘Strong-jet’  
2.3 Evolution and Studies Related to ‘Jets-in-Crossflow’  
  2.3.1 Jet in Cross Flow [JICF]: Empirical Model  
2.4 Research Outline and Objectives  

## Chapter-3: 3D Experimental Design and Set-up

3.1. Introduction  
3.2. Model Design and Manufacture  
  3.2.1 Mounting Structure  
  3.2.2 Wing Geometry, Design and Mounting Arrangement  
  3.2.3 Wing Manufacture  
  3.2.4 Construction of Wing Panels for Pressure Measurement  
3.3 Pre Tests Verification and Assessment  
3.4 Testing Techniques  
  3.4.1 Surface Flow Visualisation  
    3.4.1.1 Flow Visualisation Techniques and Methodology.  
    3.4.1.2 Selection of Aspect ratio and Incidence Range  
  3.4.2 Aerodynamic Force Measurement  
  3.4.3 Wind Tunnel Correction  
  3.4.4 Experimental Data Validation, Accuracy and Repeatability  
    3.4.4.1 Force Tests Hysteresis and Repeatability  
    3.4.4.2 Baseline [datum] Selection  
    3.4.4.3 Baseline Undamaged Configuration  
  3.4.5 Pressure Data Accuracy: Pressure Distribution Repeatability
Chapter-4: Review of ['2d'] Results Compared with CFD Simulation

4.1 Introduction

4.2. The Geometry and Grid Generation

4.3 Grid Refinement Study

4.4. Selection of Turbulence Modes for CFD Simulation

4.5. Testing of K-e and SA Models

4.6. CFD Simulation vs. ‘2D’ Experimental Results: The Synergy

4.6.1. Introduction

4.6.2 CFD Simulation vs. ‘2D’ Experimental Results Comparison

4.6.2.1 Undamaged Wing: Aerodynamics Coefficients Comparison

4.6.2.2 $C_\alpha$ – $C_D$ Plots for CFD Prediction vs. Experimental

4.6.2.3 The increments Comparisons: $d[C_i]_{exp}$, $d[C_D]_{exp}$, $d[C_m]_{exp}$

4.6.2.4 Pressure Distribution: -$C_D$ vs. $X/C$ Comparisons

4.6.2.5 $C_D$ Contour Comparison

4.6.2.6 Flow Visualisation Comparison

4.6.3 Discussion of The Synergy Between the CFD and Experiments

4.7 ‘Cavity – Jet in Crossflow’ Interaction Phenomena:
Further Analysis on Battle Damaged Wing CFD Prediction

Chapter-5: CFD Simulation of Battle Damaged Finite Aspect Ratio Wing

5.1. The Grid and the Solver

5.2. Aerodynamics Coefficients Prediction

5.2.1 The increments Predictions: $d[C_L]_{f6}$, $d[C_D]_{f6}$, and $d[C_m]_{f6}$

5.3. Prediction of $C_D$ at Root, Mid and Tip Damaged Locations.

5.3.1 Case-1: Undamaged vs. Damaged Hole Centreline Prediction

5.3.2 Case-2: $C_D$ vs. $X/C$: Spanwise Variations

5.3.3 Case-3: $C_D$ vs. $X/C$: Effect of Incidence at 4°, 8°, and 10°

5.3.4 Case-4: $C_D$ vs. $X/C$: Profiles Asymmetry [Inboard vs. Outboard Rows]

5.4. ‘Weak-jet’ and ‘Strong-jet’ Visualisation

5.5 2D vs. 3D CFD Predictions Comparison

5.6. Empirical Model Application on Battle Damaged Wing

Chapter-6: An Experimental Investigation of Single Hole Damage of Finite Aspect Ratio Wing

6.1. Changes of aerodynamics coefficients due to Damage

6.2. The Increments and Trends of AR6, AR8 and AR10

6.2.1 ‘$d[C_i]$’: The Lift – Loss

6.2.2 ‘$d[C_D]$’: The Drag Increment

6.2.3 ‘$d[C_m]$’: The Changes of Pitching Moment.

6.3 Effect of Tip Damage on AR6, AR8, and AR10
6.4 Single Hole Damage: Finite Wing – AR8 Flow Visualisation at the Tip, Mid and Root Damaged Locations

6.4.1 'Weak-jet' at Incidence 4°

6.4.2 'Transition to Strong jets' at Incidence +8°

6.4.3 'Strong-jet' at Incidence 10°

6.5 Finite Wing – AR8 Pressure Measurement

6.5.1 Damage Centreline and Spanwise Pressure Data

6.5.1.1 Tip Damage

6.5.1.2 Mid-Damage

6.5 Mid Damage AR8 vs. Irwin’s-2D

6.6.1 Flow Visualisation Results Comparison

6.6.2 Pressure Analysis Based on Flow Structures Visualisation

Chapter-7: Comparison Analysis of 3D CFD Simulation and 3D-Experimental Investigation

7.1 Introduction

7.2. The increments Comparison:

\[ \Delta C_L \mid \text{bd} - \Delta C_D \mid \text{bd}, \Delta C_L \mid \text{Mid-Exp} - \Delta C_D \mid \text{Mid-Exp}, \Delta C_M \mid \text{bd} \text{ Exp} \]

7.3 Pressure Profile Comparison

7.3.1 Profiles Comparison at Weak and Strong Jets Incidences

7.3.2 Spanwise Distance Profiles Comparison [*1R* Inboard/Outboard]

7.3.3 Profiles Comparison of Tip, Mid and Root Damage

7.4 Weak and Strong Jets Visualisation Comparison - Case: Mid Damage – AR8

7.5 JICF, Empirical Model, and Cavity Phenomena into CFD and Experimental Results of Battle Damaged Wings

7.5.1 Introduction

7.5.2 JICF, Empirical Modelling and JICF-Cavity Phenomena on Battle Damaged Wings

7.5.3 JICF, Empirical Model, and Cavity correlation into CFD and Experimental Results of Battle Damaged Wings

Chapter-8: Conclusions

Chapter-9: Recommended Further Work

References

Appendices:

A. Airfoil and Wing Geometries including Published Data

B. General Assembly of Experimental Rig

C. Hardware Arrangements and Schematic of Data Acquisition System

D. Applied Formulas and Equations for Wall Correction

E. Sample of Battle Damaged Wing Empirical Calculation
NOMENCLATURE

- $a_i$: Lift curve slope = $\frac{\delta C_l}{\delta \alpha}$
- $a_0$: 2D Lift curve slope = $\frac{\delta C_l}{\delta \alpha}$
- $A$: Cross-sectional area of wing section
- $AR$: Aspect Ratio ($=\frac{4s^2}{S}$)
- $AT$: Transform quantity ('ESDU' program for wing planform identification)
- $b$: Wind-tunnel working-section dimension in y-direction
- $b'$: $b/2$
- $c$: Working-section cross-sectional area ($c = 4b'h'$ for rect. Cross-section, $c = 4b'h'(1-0.5f)$ for cross-section with corner fillets)
- $C$: Chord (mm) or mean chord = $\int_c c(\eta)d\eta$, or local wing chord ($= c(\eta)$)
- $C_d$: Drag coefficient - 2D
- $C_D$: Drag coefficient - 3D
- $CD_i$: Induced drag
- $CD_o$: Minimum 'profile' drag
- $C_l$: Lift coefficient - 2D
- $C_L$: Lift coefficient - 3D
- $C_{lw}$: Zero-lift-incidence
- $C_{LL}$: Sectional lift coefficient ($= C_{ll}(\eta)$)
- $C_m$: Pitching moment coefficient (about $c/4$, Positive nose up) - 2D
- $(C_m0i)_{o}(C_m0i)_{n}$: Pitching moment coefficient if the camber line is unchanged across the span
- $C_M$: Pitching moment coefficient (about $c/4$, Positive nose up) - 3D
- $C_n$: Yawing moment coefficient
- $C_p$: Pressure coefficient
- $C_r$: Centre-line or root chord
- $C_t$: Rolling moment coefficient - 3D
- $C_t$: Tip-chord
- $C_r$: Side-force Coefficient
- $d$: Damage diameter (mm) - 2D
- $D$: Damage diameter (mm) - 3D
- $D$: Drag
- $d[C_d]$: Change in drag coefficients due to damage (at a given incidence) - 2D
- $d[C_D]$: Change in drag coefficients due to damage (at a given incidence) - 3D
- $d[C_l]$: Change in lift coefficients due to damage (at a given incidence) - 2D
- $d[C_L]$: Change in lift coefficients due to damage (at a given incidence) - 3D
- $d[C_m]$: Change in pitching moment coefficient due to damage (at a given incidence) - 2D
Change in pitching moment coefficient due to damage (at a given incidence)-3D

Oswald's shape factor \( e = \frac{1}{1 + \frac{1}{a}} \)

Jone's edge velocity factor \( E = 1 + \frac{2\lambda}{AR(1 + \lambda)} \)

Fillet ratio

Factor related to \( f \) and \( s/b' \)

Wind-tunnel working-section dimension in \( z \)-direction;

Tunnel height (mm)

\( h' \)

\( h/2 \)

Lift

Undisturbed stream Mach number

Static pressure of undisturbed tunnel flow (N/m²)

Local surface tapping static pressure (N/m²)

Velocity Ratio

Damage hole radius (mm)

Wing semi-span

Wing planform area

Aerofoil thickness; Variation of \( \delta \) over entire wing planform

Turbulent Intensity

Undisturbed stream speed

Tunnel velocity (m/s)

Geometric incidence of wing datum-chord (i.e. wing centre-line chord);

Local geometric incidence of wing mean surface \( (= \alpha(\eta)) \)

3D Lift curve slope

Equivalent freestream incidence (degrees)

Effective local incidence of wing mean surface, allowing for non-uniform upwash \( (=\alpha_{\text{eff}}(\eta)) \)

Local effective wing twist relative to datum chord, allowing for non-uniform upwash \( (=\alpha(\eta)) \)

Linearised-theory compressibility factor \( (=1-M_{\infty}^2)^{\frac{n}{2}} \); sideslip angle

Upwash interference parameter \( (=\delta(\eta)) = (w/U)(C/SC_{U}) \)

Mean value of \( \delta \) over wing planform

Value of \( \delta \) related to \( \Delta\alpha_{D} \)

Value of \( \delta \) related to \( \Delta\alpha_{L} \)

Upwash interference parameter for rectangular cross-section

Representative value of \( \delta \) in region of wing

Parameters defined as the value of \( \delta \) at the wing centre of pressure

Proportional to the streamwise gradient of \( \delta \)
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\delta_1$</td>
<td>Maximum variation of $\delta$ over wing chord</td>
</tr>
<tr>
<td>$\Delta_1$</td>
<td>Maximum variation of $\delta$ over wing span</td>
</tr>
<tr>
<td>$\Delta_2$</td>
<td>Effective additional incidence of datum chord associated with upwash interference</td>
</tr>
<tr>
<td>$\Delta \alpha$</td>
<td>Incidence correction (degrees)</td>
</tr>
<tr>
<td>$\Delta \alpha_0$</td>
<td>Correction to incidence of datum chord, related to vortex drag</td>
</tr>
<tr>
<td>$\Delta \alpha_i$</td>
<td>Correction to incidence of datum chord, related to lift</td>
</tr>
<tr>
<td>$\Delta C_p$</td>
<td>Difference between (notional) $C_p$ values computed for uniform an non-uniform Upwash interference</td>
</tr>
<tr>
<td>$\Delta x_{ac}$</td>
<td>Aft shift of aerodynamic centre due to non-uniform upwash</td>
</tr>
<tr>
<td>$\Delta C_L$</td>
<td>Pitching moment coefficient correction, change of lift-coefficient</td>
</tr>
<tr>
<td>$\Delta C_m$</td>
<td>Total blockage correction factor</td>
</tr>
<tr>
<td>$\tau_T$</td>
<td>Solid blockage correction factor</td>
</tr>
<tr>
<td>$\tau_{SB}$</td>
<td>Solid blockage correction factor (at $0^\circ$ incidence)</td>
</tr>
<tr>
<td>$\tau_{wb}$</td>
<td>Wake blockage correction factor</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>Lift loss factor</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Undisturbed stream density (kg/m$^3$)</td>
</tr>
<tr>
<td>$\eta$</td>
<td>Local effective additional incidence associated with upwash interference</td>
</tr>
<tr>
<td>$\Lambda_n$</td>
<td>Sweepback angle of mid-chord line (negative for forward-swept wings)</td>
</tr>
<tr>
<td>$\lambda$</td>
<td>Taper ratio ($= c/c_0$)</td>
</tr>
<tr>
<td>$\zeta$</td>
<td>Local non-dimensional longitudinal coordinate ($= (x-x_d)/c$)</td>
</tr>
<tr>
<td>$\tau_1$</td>
<td>Factor of AAE's rectangular tunnel cross section [and wingspan to tunnel-to-width ratio]</td>
</tr>
</tbody>
</table>

**Subscripts:**

- $c$: Corrected for Blockage and Upwash Corrections
- $u$: Uncorrected for Blockage and Upwash Corrections
- $corr$: Corrected for upwash interference
- $freestream$: Crossflow freestream velocity (m/s)
- $jet$: Crossflow jet velocity (m/s)

**Abbreviation:**

- **CRVP**: Counter Rotating Vortex Pair
- **LSB**: Laminar Separation Bubble
- **JICF**: Jet in Cross Flow
Chapter 1: Introduction

1.1 Background

During modern wars or periods of conflict between forces, the involvement of modern battle aircraft would be the primary tool influencing the outcome. However, when these aircraft are damaged in battle, then several problems may arise.

The first problem is, can the aircraft complete the mission if it is damaged? The answer will depend on the condition of the aircraft, whether partially or severely damaged and which parts of the aircraft are actually damaged.

The second problem is whether the damage may cause the loss of the aircraft due to aerodynamic or structural effects.

When the damaged aircraft has managed to return to its base, then the question is can the aircraft land? -- since frequently returning aircraft stall on final approach. A further question is can the aircraft take-off again and be used in another battle -- or at least is it safe enough to take-off to another base to be repaired? Answers to these questions need an in depth explanation concerning survivability, susceptibility and vulnerability [See section 1.2. for the meaning of these terms].

Initially sponsored by BAe Systems, an investigation into the aerodynamic effects of simulated battle damage to a wing was studied and assessed in the Aeronautical and Automotive Engineering Department (AAE) of Loughborough University, UK. This was ten years ago when BAe Systems produced and specified a preliminary definition of battle damage for aerodynamic studies as stated in Andrew J. Irwin's [1] reference.

The current research is an extension of the two-dimensional investigation carried out by Irwin.
1.2 Survivability, Susceptibility and Vulnerability of Military Aircraft

The subject of survivability is one of critical concern to many disciplines, whether military or civil. It is of particular concern when one is operating in essentially hostile environments. There have been notable examples where damage-intolerant design has led to failure, some with quite disastrous and tragic outcomes [2]. Therefore, survivability is a demanding field of inquiry, and essentially related to broader areas of study such as human factors and safety, modern technological development and military concepts of operations.

In many ways, the history of aircraft development through the years has really been the quest for the survivable airplane -- in particular, military aircraft. Aircraft combat survivability (ACS) is defined as “the capability of an aircraft to avoid and or withstand a man-made hostile environment” [3]. The key words in this definition are “to avoid and or withstand”.

If the aircraft is unable to avoid the radars, guns, ballistic projectiles, guided missiles, exploding warheads and other elements that make up the hostile air defence environment, then the aircraft is categorized as susceptible -- often referred to as the susceptibility of the aircraft.

Aircraft susceptibility can be measured by the probability that the aircraft will be hit while on its mission. Thus, slow, low-flying aircraft that are easily detected, tracked, engaged and eventually hit with one or more damage sites are possibly susceptible. Fast, high-flying aircraft those are difficult to detect, difficult to track if detected, difficult to engage if tracked and difficult to hit if engaged are relatively unsusceptible. However, those assumptions are not always valid. The A-10 aircraft for instance, is the first military aircraft specially designed for close air support of ground forces. It has excellent maneuverability at low air speeds and altitude. Specific survivability features include a titanium armour plated cockpit, a redundant flight control system separated by fuel tanks, a manual reversion mode for flight controls, foam filled fuel tanks, ballistic foam void fillers and a redundant primary structure providing “get home” capability after being hit.

If the aircraft is unable to withstand any hits by the hostile environment, then the aircraft is categorized as vulnerable -- often referred to as the vulnerability of the aircraft. As with susceptibility, the vulnerability of aircraft can also be measured by the conditional probability that the aircraft is killed given a hit.
Examples of vulnerability may be seen if an aircraft having only one engine, no fuel system, fire or explosion protection, or an unprotected pilot. These are categorised as very vulnerable. Aircraft with two widely separated engines, protected fuel systems, and shielding around the pilot are categorised as relatively invulnerable.

So therefore [4], the ease with which an aircraft is killed by the hostile environment is measured by the probability of the aircraft being killed. The probability of kill is the product of the probability of hit (the susceptibility) and the conditional probability of kill from a given hit (the vulnerability).

1.3 Survivability of Military Aircraft Design

Combat survivability, as a formal design discipline for aircraft, is a relatively new concept. This discipline and its importance in the design of military aircraft increased dramatically in the middle 1960s when many aircraft, not specifically designed to be survivable, were shot down in South-east Asia.

The current generation of military tactical aircraft now in development (or low rate initial production), e.g., the F-22 Raptor, F/A-18E/F Hornet, V-22 Osprey and RAH-66 Comanche Helicopter [3] and the latest Joint Strike Fighter (JSF) have strong survivability requirements [5]. Both susceptibility and vulnerability are being reduced using the technology that has evolved over the last 30 years.

A balanced design between susceptibility and vulnerability issues is achieved using trade-off studies to determine the proper balance for the different aircraft with their different missions. This approach is expected to continue into the future, with an improved capability for conducting integrated survivability assessment and trade-off studies [5]. The fundamental approaches to solving those problems remain the same: reduced susceptibility and reduced vulnerability.

1.3.1 Modelling a Battle Damaged Wing in Military Aircraft Design

Modelling and simulation is an essential tool in the design of military aircraft survivability. The use of modelling and simulation has paralleled the growth of computing technology and the reduction in costs. With modelling, the concept can be modelled in a computer and tested in simulated operational environments and against simulated threats in a fraction of the time and with a minimal financial investment.
In the modelling and simulation, aircraft and weapons are digitally defined, based on physical, performance and signature characteristics. Analysts employ a family of survivability models to determine the probability of an aircraft being hit by a weapon (susceptibility) and the probability of aircraft survival after a hit (vulnerability). These analyses are conducted at the aircraft, component, and subcomponent levels such as cockpit, fuel tanks, and flight controls. These results of the analyses are fed back into survivability design and improvement programs. The simulations are limited to the aero-structure-system simulation of wing damage -- not aerodynamic simulation.

A typical simulation may be seen in the Figure-I.1 below where Boeing [5] performed the assessment to characterise wing damage of the hydraulic system analysis during Joint Strike Fighter survivability design.

![Figure-I.1 Boeing's Assessment to characterise wing damage](image)

Objective

Characterize Wing Damage from Hydraulic RAM Analysis

**Figure-I.1** Boeing’s Assessment to characterise wing damage

1.3.2 Threats: Their Characteristics and Damage Processes

The survival of military aircraft on a particular mission is related to the type of mission being conducted, the amount of support from friendly forces and the intensity and effectiveness of any hostile environment encountered during the execution of that mission. The hostile environment can be made up of numerous threat elements, each having a distinct set of characteristics and capabilities. Projectiles and missiles are the main threat mechanism that are considered to be of primary significance and are specially designed to cause damage to aircraft.

This project, as was intended initially, was restricted to the one of the primary methods of inflicting aircraft damage [1] i.e.: penetration. This was chosen because it is a common form of survivable damage.
1.3.3 Unmanned Aerial Vehicles (UAVs) Engineering
Survivability

The relevance of this subject for chapter one is based on two reasons: first, the photograph of 'Global Hawk' below with a high aspect ratio wing is shown in Irwin's thesis. Global-Hawk is a UAV type aircraft that is representative of possible 'future military aircraft'. Secondly, as suggested [by Dr G. Page], the shape and aspect ratio of the current investigation is more suitable to the UAV's wing type application.

Describing the UAV's survivability performance in combat [6], during the 1999 war in Serbia, allied forces lost some 20-25 unmanned aerial vehicles, among them Predators, Hunters, and French/German CL-289s. Two Predators were lost in operations over Iraq this past year and five in Afghanistan as of early 2002. The cause of a loss, by hostile fire or accident, is sometimes unknown when an air vehicle disappears or fails to return to base.

In December 2003 [7], during pre-war with Iraq, the Iraqi air forces shot down a US Predator Unmanned Aerial Vehicle (UAV) that was operating in the southern No-Fly Zone.

![Source: FAS-Intelligent Resource Program Website, April 2005](image)

These operations exposed the susceptibilities and vulnerabilities of UAVs. Recognizing the need to address shortfalls in existing and proposed UAV designs, engineers and program managers have more aggressively introduced survivability concepts to UAV program offices and airframe manufacturers.
Chapter 2: 
Literature Survey

A literature survey on the aerodynamic characteristics of damaged wing aircraft has been undertaken. However, very little published work on the topic is available. Several of the literature sources described below was not available - simply due to their age. If that is the case, then details obtained from the description given by other workers, to which the literature was available will be re-interpreted. The survey consists of three main approaches, i.e. a survey that related to “jets-in-crossflows”; a survey that related to simulation and modelling of battle-damaged wings; and lastly, a survey that related to the experimental method and techniques of battle-damage wing investigations.

2.1 Studies Related to Aerodynamics of Battle Damage: Modelling,Simulation, Methods and Testing Techniques

It was Schemensky & Howell of General Dynamics Corporation, who in 1978, as referred to by Lincoln [8], are believed to be the first workers to undertake research on the modelling and simulation of the influence of damage on the aerodynamic characteristics of aircraft. They developed an empirically based computer program for determining the lift, drag and pitching moment characteristics of aircraft that have sustained nuclear damage. The program assessed the effects on aircraft aerodynamics of damage such as rough, bent and burnt surfaces, loss of radomes and panels, and the asymmetric loss of wing or trim surfaces. The program was also designed to predict baseline aerodynamic data for undamaged aircraft. The characteristics of damaged aircraft were obtained by simply adding the incremental components to those for the undamaged aircraft.

To verify the accuracy of the computer program, Lincoln of DSTO [Defence Science and Technology Organisation] - Australia, reported in 1998, that predictions of aerodynamic coefficients, $C_L$, $C_D$, $C_M$ were made for a range of values of incidence, $\alpha$ and Mach number, $M$. The prediction was compared with experimental data and was made for both damaged and undamaged aircraft. It was found that predictions and experimental data compared favourably thus giving credibility to the program.
A year later, the University of Texas at Austin undertook research on the modelling and simulation of the damage influence on the characteristics of lifting surfaces when Westkamper et al., as referred in Ref. [1], produced two reports: One report concentrated on aero-elastics failure mechanism and the other report concentrated on aerodynamic characteristics. The use of a full size Cessna T-38 horizontal tail plane at relatively low Reynolds Number produced inaccurate results and was unverified. The inaccuracy of the characteristics was attributed to leading edge separation effects. However, Westkamper's modelling suggested that little variation in the chordwise pressure coefficient, $C_p$ was likely to occur either upstream or downstream of the damage hole and that calculated $C_p$ was largely attenuated within approximately one hole width on either side of the damage area.

The earliest studies related to the aerodynamics of battle damage, i.e. the vulnerability of aircraft, were started in Cornell Aeronautical Laboratory. The studies were begun by Gail et al., as referred to in Ref [1] in 1952, when they estimated the aerodynamic vulnerability of subsonic aircraft. In the same year, Reece - also referred to in Ref. [1] from the same laboratory investigated drag changes due to a holed wing and the characteristics of flow within the aircraft structure. Gail and Reece undertook the tests on hollow airfoils, which correctly simulated the effects of through-flow between the upper and lower surfaces. These hollow airfoils were tested at subsonic Mach number of 0.3 to transonic range Mach numbers of 0.7 to 0.85. They found that the resulting changes in $C_l$ and $C_d$ were function of damage hole size. This was 10% and 15% of chord in diameter.

Nearly ten years after Gail's investigation, Hayes at NASA — as referred to in Ref. [1] and Ref. [8] produced work on battle-damaged wings from 1968 to 1976. Hayes conducted his investigation in a supersonic wind tunnel to determine the effects of simulated wing damage on the static aerodynamic characteristics of a generic swept-wing aircraft model. The model shown in Figure-II.6 consisted of an ogive-cylinder fuselage, swept wing and vertical tail.

![Figure-II.6: Generic Swept-wing aircraft model, showing regions of damage (shaded)](image)
The damage was simulated by removing the leading-edge/trailing edge section of a wing. Removal of the leading edge resulted in an 11% reduction in the total wing area and removal of the trailing edge resulted in a 17% reduction. Coefficients, $C_L$, $C_D$, $C_Y$, $C_n$, $C_m$ and $C_r$, as well as parameters of lift-to-drag-ratio, $L/D$, $C_m$ and $\Delta C_m/\Delta C_n$, were determined for a range of values of angle-of-attack $\alpha$, sideslip angle $\beta$ and $M$ ($-4^\circ < \alpha < 22^\circ$), ($-5^\circ < \beta < 10^\circ$), (1.7 $< M < 2.86$) for a constant Reynolds number of $7.38 \times 10^6$/m.

It was found that removing the leading edge or the trailing edge led to a decrease in the lift curve slope and in the maximum lift/drag ratio. At the lower Mach numbers, removal of the trailing edge caused a rolling moment slightly larger than that caused by the removal of the leading edge, but the effect was reversed at higher Mach numbers, even though the trailing edge had more area than the leading edge. When the leading edge or trailing edge was removed, it was possible to trim the aircraft by altering the angle of sideslip, while maintaining angles of incidence and sideslip within reasonable limits, but trim could not be achieved when the entire wing was removed.

NASA's eight-years of continuous work on battle-damaged wings concluded with the work of Betzina & Brown in 1976 - as referred to in Ref. [1] and Ref. [8]. The work measured the static aerodynamic characteristics of a McDonnell-Douglas A-4B aircraft with both simulated and actual gunfire damage to the starboard wing. A full-scale aircraft was used for the experiments that were carried out in the NASA-Ames 40x80 foot wind tunnel. Three different wings were attached to the aircraft for the tests. One of the wings had sections removed from the upper and lower surfaces and these were replaced with eleven detachable panels, as shown in Figure-II.7a.

![Figure-II.7a](image-url)
By removing different combinations of panels, fourteen different simulated damage cases were obtained. The wing was tested both with and without panels removed. The other two wings used had been damaged by actual gunfire. A 25mm projectile was fired at one wing and a 30mm projectile was fired at the other. The two projectiles were fired at the same region of each wing from an angle of $15^\circ$ above and behind each wing. The 25mm projectile created a large hole on the upper wing surface and blew off part of the landing gear fairing on the lower surface. The 30mm projectile damaged mainly the lower wing surface, where the landing gear fairing was torn off. As damage size increased so did the magnitude of lift loss. With relatively large damage present – in addition to significant reduction in lift, as can be seen in Figure-II.7b, the onset stall was observed to have been delayed some $2^\circ$ to $3^\circ$; whilst drag increases were minimal – approximately 5% at minimum drag coefficient, $C_d$.

![Figure-II.7b: Lift and Drag Coefficients for A4B Aircraft](image)

Following the conclusion of the high speed model testing at NASA Ames in 1979, no further publications were made on the subject until 1982, when wind tunnel tests were undertaken by Spearman – as referred to in Ref. [1] and Ref. [8] – to investigate the effects of damage on airplane and missiles static aerodynamic characteristics. The investigation was undertaken to help determine the extent of damage that an aircraft can sustain and still complete a mission or return to friendly territory. Spearman summarised transonic wind tunnel tests carried out at the
Langley Research Centre using models of undamaged and damaged aircraft. Three types of aircraft models were used, namely a swept-wing aircraft, a delta-wing aircraft and a trapezoidal-wing aircraft. Damage to an aircraft was simulated by the removal of all or part of a wing, horizontal tail or vertical tail. An example of typical simulated damage is given diagrammatically in Figure-II.8 for the trapezoidal-wing aircraft.

![Figure-II.8 : Trapezoidal-wing aircraft model](image)

Shaded Areas were removed to simulate damage.

For a given set of measurements, the damage was confined to a specific region of the aircraft, e.g. a wing was not damaged at the same time as a horizontal tail.

It was found that, for the delta wing and the trapezoidal-wing aircraft at supersonic velocities, major damage to a wing might be sustained without necessarily losing the aircraft. Also, the loss of major parts of the horizontal tail may cause an aircraft to be catastrophically unstable in the subsonic range, but stable at low supersonic velocities. Thus, even though it may not be possible for a pilot to land the aircraft at subsonic velocities, the aircraft could possibly be flown to friendly territory at low supersonic velocities before the pilot must eject. Spearman also found that the loss of a major part of the vertical tail would result in the loss of an aircraft for both subsonic and supersonic velocities.

Almost ten years after Spearman, Leishman et. al. [9] at the University of Maryland from 1993 to 1998 went 'back to basics' by undertaking wind tunnel tests on a UH-60A Black Hawk helicopter main rotor airfoil. The experiments were conducted to estimate the effects of ballistic damage on the aerodynamic characteristics. During the experiments, the lift, pitching moment and drag were measured on nominally two-dimensional blade specimens with representative prescribed and actual ballistic damage. The measurements were made at chord Reynolds
numbers between one and three million and Mach numbers up to 0.28. Force balance measurements were complemented by chordwise and spanwise pressure measurements to assess the three-dimensional nature of damage on the aerodynamics. The quantitative data were supplemented by surface oil flow visualization. Generally, it was found that ballistic damage degraded the aerodynamic performance of the blade specimens, with a reduction in lift accompanied by significant increase in drag and change in the centre of pressure.

In the same period as Leishman's investigations, Irwin et. al. in 1995 [10] and Irwin and Render in 2000 [11] of Loughborough University, under BAe sponsorship, investigated the effects on airfoil static aerodynamic characteristics of idealized gunfire damage, simulated by drilling circular holes in two-dimensional hollow airfoil sections [Figure-II.9]. Damage was considered the consequence of a hit by a single anti-aircraft artillery [AAA] gunfire round. Live-fire testing, as referred to in Ref. [1], has indicated that the most common damage shape is circular. While this shape might be expected from a non-explosive armour-piercing shell, the results showed that similar damage was also obtained from high-explosive shells. Therefore, the damage was modelled as a circular hole.

Airfoils having chord 200mm were used and holes were located on the mid span of the airfoil, at the leading and trailing edges as well as at the 25% and 50% chord positions. Holes were drilled normal to the chord line and holes of four different diameters were used, between 0.1C and 0.4C; where C is the chord length. The airfoils were placed in a low-speed wind tunnel and only one hole in an airfoil was used at a time for any given set of measurements. The flow patterns around the airfoils were modified as a result of air flowing through a hole from a high pressure to a low-pressure region and the aim of the investigation was to study how airfoil characteristics changed for different hole locations, hole sizes and airfoil sizes. Values of $C_l$, $C_d$ and $C_m$ were measured for a range of values of alpha (-10° < alpha < 15°) for Reynolds number of 5.0 x10^5.

The researchers found that:

a) The holes at 25% and 50% chord locations had more effect on the lift, drag and pitching moment coefficients than the leading edge holes, whereas the trailing edge holes were found to have little effect on the coefficients.
b) The coefficients were affected very little by the holes diameter 0.1C, and greater effects were evident for holes of diameter 0.2C, 0.3C and 0.4C.
c) The coefficients were found to be largely independent of Reynolds number for the two Reynolds numbers used. The above findings apply to solid airfoil sections as well as hollow sections.
The work then continued with more emphasis on the effect of mid-chord damage on the aerodynamic characteristics of 2D wings. The principal outcomes of this work were:

a) The damage through-flow was driven by the pressure differential between upper and lower wing surfaces, and took one of two forms: firstly, a ‘weak-jet' forming an attached wake with minimal localized $C_p$ changes. $C_l$, $C_d$ and $C_m$ changes were relatively small. Secondly, a ‘strong-jet' resulted from either increased incidence or damage size, where through-flow penetrated further into the cross-flow, resulting in the detachment of the oncoming surface flow and development of a separated wake region and reverse flow.

Note that the weak-jet characteristics that happen on the upper surface for low positive incidence range, and on the lower surface for low negative incidence range depend on the hole location and size relative to the surface chord. The larger the hole size or the more rearward the hole location would change the incidence range that characterised as weak-jet -- for both upper and lower surfaces.

Irwin discovered that increasing the damage hole size to relatively large hole diameters, e.g. 30% to 40% relative to typical NACA wing surface chord, would likely result in no weak-jet characteristics on either the upper or lower surfaces.

b) $C_l$, $C_d$ and $C_m$ effects on strong-jet indicated significantly greater lift-loss, drag increments and pitching moment changes than those seen for the weak-jet. Lift-losses resulted primarily from the reduction in pressure peak upstream and either side of the damage, whilst the $C_p$ within the detached wake region resulted in significant $C_d$ increments and $C_m$ variations.

c) Pressure field influences extended significantly up to and beyond a spanwise distance of 'SR' -- 5 times the damage radii $R$, from the damage centre.

Irwin also investigated the characteristics of simulated missile damage -- modelled with uniform holes following fragmentation patterns of missiles. The aerodynamic effects of such damage were assessed, and the consequence of varying two key variables: fragment density and damage...
hole size, were considered. As with previous gunfire damage testing, both the measurement of aerodynamic coefficient changes and surface flow visualisation techniques were undertaken. No surface pressure measurements were made.

The main finding of this investigation was that low fragment densities and smaller damage sizes resulted in a complex surface flow structure made up of boundary layer growth, attached wakes and detached surface flow. The individual hole pattern reflected a similar flow mechanism to those seen on a larger scale for gunfire damage cases. Increased fragment density and hole size resulted in upper surface separation at the first row of holes at low incidences.

2.2. General Definition of ‘Weak-Jet’ and ‘Strong-Jet’

2.2.1 Introduction

The terms ‘weak-jet’ and ‘strong-jet’ are used to identify two flow structures. The use of ‘jet’ terminology is adopted from previous investigations of ‘Jets-in-cross-flows’. The structurally interesting phenomena observed in jet-in-crossflow and expected to happen in the damage hole case are roll-up in the jet, formation of a counter-rotating-vortex-pair [CRVP], a horseshoe vortex system in the cross-flow boundary layer upstream from the jet exit and the creation of wake vortices.

2.2.2 ‘Weak-jet’ Flow Characteristics

The characteristics of a weak-jet is that the jet momentum through the damage hole is too low to allow penetration into the freestream flow; and for that reason, the jet exiting at the rear edge of the hole is deflected instantly. The jet exits at the rear edge because it follows the direction of the freestream flow, which has a higher velocity than the jet velocity.

A possible situation may happen where the weak-jet does not exit at the rear of the hole when the freestream crossflow velocity is significantly lower than the jet momentum. Schematics are shown in Figure-II.10.

At the point where the surface flow meets the jet, an adverse pressure gradient is created, causing separation ahead of the damage hole. This is the forward separation line. The secondary separation line is the line that forms due to flow reattachment. So called because flow through the damage pushes forward from the damage hole, and separates where it meets the horseshoe vortex. This horseshoe vortex therefore, lies between the forward and secondary separation lines. Just ahead of the damage hole, a region of upstream 'reverse-flow' is also present, which is subsequently entrained rearward and adds to the damage wake.
Figure II.10: Schematic of 'Weak-jet' Flow and Wake Characteristics

Figure II.11: Schematic of 'Strong-jet' Flow and Wake Characteristics
At the rear of the damage hole, a pair of counter rotating vortices [CRVP] forms. The positions of the CRVP vary with jet strength momentum; tend to forms downstream of the damage hole circumference at low incidence, and move forward around the damage wake with increased incidence. The CRVP remaining approximately symmetrical about the hole centreline during the change of incidence.

Downstream of the hole where the damage wake attaches to the surface, there is a laminar separation bubble [LSB], this is located at approximately half to two-thirds of the surface chord - purely because of airfoil geometry. The LSB is split and cut-through by the damage hole wake but it has no connection with the damage jet itself.

At the hole exit, as the jet instantly distorts, it is expected that the ‘height’ -- a distance from the surface to the jet boundary, is small. As the flow in the wake moves in the freestream direction, a surface velocity gradient is likely to happen within the wake from the centreline outwards. Also, the edges of the wake will have a greater velocity than at the centre [1].

2.2.3 ‘Strong-jet’ Flow Characteristics

In ‘strong-jet’ characteristics, the jet no longer distorts instantly upon exiting the damage hole, but instead, penetrates into the freestream and becomes detached from the wing surface. This in turn creates a separated region between the jet and the upper surface extending from immediately behind the damage to the trailing edge. The flow is three-dimensional with significant reverse flow within the separated region or wake. These are schematically sketched in Figure-II.11.

In a strong jet, the surface flow immediately upstream of the damage is turbulent, and the formation of the forward and secondary separation lines show little change in position from the weak jet [1].

As with weak-jets, moving chordwise, the path of the separation lines and the vortex deflects into a horseshoe configuration, but with a greater spanwise extent. This significant increase in wake size indicates the extent of the large region of reverse flow beneath the detached strong jet.

At the rear of the damage hole, the CRVP form and are symmetric about the damage-hole centreline, but their position moves forward around the damage hole as the incidence increases and the jet gets stronger. In this region, just downstream of damage hole, a slow-moving flow exists; and the wake shows significant reverse flow. The reverse flow then separates when it meets the exiting jet. Large surface vortices form because surface flow is deflected around the large jet. Reverse flow is entrained from the surface between the two large vortices – which are rotating in opposite directions.
2.3 Evolution and Studies Related to ‘jets-in-crossflow’: Experimental Investigations

There is a similarity between battle damage and a ‘Jet in Crossflow’ [JICF]. Despite the irregular geometry of battle damage, the jet exiting from the damaged wing is interacting with the freestream flow -- the main principle of the JICF. Jet in crossflow is a basic flow-field that is relevant to a wide variety of applications where a jet exhausts from the hole, nozzle, or pipe into a freestream flow. The jet, depending on parameters like its momentum, pressure, geometry, incidence, etc. will either bend to the directions of the crossflow – ['weak-jet' case] or penetrate into the freestream flow ['strong-jet'].

JICF have been used in many technological applications and the objectives are different depending on the specific applications. In aeronautics, it applies in V/STOL, turbine blade cooling, fuel injection for burners; thrust reversers for propulsive systems, and battle-damage wing research. In industry, the applications are in chemical plants, fuel injection in engines and chimneys.

In aeronautics, several experimental studies have been undertaken to examine the characteristics of JICF: e.g. jets through flat-plate flow cases; jets directed normal to the freestream; circular jet normal to the wind; the path of a jet directed at angle(s) to the freestream; pressure distribution of jets; and inclined jets. Although the most extensive research is connected with VTOL aircraft application, the mechanics of flows described in those cases show similar characteristics to the current investigation. A brief discussion of the evolution of these studies can be found below.

One source Margason [12] covers and summarises the evolution of JICF studies from the 1940’s to the 1990’s. He emphasized two experimental cases, which were taken from Andreopoulus and Rodi in 1982 [13]. The first was a uniform jet which exits a nozzle on an infinite plate into a uniform cross-flow \( r = \frac{V_{\text{jet}}}{V_{\text{freestream}}} \), where the jet angle relative to the flat-plate and jet nozzle shape were the most important parameters [low \( r \) case]. The second case was the tests where the velocity of the jet was significantly greater than that of the crossflow i.e. higher \( r \).

The complicated nature of the jet in crossflow is shown in Figure-II.12[a] and II.12[b] is the composite pictures of the flow development that were presented for the velocity ratios, \( r = 0.5 \) and \( r = 2 \) respectively . These two pictures represent the first and the second cases.
The most obvious feature of these two figures is the mutual deflection of both jet and crossflow; and it is clear how these two figures differentiated. In the small \( r = 0.5 \) case, the jet is bent over by the cross-stream, while the latter \( [r = 2] \) is deflected as if it were blocked by a 'rigid obstacle'; the difference being that the jet interacts with the deflected flow and entrains fluid from it.

In the small \( r = 0.5 \) case, the behaviour is as if a partial, inclined 'cover' were put over the front part of the exit hole, causing the jet streamlines to start bending while still in the discharge tube and the jet to bend over completely right above the exit which lifts-up the oncoming flow over the bent jet.
In Margason’s second case of the higher velocity ratio \( r = 2 \), the jet is only weakly affected near the exit and penetrates into the cross-stream before it is bent over.

The description that applies in both cases that, wake regions with very complex three-dimensional flow patterns form in the lee of the jet. In this region, the longitudinal velocity accelerates and the conservation of mass requires fluid to move from the sides towards the plane of symmetry. Very close to the wall, a reverse flow region forms, and cross-stream fluid has been observed to enter this region, travel upstream and then to be lifted upwards by the jet fluid and to be carried downstream together with it.

Margason then further assessed these cases by discussing the vorticity characteristics. Analysis showed that an important feature of this flow is the deflection of the freestream flow lines in the \( x \) and \( z \) direction and the associated reorientation and generation of vorticity.

It was found that the presence of streamwise vorticity downstream of the exit, which is contained in the secondary motion, was formed by two counter-rotating vortices, which give the bent-over jet a kidney shape. Considering the vorticity of the approach flow, its interaction with the jet forms a horseshoe vortex similar to that found when a boundary layer is deflected around an obstacle. In addition, the oncoming boundary layer separated upstream of the jet as indicated in figures Figure-II.12.a and II.12b. Also that the vorticity present causes most of the entrainment of cross-stream fluid into the wake region and into the deflected jet.

Previously before the Andreopoulus and Rodi investigations, Mosher in 1970 [44] in his thesis investigated the interference phenomenon occurring when a subsonic turbulent jet exhausts normally from a large flat plate into a low speed crossflow. This was experimentally investigated in the Georgia Tech nine-foot wind tunnel. Static pressures were measured on the flat plate’s surface around the jet. In the region off the surface, including the jet plume, wake and surrounding areas, the average total and static pressures and the average velocity magnitudes and directions were determined. Three jet exit configurations were studied, one circular and two slot-shaped with width to length ratios of 0.3 and 3.4. All had the same exit area. These widths to length ratios respectively represent weak and strong jets characteristics. The velocity ratio was varied, for each of the exit configurations, over the range 4.0 to 12.0.

Analysis of the data indicated that the static pressure distributions induced on the surface are a combined result of the jet’s blocking and entraining effects on the cross flow with entrainment becoming the more dominant of the two as the effective velocity ratio is increased. This relative dominance causes a rise in the low static pressures in the wake region as the effective velocity ratio increases.
Crabb et al. in 1980 [46] investigated the velocity characteristics using Laser-Doppler Anemometry in the upstream region, where the turbulence intensities were larger; and hot-wire anemometry in the downstream region. Jet to cross-flow velocities ratio [$r$] of 2.3 and 1.15 were used. These velocities ratios represent strong-jet characteristics. The results confirmed the presence of a counter-rotating-vortex-pair [CRVP] in the jet that was previously identified by Margason [12] and demonstrated that these are associated with fluid emanating from the jet.

The vortex structure of the JICF as sketched in Figure-II.13 has also been the subject of more rigorous investigation during this period. In an experimental investigation, Fric and Rosko in 1989 [55], worked with transverse jets of air in a wind tunnel using smoke streak-lines vaporized from a transverse wire and smoke released with the jet or from the cross-flow boundary. They identified and photographed the four types of vortical structures that exist in the JICF near field and identified their sources:

Jet shear layer - distorted shear layer ring vortices at the upstream circumference of the deflected jet. In their photograph, it shows what appear to be growing vortices on the upstream face of the deflected jet.

Vortex pair - the dominant longitudinal embedded rotating vortex pair evolves from the shear layer vorticity of the jet, although they have not produced an argument to demonstrate this convincingly. Their results indicate that the contra-rotating pair of vortices begin forming quite early and are already developed at one jet diameter downstream of the center of the jet.

![Figure-II.13: Sketch of the vortex systems associated with the Jet in Cross Flow.](image-url)
**Horseshoe vortexes** - due to the adverse pressure gradient just ahead of the jet on the cross flow wall. The horseshoe vortices are regarded as fed by the vorticity advected with the cross-flow boundary layer;

**Wake vortex street** - Unsteady Wake Vortex street shedding immediately downstream of the jet exit. Fric and Rosko argue that the formation of wakes behind deflected jets and in particular to the appearance of upright vortices, derive from the vorticity of the cross-flow boundary layer.

The jet shear layer vortices and horseshoe vortex indicated by Fric and Rosko above was then elaborated by the experiments of Kelso et al. in 1996 [54] in both water and air. Kelso also found that the jet shear layer vortices are produced directly at the jet orifice. The two streams [jet stream and crossflow] form a mixing layer, which cause a roll-up near the edges of the jet. This is shown in Figure-II.14 [a] and Fig.-II.14[b].

![Jet Shear Layer Vortices and trajectory of the horseshoe vortex;](image)

![Horseshoe vortex and recirculation bubble at the upstream edge of the jet orifice.](image)

Morton et. al. in 1996 [14] carried out a series of experiments on deflected jets in water channels. The jets were injected vertically downward through an orifice flush with the Perspex ceiling plate. Horizontal cross-flow hydrogen bubble wires were used for flow visualization.

Morton found -- as had already been discovered by Margason that an embedded contra-rotating-vortex-pair forms in such deflected jets. It confirmed that deflected jets must necessarily contain embedded pairs of vortices of modest strength with vortex axes approximately parallel to the curved jet axis and occupying most of the jet cross section.

Also, in water tunnel experiments, Haven et. al. of NASA in 1997 [15] examined the effect of hole exit geometry on the nearfield characteristics of crossflow jets. Hole shapes investigated were round, elliptical, square, and rectangular, all having the same cross-sectional area [Figure-II.15]. Laser-induced fluorescence (LIF) and particle image velocimetry (PIV) were used. Haven found that the jet from the high-aspect-ratio holes, with increased separation distance between the sidewall vortices, stays attached to the surface for higher velocity ratios than for the
low-aspect-ratio holes. Thus, by manipulating the hole geometry alone, without increasing the hole cross-sectional area, one can delay separation of the jet.

In a number of cases, it is necessary to deal with a jet expanding into a stream of fluid at an angle to it -- which is the actual condition of the present investigation. Abramovich [47] described the mechanism as that of a jet of fluid entering into a flow of the same or some other kind of fluid, which is moving at an angle to the jet's axis, is bent and becomes curved. The flow being slowed by the jet at its leading [convex] edge, creates an increased pressure, while at the rear [concave] side rarefaction occurs. The pressure difference creates the centripetal force necessary to deform the jet. This description however, is also valid for a jet that is normal to the flow.

Figure-II-16 represents the contours of a jet of air from a hole diameter of 20mm with velocity ratio $r = 0.46$. The figure shows the velocity and the pressure fields in the plane of symmetry. The solid line gives the profile of total pressure; the static pressure profile, by the dotted line; and the arrows designate velocity vectors. The figure were defined based on measurements made by means of a cylindrical probe traverse along directions normal to the axis of the jet which is represented by the geometric location of the points with the maximum velocity values.
The physical explanation for pressure contours of this figure is that the total pressure changes sharply, decreasing at the edges of the jet. The line of maximum total pressure is located closer to the forward edge of the jet than the line of maximum velocities, which is explained by the character of variation of the static pressure in the cross sections of a curved jet. Because of the inflow of fluid to the jet, the increase in pressure at its forward edge is somewhat less than it would be at the wall of the solid body of the same form as the curved jet. In the jet itself, the static pressure is continuously decreased from its forward edge to the rear.

In the early region of the jet deflected by the freestream flow -- which is a region not influenced by turbulent mixing [relatively unaffected by the crossflow – See Fig.-II.16], there is a core of constant total pressure – but of variable velocity. The velocity in the cross section of the potential core increases toward the back edge of the jet because of the decrease in static pressure. The velocity vectors behind the jet have components directed counter to the velocity in the deflecting flow, which indicates a complex configuration of circulatory motion behind the jet.

Continuing Abramovich's study, Fearn and Weston in 1979 [45] of NASA reported an investigation of a subsonic round jet injected from a flat plate into a subsonic crosswind of the same temperature but with several injection angles. Velocity and pressure measurements in planes perpendicular to the path of the jet were made for nominal jet injection angles of 45°, 60°, 75°, 90°, and 105° and for jet cross flow velocity ratios of four and eight. The velocity measurements were obtained to infer the properties of the vortex pair associated with a jet in a cross flow. It was discovered that jet centreline and vortex trajectories could be determined from an empirical relationship that includes the effects of jet injection angle, jet core length, and jet cross flow velocity ratios. For these parameters, Fearn and Weston results shows further the effect of jet deflection angle on the velocity decay for two velocity ratios, $r$ of 4 and 8. For all injection angles, the jet decay was faster in a crossflow and has a modest further decay increase as the deflection angle increases to perpendicular. There was no explanation of the jet decay when the deflection angle increased to 105°.

Gustaffson [54] in 2001 conducted detailed three-component velocity measurements in the wake of an inclined jet in a cross-flow with Laser Doppler Anemometry [LDA]. The operational parameters used in this investigation was, $U_{jet}/U_{freestream} \ [r] = 0.8$; $T_{jet}/T_{freestream} = 1$ -- no temperature difference between the jet flow and crossflow; and $T_{jet}/T_{freestream} = 1$ -- no velocity difference between the jet flow and crossflow. In addition, the injection hole was inclined at 30° to the freestream flow direction [$x$-axis].
One major finding of Gustaffson’s LDA measurements in the wake shown in Figure-II.17 was the two counter-rotating foci close to the wall downstream of the jet that could be observed an indication of a 3D separation.

Horseshoe vortex lines [red lines] at the side of the jet and vortex lines originating from the foci are shown. The black streamlines are located at the wall and the blue ones are located at the plane of symmetry. Aft of the saddle point, S a separation line is seen consistent with the upward motion caused by the CRVP.

2.3.1 Jet in Cross Flow [JICF]: Empirical Model

Introduction

Empirical models present the simplest means of predicting global properties of jets in cross flow. They depend largely on the correlation of experimental data, and the accuracy of the predictions will depend on the closeness of the conditions of the particular problem of interest to those in the database used for the correlation. In the earliest studies of jets in cross flow, empirical models were developed to correlate experimental data obtained under various idealized conditions. Abramovich [47] and Demuren [49] review such models in detail, most of which consider the jet trajectory.

The Velocity and Momentum Ratio

The dominant quantity to characterize a JICF is the velocity ratio $r$ defined as [9]:

$$r = \left( \frac{V_{\text{jet}}}{V_{\text{freestream}}} \right)$$

where, $V_{\text{jet}}$ and $V_{\text{freestream}}$ are jet and freestream velocities.

Alternatively, the momentum flux ratio, $J$: 
\[ J = \left( \frac{\rho_{\text{jet}}}{\rho_{\text{freestream}}} \right)^2 \]  

where, \( \rho_{\text{jet}} \) and \( \rho_{\text{freestream}} \) are jet and freestream flow densities. In most cases, the jet fluid and the cross flow fluid consist of the same fluid and have the same density.

Survey of Modelling Analysis

For modelling analysis, the approach used by Domuren [48] considers the plane of symmetry of a single round uniform jet in confined cross flow is illustrated in Figure-II.18.

As shown in Figure-II.18, the jet in a cross flow has three main regions: the potential core region within which the central portion of the jet remains relatively unaffected by the crossflow (I) -- as shown in Figure-II.17; the zone of maximum deflection (II) or curvilinear zone where the jet experiences the most deflection; and the final region (III), the far-field zone where the jet axis approaches the crossflow direction asymptotically.

The most common parameter given by empirical models is the ‘jet trajectory’. For a single circular turbulent jet injected normally into a cross flow, the trajectory has the form [Ref. 48]:

\[ \frac{z}{D} = aJ^b \left( \frac{x}{D} \right)^c \]  

where,

\( a, b, \) and \( c \) are constant parameter;

\( x, z \) are the coordinated of the points shown in Figure-II.19;

\( D, \) is the diameter of the jet at exit; and

\( J \) is jet-to-cross-flow momentum flux ratio.

for the range of \( J \) between 2 and 2,000. The range is simply far too large to be implemented to weak and strong jets cases in battle-damaged wing; where the typical \( r \) and \( J \) values are shown in
Table-II.1. [See further on Chapter-5, Section 5.6.1 and 5.6.2: ‘Empirical Model Application on Battle Damaged Wing’].

<table>
<thead>
<tr>
<th>Jet Conditions</th>
<th>Infinite Wing [2D] with Damage Hole at 25% Chord</th>
<th>Finite Wing [3D] with Damage Hole at 50% Chord</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weak-jet</td>
<td>$r = 0.5225$, $J = 0.273$</td>
<td>$r = 0.4875$, $J = 0.2376$</td>
</tr>
<tr>
<td>Strong-jet</td>
<td>$r = 1.0225$, $J = 1.045$</td>
<td>$r = 0.9375$, $J = 0.8789$</td>
</tr>
</tbody>
</table>

Table-II.1: Typical $r$ and $J$ values of weak and strong jets cases in battle-damaged wing

Although Pratte and Baines (1967) due to [48] suggested that the values $a = 0.85$, $b = 0.47$ and $c = 0.36$ appear to be a good compromise for the above range of $J$, the range is still unlikely to be applicable.

Correlations for predicting an ‘air jet axis’ or jet trajectory from a circular jet discharged at oblique angles into the cross flow may be more relevant to the current investigation. Jet trajectory may be calculated using the empirical equation by Shandorov [1957] due to Abramovich [47]:

$$\frac{x}{D} = J^{-1} \left( \frac{z}{D} \right)^{2.55} + \frac{z}{D} \left( 1 + J^{-1} \right) \cot \theta$$ ...

where:

$\theta$, is the angle between the jet’s axis and the direction of the deflecting flow;

The experiments -- from where the empirical equation is formulated --, were made with a uniform velocity field in the deflecting flow. The ratio of $J$ was in the range from 2 to 22, whilst $\theta$ was from 45° to 90°.

Another empirical equation was derived by Ivanov [1971] from Abramovich [47] and appears in Margason [12]:

$$\frac{x}{D} = J^{-1.3} \left( \frac{z}{D} \right)^{3} + \frac{z}{D} \cot \theta$$ ...

In this experiment, the initial angle of the jet was changed from $\theta = 60^\circ$ to $120^\circ$. The ratio of $J$ was in the range from 12 to 1000. Abramovich then assessed that value of $\theta$, $J$, and the trajectories of the jet for both equations -- Shandorov and Ivanov--, the results were close to each other and predict similar trajectories in the whole range of the experiments.

The most comprehensive review of experimental studies concerning JICF is probably found in Margason [12] where expressions were compiled and reassessed from some investigators. The
results were compared for the ‘path of maximum velocity’ for variable jet deflection angle, $\theta$ and variable ‘effective velocity ratio’, $V_e$. Margason then identified the best correlation using the following equation and improved by taking the z origin at the end of the jet potential core. The correlation is given by:

$$\frac{x}{D} = FV_e^n\left(\frac{z}{D}\right)^m + \frac{z}{D} G \cot(\theta)$$

...[6]

where:

$F$, $G$ are jet trajectory coefficients;

$n$, $m$ are jet trajectory exponents;

$\theta$ is the angle between the jet’s axis and the direction of the deflecting flow, and

$V_e$ is effective velocity ratio $= \sqrt{\frac{q_{\text{jet}}}{q_{\text{free stream}}}}$; where, $q_{\text{jet}}$ and $q_{\text{free stream}}$ are jet and freestream dynamic pressure. Since fluid for free-stream and jet are identical, and $r = \left(\frac{V_{\text{jet}}}{V_{\text{free stream}}}\right)$, hence effective velocity ratio, $V_e = \frac{1}{r}$. The range of $V_e$ for which Equation -6 is valid is $V_e > 0$. At no crossflow, $V_e$ is always zero.

Comparisons of the coefficients ($F$ and $G$) and exponents ($n$ and $m$) used in the above equation by those investigators were then presented by Margason as shown in Table-II.2.

The table is specifically defined for a jet at an arbitrary deflection angle to the free stream where $G$ is usually 1. The table shows that the $F$ term tends to be either a numerical constants or variable involving a $\sin \theta$. The $n$ and $m$ exponents tend to vary from 2 to 3.

<table>
<thead>
<tr>
<th>Author</th>
<th>$F$</th>
<th>$n$</th>
<th>$m$</th>
<th>$G$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Vakhlamov [1964]</td>
<td>$\frac{1}{\sin \theta}$</td>
<td>2</td>
<td>See below*</td>
<td>1</td>
</tr>
<tr>
<td>Visel &amp; Mostinski [1965]</td>
<td>$\frac{5}{(4\sin \theta)}$</td>
<td>2</td>
<td>2</td>
<td>1</td>
</tr>
<tr>
<td>Shandorov [1966]</td>
<td>1</td>
<td>2</td>
<td>2.55</td>
<td>$1 + V_e^2$</td>
</tr>
<tr>
<td>Margason [1968]</td>
<td>$\frac{1}{(4\sin^2 \theta)}$</td>
<td>2</td>
<td>3</td>
<td>1</td>
</tr>
<tr>
<td>Ivanov [1971]</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>1</td>
</tr>
</tbody>
</table>

* $m = 2.53 \zeta + 1.59 \zeta^2 + 0.143 \zeta^3$ where $\zeta = 2/D$

**Table-II.2: Jet Trajectory terms for Equation -6 arbitrary Jet angle, $\theta$**
2.4 Research Outlines and Objectives

2.4.1 Scope of the Investigation

This investigation has focused upon the effects of simulated battle damage on the low-speed aerodynamics of a finite aspect ratio wing. The work was an extension of the infinite wing ['2D'] studies carried out by Andrew J. Irwin, which lead to a successful PhD submission in 1998. The size and flow quality of AAE's new wind tunnel made it possible to extend Irwin's work to the Finite Wing case. A flowchart of the research sequence is shown in Figure-II.12.

2.4.2 Project Aims

1. Investigate experimentally the flow mechanism occurring in single hole damage on a finite aspect ratio wing by force measurement, flow visualisation and spanwise pressure measurements.

2. To determine (by means of force measurements for different wing aspect ratios) experimentally how spanwise location of a single hole simulating gunfire damage influences the aerodynamic characteristics of a finite aspect ratio wing.

3. To determine how 2D battle damage aerodynamic data can be applied to finite aspect ratio wings.

4. To verify the results of force, pressure and flow visualisation using CFD simulation.

5. Predict the flow structure characteristics and pressure distributions of a Finite Wing by CFD Simulation and to compare these with experiments.
Flowcharts of Research Sequence: Aerodynamics of Battle-Damaged Finite Aspect Ratio Wing

Figure II.12. A Flowchart of Research Sequence
Chapter 3:
Experimental Design and Set-up

3.1 Introduction

The wind tunnel was open-circuit, closed-throat with 1.92m-wide x 1.32m-high x 3.6m-long working section and contraction-ratio of 7.4:1. This working-section design criterion of maximum velocity 45 m/s could allow representative Reynolds numbers for a simple and or sectional wing to be achieved, which was approximately $1.2 \times 10^6$ to $1.85 \times 10^6$ -- based on wing chord of 400 to 600 mm. However, no such chord dimension had been tested in this test section before the current experiment. With the working section turbulence intensity of 0.15%; and velocity-variation of less than 0.3%, high quality aeronautical research may be expected. The design criteria for 'half models' also allow tests to be performed at higher Reynolds Numbers, and ensure that the point of resolution of the balance can be at the floor of the working section [16].

3.2 Model Design and Manufacture

3.2.1 Mounting Structure

The rigid ‘versatile’ main structure was designed [17] [18] and constructed to support the half-model. This includes the ‘Interface-Block’ that was the ‘heart’ of the experimental rig. This part interfaces the wind tunnel model and the main-structure. The interface-block was positioned in such a way that the ‘balance-virtual-centre’ [25% chord of the model] and the exact positioning of the main structure were in-line with the balance. The magnitude of displacement, slope and stressing of the mounting structure were within acceptable levels and was considered safe enough for the maximum applied load. Figure-III.1 shows schematically the arrangement of the mounting structure, wind tunnel model and the tests section.

3.2.2 Wing Geometry, Design and Mounting Arrangement

As it was decided in Irwin’s 2D investigation [1], all forms of damage would be applied to single wing geometry where plan-form and airfoil section characteristics would be constant throughout all the required testing. The NACA 641–412-airfoil section was chosen as a result of thorough consultation between the previous investigator and the project sponsor.
There were actually some good reasons why this section was chosen [1]. Among these were: the section has well-known characteristics at Low Reynolds numbers; the capacity to provide a sufficient thickness-to-chord-ratio to facilitate internal modelling and pressure tapping; and it could be representative of possible future military airfoils. It had also been previously used by Irwin. [See Appendix-A, taken from Ref. [27]: 'NACA 64-412 Airfoil Section Geometry and Published Data'].

The research model was three-dimensional, zero-taper, zero-twist and the chord was kept the same as for the previous investigation [1] i.e. 200mm. This was to make comparison between 2D and 3D characteristics easier. This finite span model was vertically installed in the wind tunnel and designed for three different aspect ratios (AR6, AR8, and AR10) — defined as the ratio of the
span \(b\) and the average chord \(c\); where for a rectangular wing, the aspect ratio is simply \(AR = \frac{b}{c}\).

A sketch showing the arrangement of panels for the three AR’s is shown in Figure-III.2.

**Arrangement of wing panels of three Aspect Ratios**

<table>
<thead>
<tr>
<th>AR</th>
<th>Wing Chord (c)</th>
<th>Wing Span – incl.Tip (b)</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>200 mm</td>
<td>612 mm</td>
<td>2 (two) 300 mm panels +12 mm tip</td>
</tr>
<tr>
<td>8</td>
<td>200 mm</td>
<td>812 mm</td>
<td>2 (two) 300 mm panels and 1 (one) 200 mm panel+12 mm tip</td>
</tr>
<tr>
<td>10</td>
<td>200 mm</td>
<td>1012 mm</td>
<td>2 (two) 300 mm panels and 2 (two) 200 mm panels+12 mm tip</td>
</tr>
</tbody>
</table>

**Figure-III.2 Arrangement of wing panels of three aspect ratios**

Constraints and requirements that needed to be addressed in the wing design were: first, the model was designed with solid construction; to shape the airfoil section, so called ‘panels’, were made with composite material (See ‘Wing Manufacture’). For reinforcement, 2 rods supported the model structure. The ‘leading-rod’ – positioned exactly at 25% of the model chord was designed to be slightly bigger than the ‘trailing-rod’. The idea of this was because, due to the aerofoil pressure-distribution, more loads occur towards the front of the aerofoil. The 25%c leading-rod location was also intended as a ‘pivot’ for incidences, which was in-line with the balance-virtual-centre. The rod’s length was designed for basic AR6 tests. For AR8 and AR10
tests, two-pairs of different length rods were screwed into each basic rod respectively. The basic rods itself were welded, rather than screwed into the interface-block in respective cylinders. This prevented misalignment and easy dismounting. Four panels (2x 300 mm; and 2x200 mm) were designed to cover all aspect ratios for undamaged and damage tests. To enable the panels' alignment, and to make sure that the connection was smooth, two dowel-pins were provided in each side of every panel. Damaged location in the chordwise position was limited to 50% chord, with the diameter of 20% chord (40mm). This was selected to compare with the previous investigator of 2d test results.

All of the design features above were then carefully calculated for displacement, slope and stressing [20][21][22] including considering the ‘contact stress’ loads on every single panel due to the possible gap between rods and panel. An individual stressing analysis of rods per panel was performed based on estimated elliptic load distribution from the maximum predicted load for every rod. There was a slight twist at the wing tip due to slope differences between the leading-rod and the trailing-rod. However, as predicted, this did not greatly influence the aerodynamic characteristics of the wing. The magnitude of the other calculation -- as mounting structure design -- was within acceptable levels and was considered safe enough for the maximum applied load.

The general assembly and detail-drawings of mounting structure and wind tunnel model were therefore produced for manufacture [See Appendix-B: General Assembly of Experimental Rigs: Mounting Structure and Wind Tunnel Model].

3.2.3 Wing Manufacture

The wing ‘panels’ were manufactured in the AAE Dept. shop facility. The mould -- already available from previous projects [1] -- was made from thermally stable epoxy resin, and was made in two halves; one upper and one lower. However, manufacturing accuracy of the previous project was limited by the resolution of the profile calculation, the milling machine tolerance, and wear on the cutting tool.

To illustrate the accuracy, the previous investigator found that at the centreline maximum thickness location of 40%C, where C is sectional wing chord, the profile thickness definition was 23.924mm, whilst the actual measured value was 23.895mm; a difference of 0.029mm (0.0145%C). This accuracy was therefore considered for the current research model. Two templates made from aluminium that represented the actual wing profile, rod positions (25%C and 66.25%C) and ‘squared hole’ (for pressure-routing outlet) was manufactured using a numerically-controlled [NC] machine. The rods holes were drilled accurately with the aid of a
jig, ensuring their exact location at a distance of 0.4mm from the chord-line. These templates were then placed at the bottom and topsides of the moulds in a vertically position. Dummy-rods were specially produced to connect these two templates as alignment guides. The final condition of the model cast would therefore have accurate hole positions.

3.2.4 Construction of Wing Panels for Pressure Measurement
3.2.4.1 Configuration and Arrangements

Only the AR8 arrangement as shown in Figure-III.2 was selected for pressure measurement. This aspect ratio required three wing panels; where two of the 300mm panels were replaceable for both damage and undamaged configuration. The main interest was to see the pressure distribution in the damage hole and its close spanwise positions.

The model was installed vertically as for force and flow visualisation tests. Figure-III.3 shows the set-up configuration and arrangements for pressure measurements. Based on this figure and the summary in Table-III.1, the panels that had been used for force and flow visualization experiments were modified to accommodate those arrangements. Some constraints had been applied to design and construct the panel modifications. These constraints were as follows:

(i) No pressure measurement was required in lower wing surfaces. This decision was taken by assessing the lower wing pressure distributions found in infinite wing results [1], both for undamaged and damaged configurations.

(ii) Pressure rows were constructed in both outboard and inboard sides of the model panel centreline [Damaged configurations]. The idea was to assess the unsymmetrical flow patterns found in damage hole flow visualization experiments.

3.3 Pre-Tests Verification and Assessment
3.3.1 Safety, Pre-test Planning and Preparation

The objective and the scope of the tests were as follows:

a) Model static displacement tests. The model with its structure was installed on the AAE’s workshop universal milling machine. Model displacements [machine y-axis] were measured by digital gauge for AR6, AR8 and AR10. The magnitude of displacements and slopes were within acceptable levels and was considered safe enough for maximum applied load for wind-on. This was done before the first run was conducted.

b) Model and mounting structure installation setting. It was found that the manufacturing and installation accuracies of the wing-panels and mounting structure assembly gave the model pre-incidence of -0.429° relative to the tunnel centreline. This model-setting error was adjusted to zero and was applied for both undamaged and damaged tests at all aspect ratios.
Figure III.3. Spanwise Arrangement of AR8 for Pressure Measurement

<table>
<thead>
<tr>
<th>Features</th>
<th>Finite Wing AR8</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wing Pressure</td>
<td>Replaceable</td>
</tr>
<tr>
<td>Pressure Rows</td>
<td>7 Rows: Mid-span, 1R, 2R and 3R</td>
</tr>
<tr>
<td>Damaged Wing</td>
<td>for each sides of Damage hole.</td>
</tr>
<tr>
<td>Side of Pressure Tapping</td>
<td>Both Outboard and Inboard sides of Damage hole</td>
</tr>
<tr>
<td>No. of Pressure Tapping</td>
<td>21 and 20 Upper Surfaces</td>
</tr>
<tr>
<td></td>
<td>(at different row)</td>
</tr>
</tbody>
</table>

Table-III.1: Summary of features for pressure measurement
3.4 Testing Techniques
3.4.1 Flow Visualisation

The main objective of flow visualisation was to provide evidence and to gain an understanding of flow structures for flows through the damage hole in a finite aspect ratio wing. In addition, to investigate how the flow visualisation changed compared with the 2D work done previously. The tunnel had never been used for flow visualisation. It is new and because of the presence of balance just under the tunnel floor, it seemed desirable to minimise the number of flow visualisation runs.

3.4.1.1 Flow Visualisation Techniques and Methodology.

Flow visualization is an experimental means of examining surface flow. It involves covering the surface of the wing model with a thin layer of pigment — in current research this was a fine mixture of approx. 10ml TiO₂-Titanium Dioxide mixed with 2ml raw Linseed oil and 30ml Paraffin [23]. This amount of mixture was sufficient for about 10 - 15 runs. The mixture needed to have sufficient viscosity so that it would not flow rapidly under the influence of gravity — because the model was installed vertically. This mixture was then applied evenly across the surface of the model then immediately the tunnel was brought up to a speed of 40 m/s. The tunnel was run until a steady flow pattern emerged. The TiO₂ pigment tends to flow in the direction of the shear stresses at the surface, leaving a pattern of streaks.

In this experiment, the pigment was flowing under the action of ‘jets-in-crossflow’, pressure gradients and under the action of gravity. As can be seen in the photographs later in Chapter-VI, this oil-flow visualization technique proved good for detecting laminar and turbulent separations, wakes, reattachment, and vortices. These flow patterns were recorded using an ‘Olympus – C350’ Digital Camera provided with a zoom lens and 3.2 mega pixels high quality resolutions.

3.4.1.2 Selection of Aspect Ratio and Incidence Range

To limit the number of runs, the flow visualisation programme was concentrated on AR8 and a small number of pre-selected incidences to show the transition from weak to strong jets. The categorisation of weak [4°, 6°] strong [8°, 10°] and transition [8°] jets shown in Table-III.2 is based on the knowledge derived from previous 2D flow visualisation results [1].

<table>
<thead>
<tr>
<th>AR</th>
<th>Damage Locations [from floor]</th>
<th>Flow Structures Visualisation</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>Tip Damage - 652mm</td>
<td>Weak-Jets</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>4°, 6°</td>
<td>10°</td>
</tr>
<tr>
<td></td>
<td>Mid Damage - 452mm</td>
<td>Weak-Jets</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>4°</td>
<td>10°</td>
</tr>
<tr>
<td></td>
<td>Root Damage - 152mm</td>
<td>Weak-Jets</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>4°, 6°</td>
<td>10°</td>
</tr>
</tbody>
</table>

Note: 4°, 6°, etc. are model incidences relative to tunnel flow.

Table. III.2 AR8 Damage Locations and Incidence Range Programme
3.4.2 Aerodynamic Force Measurement

Force measurements were conducted using ‘Aerotech’ six-component under-floor Balance [24], which had undergone full calibration by the manufacturer. The accuracy of all components [Full Model] relative to balance load ranges is shown in Table-III.3. [See also Appendix-C: Hardware Arrangement and Schematic for Data Acquisition System].

<table>
<thead>
<tr>
<th>Component</th>
<th>Balance Load Range</th>
<th>Accuracy [% Full Scale]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Drag</td>
<td>± 120 N</td>
<td>0.010</td>
</tr>
<tr>
<td>Side Force</td>
<td>± 420 N</td>
<td>0.005</td>
</tr>
<tr>
<td>Lift</td>
<td>± 500 N</td>
<td>0.010</td>
</tr>
<tr>
<td>Roll Moment</td>
<td>± 150 Nm</td>
<td>0.010</td>
</tr>
<tr>
<td>Pitch Moment</td>
<td>± 60 Nm</td>
<td>0.010</td>
</tr>
<tr>
<td>Yaw Moment</td>
<td>± 45 Nm</td>
<td>0.015</td>
</tr>
</tbody>
</table>

Table III.3 Balance Components and Accuracy [For typical Full Model]

3.4.2.1 Undamaged and Single Hole Damage Force Tests

The damage type chosen was limited to single-hole, and circular — to simplify the simulated gunfire. The single-hole was arranged so that damaged positions were comparable from one tested aspect ratio to another. A sketch of the spanwise arrangement for every aspect ratio is shown in Figure III. 4. It can be seen that the wing aspect ratio limits the number of damage positions possible, i.e. only 2 positions in AR6, 3 positions in AR8 and 4 positions in AR10. Note that only one exchangeable panel with damage hole was provided.

3.4.2.1.1 Single Hole Damage: AR6

Tests of a single damaged AR6 configuration were done after finishing all undamaged configuration tests. The first single hole — [located at 50% chordwise with 20% chord size] — was made at about 50% of AR6 span. This made the hole damage approximately 152mm distance from the tunnel floor. It needs two 300mm panels for this AR6, and when swapped, made the 2nd damaged hole location at the distance of ±452mm.

3.4.2.1.2 Single Hole Damage: AR8

This AR8 was the most interesting in terms of the number of possible single-hole damage locations. Its position and its characteristics exist between AR6 and AR10. Due to that, this AR8 was chosen as the first priority of the investigation. Unlike AR6, in this AR8, tests of a single damaged configuration were performed interchanged with undamaged configurations. The idea was to see whether its repeatability changed by interchanging the tests with damaged-clean-damaged configurations. This investigation repeated the undamaged runs five times. The results are shown and discussed in Chapter-6, Section 6.1.2.
Spanwise Damage Positions of AR6, AR8, and AR10

<table>
<thead>
<tr>
<th>AR (Actual)</th>
<th>Span (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.21</td>
<td>612</td>
</tr>
<tr>
<td>8.24</td>
<td>812</td>
</tr>
<tr>
<td>10.27</td>
<td>1012</td>
</tr>
</tbody>
</table>

Damage Hole: Constant distance 162mm from the Wing Tip

---

Wing Tip Partition Line
Damage Size and Location: Half Chord- [50% C] Diameter, 20% C
Model Panel Partition Line
Tunnel Floor line with 2mm gap between model root and tunnel floor

---

Figure III.4. Spanwise Arrangement of AR6, AR8, and AR10 for Force Measurement

As for AR6, the first single hole was approx. 152mm distance from the tunnel floor. The second or middle position was at a distance of 452mm. Since only one panel was available with hole damage, to make the 3rd damage location of 652mm from the floor, an additional 200mm panel was inserted between the 300mm panels.
3.4.2.1.3 Single Hole Damage: AR10

This applies for both undamaged and all damaged locations. An additional damage location i.e. 852mm from the floor was made possible by inserting one more 200mm panels into the existing AR8 configuration.

3.4.2.1.4 Effect of Root and Tip Damage on AR6, AR8 and AR10

To get the effects of tip and root damage, the spanwise arrangement of model panels was made to have the possibility to maintain the distance of the hole either from the floor or from the tip for every aspect ratio tested. Figure-III.5 shows the sketch of the root and tip damage locations arrangements.

Figure-III.5 Root and Tip Damage Location Arrangements for AR6, AR8 and AR10.

Tip damaged position in AR6 [452mm from the floor] became mid position in AR8, and the tip damaged position in AR8 [652mm from the floor] become mid position for AR10. Tip damaged position in AR10 [852mm from the floor] has a distance of 162mm from the tip.
3.4.3 Wind Tunnel Corrections

In general, the aim of wind-tunnel tests is to make measurements of aerodynamic quantities under strictly controlled and defined conditions in such a way that, despite the presence of the tunnel walls, the data can be applied to unconstrained flow. The existence of a free-air flow, which is "equivalent" to that in the tunnel, is the fundamental assumption underlying the entire framework of the theory and practice of wind-tunnel wall constraints.

During the initial preparation for data reduction, different wall correction methods were assessed. The methods were AGARD-336 [25], ESDU's 95014 [26] [27] and Maskell's as cited in Barlow [28]. Maskell's method is limited to analytic calculation of the total-blockages ($\epsilon_t$), dynamic pressure ($q_e$) and velocity corrections ($V_e$); whilst the AGARD method is used to define $\alpha_c$, $C_{Lo}$, $C_{Do}$, and $C_{Mc}$.

The ESDU method was finally not applied since results showed that the difference between ESDU and AGARD methods was very small at magnitudes where AGARD showed very small differences compared to uncorrected results. This is believed to be due to the ratio of the model thickness [0.0192 m$^2$] compared to the test section area [2.53 m$^2$] was less than 0.76%. AGARD results therefore were applied to the data reduction.

The assessment of wall correction methods can then be summarised as follows:

<table>
<thead>
<tr>
<th>Wall Correction Methods</th>
<th>Defined Correction Parameters</th>
<th>Remarks</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maskell's</td>
<td>$\epsilon_t$, $q_e$, $V_e$</td>
<td>Applied for Data Reduction</td>
</tr>
<tr>
<td>AGARD</td>
<td>$\alpha_c$, $C_{Lo}$, $C_{Do}$, and $C_{Mc}$</td>
<td>Applied for Data Reduction</td>
</tr>
<tr>
<td>ESDU</td>
<td>$\alpha_c$, $C_{Lo}$, $C_{Do}$, and $C_{Mc}$</td>
<td>Not Applied for Data Reduction</td>
</tr>
</tbody>
</table>

The application of formulas used in the applied wall corrections are described in Appendix-D: 'Applied Formulas and Equations for Wall Corrections'. The final results of Maskell's blockage correction, defined from Eq.[1] to Eq.[6], and the AGARD 'Upwash Correction' defined from Eq.[14] to Eq.[18] shown in Appendix-D are summarised in Tables III.4 and III.5.

3.4.4 Experimental Data Validation, Accuracy and Repeatability

3.4.4.1 Force Tests Hysteresis and Repeatability

Repeatability assessment of all available runs for every aspect ratio was limited to the incidence range between -4° to 8° to avoid 'stall-separated-scatter' data. Figure III.6a shows typical $C_D$ vs. Incidence repeatability and scatter check of AR8 configurations, where it shows that the repeatability and the scatter were within 3 to 5 'drag-counts' [$dc$] in that incidence range [$1 \, dc = 0.0001$].
### Table-III.4 Summary of Applied Blockage Correction [Maskell's]

<table>
<thead>
<tr>
<th>AR</th>
<th>$\varepsilon_{ab}$</th>
<th>$\varepsilon_{wb}$</th>
<th>$\varepsilon_{T}$</th>
<th>$q_{c}$</th>
<th>$V_{c}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>0.0003708</td>
<td>1.397e-04</td>
<td>5.105e-04</td>
<td>1.00102 (q)</td>
<td>1.00102 (V)</td>
</tr>
<tr>
<td>8</td>
<td>0.0004972</td>
<td>1.575e-04</td>
<td>6.547e-04</td>
<td>1.00131 (q)</td>
<td>1.00131 (V)</td>
</tr>
<tr>
<td>10</td>
<td>0.0006286</td>
<td>1.794e-04</td>
<td>8.0809e-04</td>
<td>1.00162 (q)</td>
<td>1.00620 (V)</td>
</tr>
</tbody>
</table>

### Table-III.5 Summary of Applied Upwash Correction [AGARD]

<table>
<thead>
<tr>
<th>AR</th>
<th>$\alpha_{e}$</th>
<th>$CL_{e}$</th>
<th>$CD_{e}$</th>
<th>$CM_{e}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>$\alpha_{e} + [0.0116(CL_{w})]$</td>
<td>$[CL_{w} \cos(0.12 (CL_{w})/2.5344 (0.235)) - CD_{w} \sin(0.12(CL_{w})/2.5344 (0.235))]$</td>
<td>$[CD_{w} \cos(0.12 (CL_{w})/2.5344 (0.235)) + CL_{w} \sin(0.12(CL_{w})/2.5344 (0.235))]$</td>
<td>$CM_{e} + [4.74e-6(CL_{w})]$</td>
</tr>
<tr>
<td>8</td>
<td>$\alpha_{e} + [0.0171(CL_{w})]$</td>
<td>$[CL_{w} \cos(0.16 (CL_{w})/2.5344 (0.26)) - CD_{w} \sin(0.16(CL_{w})/2.5344 (0.26))]$</td>
<td>$[CD_{w} \cos(0.16 (CL_{w})/2.5344 (0.26)) + CL_{w} \sin(0.16(CL_{w})/2.5344 (0.26))]$</td>
<td>$CM_{e} + [6.84e-6(CL_{w})]$</td>
</tr>
<tr>
<td>10</td>
<td>$\alpha_{e} + [0.0216(CL_{w})]$</td>
<td>$[CL_{w} \cos(0.20 (CL_{w})/2.5344 (0.265)) - CD_{w} \sin(0.20(CL_{w})/2.5344 (0.265))]$</td>
<td>$[CD_{w} \cos(0.2 (CL_{w})/2.5344 (0.265)) + CL_{w} \sin(0.2(CL_{w})/2.5344 (0.265))]$</td>
<td>$CM_{e} + [8.95e-4(CL_{w})]$</td>
</tr>
</tbody>
</table>
Good repeatability and scatter is also shown in Figure III.6b of $C_M$ vs. $\alpha$ of AR8 for Undamaged and all Damage Location configurations [Exaggerated scale]. It shows similar repeatability for both the damaged and undamaged cases.

The summary repeatability coefficients found in all three Aspect Ratios for undamaged configurations were within the acceptable boundaries as shown in Table III.6.

<table>
<thead>
<tr>
<th>AR</th>
<th>Coefficient Tolerances - Maximum Scatters [Error Bands] [Incidence range -4 to +8deg]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$C_L$</td>
</tr>
<tr>
<td>6.2</td>
<td>±0.004</td>
</tr>
<tr>
<td>8.2</td>
<td>±0.005</td>
</tr>
<tr>
<td>10.2</td>
<td>±0.008</td>
</tr>
</tbody>
</table>

Table III.6: Summary of Repeatability and Maximum Scatter of Aero Coefficients

3.4.4.2 Baseline [datum] Selection

Repeated runs for all aspect ratios for both undamaged and damaged configurations were averaged to get an applied data. The undamaged averaged data was then applied as a 'baseline' [datum] for the given Aspect Ratio.

As describe previously, this datum is very important to determine the incremental changes of the aerodynamics coefficients due to damage.

The averaged data of all aerodynamic coefficients for AR6, AR8 and AR10 were then compared to two-dimensional 'Irwin's 2D', which was then converted to 3D for the respective Aspect Ratios. Figure III.7a below shows a typical lift-curve of averaged results for the three Aspect Ratios of AR6, AR8 and AR10.

Baseline [datum] definition of $C_D$ vs. $C_L$ for all aspect ratios is shown in Figure III.7b. It can be seen that the $C_D$s were not really collapsed into the same value.

Figure III.7b shows a $C_D$ difference of ~5dc between AR8 and AR10, and a difference of ~15dc between AR6 to AR8 [See also Table III.9]. These differences were believed to be due to changing and swapping the panels between the different Aspect Ratios tested where a model setting relative to the incoming flow slightly changed — likely due to screws and 'dowel-pin' connection. There is another concern — though was considered relatively small, that the floor boundary layer of approx. 10mm has a disproportionately large effect on the AR6 model.

Figure III.7c shows baseline [datum] of pitching moment vs. incidence with $C_{Mo}$ difference of ~0.002 between AR8 and AR10, and a difference of less than 0.002 between AR6 to AR8. However, all Aspect Ratios almost collapsed on to the same point at negative incidences.
CD vs. Incidence - AR8
All Clean [Undamaged] Tests vs.
Damaged Location: 450mm, 650mm from tunnel floor
Drag Repeatability - 'Scattered Check' [Exaggerated]
CM vs. Incidence - AR8
All Clean [Undamaged] Tests vs.
Damaged Locations: 450mm, 650mm from tunnel floor
Pitching Moment Repeatability - 'Scattered Check'[Exaggerated]

Figure-III.6b.
CL vs. Incidence
Configuration AR6, AR8, AR10
[All Undamaged Averaged vs. Irwin’s 2D Results]

Figure III.7a Lift Curve as applied baseline for AR6, AR8, and AR10
Figure III. 7b Drag-polar averaging as applied baseline for AR6, AR8 and AR10
Figure III. 7c Pitching Moment vs. Incidence averaging as applied baseline for AR6, AR8 and AR10
3.4.4.3 Baseline-Undamaged Configuration

(i) 3D-Slope Prediction. A lift-curve-slope prediction of incompressible flows, rectangular, for a finite aspect ratio wing, was conducted based on 'Ravaglia's formula [29]. The results were summarised in Table-III.7.

<table>
<thead>
<tr>
<th>AR</th>
<th>$\frac{\partial \alpha}{\partial \alpha}$ [per Rad]</th>
<th>$\alpha_0$ [Rad]</th>
</tr>
</thead>
<tbody>
<tr>
<td>6</td>
<td>0.0666</td>
<td>0.1865</td>
</tr>
<tr>
<td>8</td>
<td>0.07507</td>
<td>0.2102</td>
</tr>
<tr>
<td>10</td>
<td>0.08088</td>
<td>0.22647</td>
</tr>
</tbody>
</table>

Table-III.7 Results of Slope Prediction Formula

Ravaglia's formula for 3D slope prediction shown in equation [19] below was selected, as it was closest to the experimental results. Figures III.8a, 8b and 8c shows the slope prediction of AR6, AR8 and AR10$^1$ compared with all available tests for undamaged configurations.

\[
\alpha_0 = \frac{a_0 \cdot AR}{\frac{\partial \alpha}{\partial \alpha}} \text{ [per radian]} \quad \ldots [7]
\]

where $a_0 = \text{Irwin's 2D actual value of 0.1}^0$  
$\Lambda_H = 0^0$; and $M_c = 0.1176$ [40 m/s]

A summary of the actual values for these corrections and uncertainty factors plus model panel conditions were shown in Table III.8.

(ii) $C_{L_{ao}}$ Comparison. Figure III.8a, 8b and 8c respectively shows comparison with 'Irwin's 2D' of the NACA 641-412 airfoil that tested at Re. $5.0 \times 10^5$. The Irwin's lift-curve-slope, $\frac{\partial C_l}{\partial \alpha}$ [quoted as 0.1000] and the zero-lift-incidence, $C_{L_{ao}}$ of published data [quoted as $-2.8^0$] shows differences to that was found in the current experiments.

---

$^1$ The actual physical dimensions, calculation and for data reductions of the model are AR6.21, AR8.24 and AR10.27. All of these Aspect ratios are written and simplified as AR6, AR8 and AR10.
Figure 11.8a: Typical Slope Predictions and Lift-Curve Repeatability of all Available Runs for AR6
**CL vs. Incidence**
Undamaged Slope Repeatability - AR8
[vs. Irwin's 2d Results, vs. 3D Slope Prediction]

![Graph showing CL vs. Incidence for AR8 with experimental slopes and predictions.](image-url)

- Irwin's 2D Results
- 1st Clean AR8
- 2nd Clean AR8
- 3rd Clean AR8
- 4th Clean AR8 [after Dam 150 Tests]
- 5th Clean AR8 [after Dam 650 Tests]

**Figure III.8b:** Typical Slope Predictions and Lift-Curve Repeatability of all Available Runs for AR8
CL vs. Incidence
Undamaged Slope Repeatability - AR10
[vs. Irwin's 2d Results, vs. 3D Slope Prediction]

Figure III.8c: Typical Slope Predictions and Lift-Curve Repeatability of all Available Runs for AR10
These differences were believed to be due to the following corrections: first, correction of the incidence setting of the model chord-line relative to the tunnel centreline -- as mentioned previously and included in all test results; secondly, corrections due to tunnel flow angularity where calibration had shown -0.3125°. Third, flow and tunnel condition uncertainties such as gust and turbulent intensity, of approximately ± 0.1°. Table-III.8 shows the actual $C_{L_{\infty}}$ after the corrections were applied.

<table>
<thead>
<tr>
<th>AR</th>
<th>Corrections and Uncertainties:</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Model Setting $= -0.429'$</td>
</tr>
<tr>
<td></td>
<td>Flow Angularity $= -0.3125'$</td>
</tr>
<tr>
<td></td>
<td>Flow Uncertainties: $\pm 0.1^\circ$</td>
</tr>
<tr>
<td>6.2</td>
<td>$C_{L_{\infty}}$ published</td>
</tr>
<tr>
<td></td>
<td>$\Delta C_{L_{\infty}}$</td>
</tr>
<tr>
<td></td>
<td>$-2.82^\circ$</td>
</tr>
<tr>
<td></td>
<td>$-0.27^\circ$</td>
</tr>
<tr>
<td></td>
<td>$-2.55^\circ$</td>
</tr>
<tr>
<td>8.2</td>
<td>$C_{L_{\infty}}$ published</td>
</tr>
<tr>
<td></td>
<td>$\Delta C_{L_{\infty}}$</td>
</tr>
<tr>
<td></td>
<td>$-2.82^\circ$</td>
</tr>
<tr>
<td></td>
<td>$-0.12^\circ$</td>
</tr>
<tr>
<td></td>
<td>$-2.70^\circ$</td>
</tr>
<tr>
<td>10.2</td>
<td>$C_{L_{\infty}}$ published</td>
</tr>
<tr>
<td></td>
<td>$\Delta C_{L_{\infty}}$</td>
</tr>
<tr>
<td></td>
<td>$-2.82^\circ$</td>
</tr>
<tr>
<td></td>
<td>$-0.17^\circ$</td>
</tr>
<tr>
<td></td>
<td>$-2.65^\circ$</td>
</tr>
</tbody>
</table>

Table III.8: Summary of Actual $C_{L_{\infty}}$ and Compared to Published Data.

(iii) 3D Drag and Moment Convert to Aspect Ratio.

The $C_L$ was applied to the formula given by McCormick [39] to predict $C_D$ [Equation 20], which depends on functions of 'profile drag' $C_{D_0}$, induced drag factor 'k' and the Aspect Ratio. This 'k' was a very important factor for obtaining an accurate 3D conversion.

$$C_D = C_{D_0} + \frac{C_L^2}{\Pi e},$$

where 'e' - 'Oswald's' factor.

The Oswald's factor, 'e' for AR6, AR8 and AR10 are needed to define 'k' that was computed from the ESDU 74035 [32] data-sheet computer program.

In addition to that, a slope gradient of 'k' was defined using $C_D$ vs. $C_L^2$ plot. The plot was made based on a particular test where the incremental change of incidence in the $C_{L_{\infty}}$ region was minimised to $\Delta \alpha = 0.1^\circ$. The idea was to get accurate 'k' data.
Table III.9 below shows a comparison summary of actual $C_{D0}$ and the value of ‘$k$’ that derived from the $C_D$ vs. $C_L^2$ plot compare to the value ‘$k$’ derived from ‘$e$’ [ESDU].

<table>
<thead>
<tr>
<th>AR</th>
<th>$C_{D0}$</th>
<th>‘$k$’ - Based on ESDU</th>
<th>‘$k$’ - Based on $C_n$ vs. $C_L$ Expr. plot</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.21</td>
<td>0.0150</td>
<td>0.0539</td>
<td>0.06015</td>
</tr>
<tr>
<td>8.24</td>
<td>0.0135</td>
<td>0.04090</td>
<td>0.04564</td>
</tr>
<tr>
<td>10.27</td>
<td>0.0130</td>
<td>0.03314</td>
<td>0.0369</td>
</tr>
</tbody>
</table>

Table III.9 Summary of 3D Conversion Parameters Coefficients

It shows that $k$ is higher for experiments. This was believed to be due to imperfect tip design and the tunnel floor boundary layer.

Figure III.9a, 9b, and 9c of $C_D$ vs. $C_L$ of AR6.2, AR8.2, and AR10.2 in the following pages show the 2D results predicted and converted into those three aspect ratios using the $k$ values from experiments. For illustration, the figures were also compared to the $k$ values from ‘ESDU and averaged undamaged of the three aspect ratios.

The idea of comparing the current experimental results with Irwin’s 2D was to demonstrate and to show the accuracy of the undamaged data. However, $C_D$ – $C_L$ plots shown in Figure-III.9a – III.9d suggest relatively large differences between the current experimental results and those of Irwin. The main reason was believed to be due to the floor boundary layer and due to joints on the model panels. The investigation into the battle damage was carried-out, however, because the interest was in the increments.

To predict and convert ‘Irwin’s 2D’ $C_m$ to $C_M$ values, another ESDU data-sheet i.e. ESDU 87001 [33] was applied. The approach with a good approximation was that the wing camber-line is unchanged across the span.

‘ESDU 87001’ was also used to convert the $C_m$ into AR6, AR8 and AR10. The conversion was given by:

$$ (C_{M,2D}) = \frac{2AR}{(2AR + 1)} \cos \Lambda_{1/2} \{F(C_{m0})\}_{unh} $$

Where $\{F(C_{m0})\}_{unh} = (C_{m0})_{unh} - (C_{m0})_n$ defined from 2D results [if the camber line is unchanged across the wingspan].

Figure III.10 is a typical $C_M$ vs. Incidence of the 2D results that converted to AR8. The plot also compared to the averaged undamaged AR8 results. It can be seen that the predicted $C_M$ was good in all incidence regions, and the actual measurement for the undamaged configuration were of right compared to the 2D result and the 3D prediction.
CD vs. CL for AR6
[All Available Clean Tests Averaged vs. Irwin’s 2D and Convert to AR6 based on ‘k-ESDU’ and ‘k-exprplot’]

Figure III. 9a.
CD vs. CL AR8

[All Available Clean Tests Averaged vs. Irwin's 2D and Convert to AR8 based on 'k-ESDU' and 'k-exprplot']

Figure III. 9b.
CD vs. CL AR10
[All Available Clean Tests Averaged vs. Irwin's 2D and Convert to AR10 based on 'k-ESDU' and 'k-exprplot']

Figure III. 9c
CM vs. Incidence AR8
[All Available Undamaged Tests Averaged vs. Irwin's 2D and Convert to AR8 based on 'ESDU']

Figure III. 10.
3.4.5 Pressure Data Accuracy: Pressure Distribution Repeatability

Pressure coefficient repeatability tests were conducted for the damaged case, for all pressure tapping locations. The error was defined as the difference between \( Cp \) values recorded at the same tapping location on two to three independent test runs. Values were recorded at incidences of \( 4^\circ \) and \( 8^\circ \).

Figure-III.11a and 11b illustrates the repeatability of the results, for this example, taken along the ‘3R’-outboard and ‘1R’-inboard at incidence \( 4^\circ \) for wing damage located at the tip. Whilst for \( 8^\circ \) repeatability, taken along the ‘centreline’ and ‘1R’-outboard for tip damage. The selected pairs of spanwise distances were measured in the same wind tunnel run. Repeatability tests were conducted over three runs for incidence \( 4^\circ \) and two runs for incidence \( 8^\circ \).

Table-III.10 shows the summary of the results that were taken for the largest \( Cp \) error over all 40 tapping locations. The larger errors were at the leading edge area [4.25%C], and at the first tapping after the downstream edge of the damage hole [62%C – Inboard and 65%C-Outboard].

<table>
<thead>
<tr>
<th>Incidence</th>
<th>Centreline</th>
<th>1R-Inboard</th>
<th>1R-Outboard</th>
<th>3R-Outboard</th>
<th>62%C [Inboard]</th>
<th>65%C [Outboard]</th>
</tr>
</thead>
</table>
Pressure Measurement - Repeatability Runs
Tip Damaged - Alpha 4 deg. [3R - OB and 1R - IB]
[Note: deleting tappings no. 40, 30 and 31]

Pressure Measurement - Repeatability Runs
Tip Damaged - Alpha 8 deg. [CL-IB and 1R-OB]
[Note: by deleting tap no. 20 and 46]

Figure III. 11a and 11b Typical -Cp vs. X/C Profiles Repeatability
[Configuration-AR8, Alpha 4 deg. at 1R-IB and 3R-OB]
Chapter 4:

Review of Infinite ['2D'] Battle Damaged Wing and Comparison with CFD Simulation

4.1 Introduction

Tremendous concurrent synergy growth during the past thirty years in computers (speed, memory, parallel architectures), in numerical algorithms, in hardware and software for handling complex geometries and for visualising massive numerical data has made us believe that Computational Fluid Dynamics (CFD) can perhaps answer everything that is needed in design and development involving fluid flows.

In industry, sometimes they are operating on a very tight budget, therefore, they are always on the look out for ways to save time and reduce project risk. CFD, and other simulation and modelling techniques allow them to do this.

In current investigations, CFD simulation has been performed to compare and confirm what was found experimentally in the infinite wing case ['2D']. By reviewing and simulating the '2D' experimental results, it will help to predict the 3D case. Another objective is to understand the physical phenomena and characteristics of the flow, which is cheaper and faster using CFD compared with experiments.

The cases that have been done for the predictions were as follows:
1. Baseline: Undamaged Simulation
2. Damaged Wing [Single Damage, 20% damage size at 25% location]

4.2 The Geometry and Grids Generation

The geometry of NACA 641-412 was taken from a public domain airfoil database -- Public Domain Aeronautical Software – PDAS [34], and then checked against the published data found in the literature [18]. The results were then compared with the airfoil section found in Ref. [1]. This geometry was exported to a pre-processor called ‘Gambit v.2.0’ [35] – software package to build and mesh models for CFD purposes. Gambit receives user input by means of its graphical user interface (GUI). The refined airfoil geometry was then exported to ‘Solid-Edge v.11’ – a
CAD software package -- to define the solid model [17]. The idea was to make the volume modelling easier and faster. In Solid-Edge, the wing profile was created by means of 'extrusion' to generate the entire wingspan and to create 'cut-out' representing the damage hole in the wing at pre-defined incidences. This solid volume was then re-exported to Gambit to create the mesh and to define the 'boundary-layer' feature - a term used to define the spacing of mesh node rows in regions immediately adjacent to edges and faces. The final mesh was then exported to a commercial CFD package, 'Fluent 6.0', for processing and post-processing [36] [37].

The grid for the 2D undamaged wing [after 'adaptation' – mesh/grid refinement] in a two-dimensional freestream boundary condition [BC] is shown in Figure-IV.1; whilst Fig.-IV.2, showing the 'infinite-wing' model with damage hole, is installed between the tunnel walls -- also in a three-dimensional freestream boundary condition. Grid adaptation was employed to get a better solution. The size of the computational domain was [4.0 x 4.0 x 0.45] m. The 0.45m refers to the tunnel width. For this CFD simulation, the height to simulate the floor and the roof of tunnel was set to 4.0m. This is done to avoid wall effects.
4.3 Grid Refinement Study

For the grid refinement study, the selected incidences were set to $-2.8^\circ$, corresponding to $C_{Lo}$, and $16^\circ C_{Lmax}$ onset. However, grid creation was not necessary for all cases as they could be represented by a zero-incidence grid and the incidences defined by changing the flow direction.

Beside 'inlet' and 'outlet', a ‘symmetry’ boundary condition was used for the right and left sides of the wing. The so-called ‘free stress’ boundary condition was employed [in Fluent’s terminology this was defined as a ‘pressure far-field’ boundary condition]. The turbulence intensity $(T_i)$ was set to 0.15% based on AAE’s wind tunnel calibration data.

The 3D steady segregated, implicit solver was employed to solve the governing equation; and the ‘Spalart- Allmaras’[S-A] turbulence model was chosen. This is because the S-A model is the most economical model for the purpose of a grid refinement study.

Results from Fluent are arranged in predefined tables for row data, so that grid types, number of cells, and converged iteration can be analysed easily. The number of cells ranged from 269,000 to 690,000 and the solution required from 348 to 1200 iterations to converge. The fastest converged solution was for an incidence of $-2.8^\circ$, which also had the smallest mesh size.

To differentiate the effect of mesh size, pressure distribution plots were created for $-2.8^\circ$ [to represent the undamaged wing configuration] and $16^\circ$ [to represent the damaged wing configuration]. The typical plots are shown in Figure-IV.3a and Fig-IV.3b. It can be seen from the two graphs that using different numbers of cells would provide slightly different peaks for the upper and lower surface of the wing leading edge – using more cells would provide a higher peak $C_{ps}$. However, these peak differences actually resulted in very small effects on aerodynamic coefficients. During the mesh refinement studies, it was found that after increasing to a certain number of cells, the results would not improve anymore. 690,000 was then chosen as the number of cells for the study.

4.4 Selection of Turbulence Models for CFD Simulation

It is an unfortunate fact that no single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model will depend on [44] considerations such as the physics involved in the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources and the amount of time available for the simulation. To make the most appropriate choice of model application, then the capabilities and limitations of the various options need to be understood.
Figure IV.3a. Cp vs. X/C Alpha -2.8° - Mesh/Grid Size Comparison
[-2.8° to represent the Undamaged wing configuration]

Figure IV.3b. Cp vs. X/C Alpha +16° - Mesh/Grid Size Comparison
[16° to represent the Damaged wing configuration]
Turbulent flows are significantly affected by the presence of walls. The near-wall modelling significantly affects the reliability of the numerical solutions; therefore, accurate representation of the flow in the near-wall region determines the successful prediction of wall-bounded turbulent flows.

4.4.1 Model Comparisons

Based on recommendations from the literature, in terms of computation, the Spalart-Allmaras model is the least expensive turbulence model provided in Fluent, since only one turbulence transport equation is solved.

The standard k-epsilon model clearly requires more computational effort than the Spalart-Allmaras model since an additional transport equation is solved. The ‘realizable’ k-epsilon model requires only slightly more computational effort than the standard k-epsilon model. Computations with the ‘RNG’ k-epsilon model tend to take 10-15% more CPU time than with the standard k-epsilon model. Like the k-epsilon models, the k-Omega models are also two-equation models, and thus require about the same computational effort [38].

Compared with the k-epsilon and k-Omega models, the Reynolds Stresses Model - RSM requires additional memory and CPU time due to the increased number of the transport equations for Reynolds stresses. However, efficient programming in Fluent has reduced the CPU time per iteration significantly. On average, the RSM in Fluent requires 50-60% more CPU time per iteration compared to the k-epsilon and k-Omega models. Furthermore, 15-20% more memory is needed [38]. Aside from the time per iteration, the choice of turbulence model can affect the ability of Fluent to obtain a converged solution.

It is a huge task to compare the performance of all turbulence models in the present study. By considering the above constraints therefore, only the Standard k-epsilon and Spalart-Allmaras models will be evaluated. The reasons for choosing these two models were as follow [36] [37]:

- The Spalart-Allmaras model was designed specifically for aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients. It is also gaining popularity for turbo machinery applications.

In Fluent however, the Spalart-Allmaras model has been implemented to use wall functions when the mesh resolution is not sufficiently fine. This might make it the best choice for relatively crude simulations on coarse meshes where accurate turbulent flow computations are not critical.
The simplest "complete models" of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard k-Epsilon model in Fluent falls within this class of turbulence model [RANS] and has become the workhorse of practical engineering flow calculations. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations.

4.4.2 Turbulence Models Evaluations and Findings
The k-epsilon model was tested along with the Spalart-Allmaras model, and converged solutions were obtained after 200 to 400 iterations with both models. In applying these models to the damaged wing problems, it was found that significant 'jet-in-crossflow' features were predicted well by both the Spalart-Allmaras and the k-epsilon models, particularly in the inner damage hole region.

4.5 Testing of K-e and SA Models
4.5.1 Demonstration of Flow Structures
CFD simulations with K-e and SA models have been done for both weak and strong jets flow structures. Figure-IV.4a to Fig-IV.4d show weak-jet results, whilst Figures IV.5a to IV.5d shows the strong-jet results of those two models respectively.

4.5.1.1 Weak-jet Flow Structure.
Fig.-IV.4a and IV.4d, show vectors of velocity magnitude using K-e and SA models at a typical weak-jet incidence of 2°. Figures-IV.4a and IV.4b show an infinite wing model with three-dimensional flow in a '2D' test set-up, where the wing is installed between two tunnel walls. It can be seen that, despite the differences shown in flow structures, the K-e and SA models also produce different velocity magnitudes on the wing surfaces and at the tunnel walls.

The SA model [Fig.-IV.4a] predicts a lower static pressure that produces a higher velocity compared to the K-e model [Fig.-IV.4b] on the wing and walls region. These differences can also be seen in the vector colours of both models where the wing on the SA model produces more speed on the upper wing surface compared to the K-e model.

The differences in pressure and velocity on the wing upper surfaces also affect the structures of the flow behind the damage hole. It shows that the K-e model produce a longer, straight, parallel, and more visible wake compared to the SA model. This is believed to be due to the difference of velocity and pressure of the jet at the exit hole. The situation of the exited weak-jets for both models can be seen in Fig.-IV.4c and IV.4d.
Figure IV.4a. Weak Jet on Upper Surface Wing with Tunnel Walls

Figure IV.4b. Weak Jet on Upper Surface Wing with Tunnel Walls

K-e Model. Alpha +2deg.

K-e Model. Alpha +2deg.
Figure IV.4c. Weak Jet Flow Structure at Damage Centre-line
S-A Model. Alpha +2deg.

Figure IV.4d. Weak Jet Flow Structure at Damage Centre-line
K-e. Alpha +2deg.
In general, both models show similarity for the main weak-jet characteristic, i.e. an attached jet at the exit of the damage hole. The similarity also shows in the jet velocity where the magnitudes cannot be differentiated, however, the K-e model seems to show a stronger jet at the exit compared to the SA model. This can be seen in the height of the jet boundary, which the K-e model shows to be higher. This situation affects the shape and formation of the wake as seen in Figures IV.4a and IV.4b.

4.5.1.2 Strong-jet Flow Structure.

In Fig.-IV.5a and IV.5b, vectors of velocity magnitude using K-e and SA models at the damage hole centreline are shown with the incidence of 8°. It can be seen that there is a similarity of flow structure in both models, i.e. a detached jet at the exit of the damage hole, reverse flow downstream of the wing and also regions of stagnation pressure at the upstream and downstream sides of the damage hole cavity. However, the magnitudes of the velocity vectors from the two models are slightly different.

By analysing the vector colours, it also shows that the lowest velocity in the stagnation areas and at the centre of the reverse flows is approximately 0.257 m/s by the K-e model, whilst using the SA model give a magnitude of approximately 0.153 m/s — only 0.1 m/s difference. The highest velocity magnitude in both models occurs upstream of the damage hole close to the leading edge. At the jet exit, it shows a magnitude of approximately 41.2 m/s by K-e and 47.7 m/s by the SA model. This gives a jet to freestream velocity ratio of $[V_j/V_\infty]_{K-e} = 1.03$ and $[V_j/V_\infty]_{S-A} = 1.119$ respectively [assuming the average $V_\infty$ in both models is 40 m/s]. Furthermore, a slightly different wake patterns can be identified in the upper surface wing velocity vectors shown in Figure-IV.5c from the K-e model, and Figure-IV.5d for the SA model. Table-IV.1 shows the main differences of both models.

In these strong jet comparisons, it was also investigated that there were no significant changes in the flow structure characteristics of both models when the incidence is changed.

It can be seen that looking closely to the features condition on both models, a preference can easily be made. However, before selection is made, a further comparison i.e. $C_p$ vs. $X/C$ should be made, which is described in the flowing section.

<table>
<thead>
<tr>
<th>Features</th>
<th>K-Epsilon Model</th>
<th>Spalart-Allmaras Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>CRVP Formation</td>
<td>Formed at mid-hole shape, unclear</td>
<td>Formed at mid-hole shape, clear</td>
</tr>
<tr>
<td>Forward Separation</td>
<td>Irregular - unclear</td>
<td>Regular - clear</td>
</tr>
<tr>
<td>Secondary Separation</td>
<td>Not formed - unclear</td>
<td>Formed - clear</td>
</tr>
<tr>
<td>Horseshoe Vortex</td>
<td>Formed, thin</td>
<td>Formed, thick, clear</td>
</tr>
<tr>
<td>Entrainments</td>
<td>Not strong</td>
<td>Strong</td>
</tr>
<tr>
<td>Wake Symmetrical</td>
<td>Symmetric</td>
<td>Symmetric</td>
</tr>
<tr>
<td>Reverse Flow and Separation</td>
<td>Formed visible</td>
<td>Formed visible</td>
</tr>
<tr>
<td>at hole downstream</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table-IV.1. Strong-jet Features Comparison
Figure IV.5a. Strong-Jet Flow Structure at Damage Centre-line
K-ε Model, Alpha +8deg.

Figure IV.5b. Strong-Jet Flow Structure at Damage Centre-line
S-A Model, Alpha +8deg.
Velocity Vectors Colored By Velocity Magnitude (m/s)
Upper Surface, Alpha + 8deg, n 600 Iteration

**Figure IV.5c.** Strong-Jet Flow Structure on Upper Surface
*K-e Model.* Alpha + 8deg

Velocity Vectors Colored By Velocity Magnitude (m/s)
Upper Surface, Alpha + 8deg, n 600 Iteration

**Figure IV.5d.** Strong-Jet Flow Structure on Upper Surface
*S-A Model.* Alpha + 8deg
4.5.2 K-e and SA Comparison: Cp vs. X/C

The comparison between the damage hole centreline Cp profiles at 8° by both K-e and SA models and the experimental results are shown in Figure-IV.6. It can be seen that from the leading edge to the damage hole, the results from the SA model are much closer to the experimental results on both the upper and lower surfaces.

Downstream of the damage hole, the SA model gives much higher values at about 40 to 60% of the chord compared with the experimental data, but the agreement is better towards the trailing edge. On the contrary, the K-e model results are following the experimental profile better with slightly smaller values than the experimental data. However, the K-e model looks almost 'constant', whilst the SA model shows variation similar to those found experimentally.

At the lower wing surface, downstream of the damage hole, all four curves show similar values, however, K-epsilon is poor upstream of the damage.

What may be summarised in this Cp vs. X/C graph is that the S-A model is closer to the experimental results in terms of Cp values. The S-A model is better forward of the damage whilst the K-e model is better behind the damage.

4.5.3. The Selection

Having analysed what has been described in weak and strong-jets flow structures [Figure-IV.4a to Fig-IV.4d] and [Figure-IV.5a to Fig-IV.5d], features in Table-IV.1, and Cp vs. X/C in Figure-IV.5a, the selection can then be made. Although K-e seems to have produced results with more similarity to the experimental ones behind the damage, the SA model will be chosen for further CFD simulation. However, this does not imply that agreement forward of the damage is more important; because when comparing the simulation shown in Table IV.1, the SA model shows more similarity in all aspects. The SA model gives more realistic results than the K-e model.

In term of the strong-jet characteristics and the flow structures, the SA model performed better. [See further validation of the SA model with experimental results in section 4.6].

In addition, having perfected the model grid used for these simulations, the author was convinced that the S-A model should be selected, because the assessment from Fluent showed that the S-A model was the best choice for relatively crude simulations on coarse meshes where accurate turbulent flow computations are not critical and where mesh resolution is not sufficiently fine.
4.6 CFD Simulation vs. ‘2D’ Experimental Results: The Synergy
[Using the selected Model]

4.6.1 Introduction

The synergy between wind tunnel experiments and CFD has long been recognised in aerodynamic research and these methods have been practiced in parallel, complementing each other. A particular gap may emerge with the rapid development of CFD simulations.

The purpose of this section is to discuss in more detail the synergy of the available 2D battle damaged wing experimental results done previously [1] and CFD simulations. In particular how the CFD helps to gain and guide the understanding of the experimental work. Usually, experimental results are used to provide reference data for validating models and methods used in CFD simulations. At the present, the discussion is the reverse i.e. is that the battle damaged wing CFD simulations -- though not promoting new methods or models, focus on problems already investigated experimentally, thus replicating experiments. The idea was to verify the theory or model and to provide a proof of accuracy, but producing more data and new information, some of which known is inaccessible to experiments. It is this kind of feedback between experimental and simulation that has shown for synergy.

4.6.2 CFD Simulation vs. ‘2D’ Experimental Results Comparison

All CFD simulations that are used for comparisons with experimental results have identical Reynolds number, mesh and grids, turbulent intensity factor, and other pre-selected SA model parameters. The parameters are not changed except the required flow direction that is needed for wing incidences. The list of values and main parameters are as follows:

- **Density [kg/m^3]** : 1.225
- **Pressure [Pascal]** : 101325
- **Viscosity [kg/m-s]** : 1.7894 e-05
- **Mach Number** : 0.1
- **Reynolds Number** : 5.0 x 10^5
- **Temperature [K]** : 288
- **Turbulent Specification Method**
  - **Turbulent Intensity [%]** : 0.4
  - **Turbulent Length Scale [mm]** : 140
- **Solution Initialisation:**
  [Typical for Incidence +8°]
  - **Pressure [Pascal]** : 101325
  - **X-Velocity [m/s]** : 36.15266
  - **Y-Velocity [m/s]** : 4.5555531
  - **Z-Velocity [m/s]** : 0
$C_p$ vs $x/c +8$ deg.

K-e vs. SA: Turbulence Model Comparison
[CFD Simulation vs. Experimental Benchmark]

Figure IV.6 – $C_p$ vs. $x/C$: K-e and SA Comparison Alpha +8 deg [Experimental data as a Benchmark]
4.6.2.1 Undamaged and Damaged Wings: Aerodynamic Coefficients Comparison

Figure IV.7a and Figure IV.7b respectively compares aerodynamic coefficients between CFD predictions for undamaged and damage wings -- all using the S-A model with Irwin’s ‘2D’ experimental data. All the cases with different incidences ranging from $-4^\circ$ up to $+14^\circ$ were computed using grids for the undamaged wing [shown in Figure-IV.1] and damaged wing [Fig.-IV.2]. The applied Reynolds number for the predictions was the same, i.e. at $5.0 \times 10^6$.

In Figure IV.7a, it can be seen that lift curve slope for the Irwin’s undamaged wing ($\delta C_L/\delta \alpha = 0.1$) is slightly higher than the CFD prediction of $\delta C_L/\delta \alpha = 0.09$. Almost at the same incidences of $12^\circ - 12.5^\circ$, the $C_{\text{max}}$ of CFD is higher than the experiment; but at the stall incidences, the CFD suddenly falls much below the experimental value where the lift declines more gradually. The experimental $\alpha_o$ was almost at the same value as in the published data [$\alpha_o = 2.8^\circ$], whilst in CFD, the $\alpha_o$ is shifted to $-3.25^\circ$.

For the damaged wing comparison shown in Figure IV.7b, the slope of the experimental result of 0.086 unlike the undamaged wing is slightly lower than CFD prediction of 0.089. However, the $C_{\text{max}}$ of CFD is unrealistic. At $12^\circ$ the value is higher than the $C_{\text{max}}$ in CFD undamaged; and there is no indication of stall even at the incidence of $14^\circ$. Whilst in the experimental $C_{\text{max}}$, the value is much more realistic where at the same incidence of $12.5^\circ$ the value of 1.132 is lower than the undamaged value of 1.175.

In this damage comparison, the experimental $\alpha_o$ was slightly shifted to $\alpha_o = -2.5^\circ$, whilst in CFD, it was also shifted, but to $-3.0^\circ$.

(Note: No experimental data of coefficients for damaged wing is available in Irwin’s thesis -- especially for 20%c; so plots in Fig.-IV.7b were defined by extracting the available increments data of $dC_L$, $dC_d$, and $dC_m$. The coefficients for the damaged wing were then calculated by:

\[
C_l\,\text{damage} = d[C_l]_{\text{exp}} + C_l\,\text{exp-undamaged} \quad \ldots [10]
\]
\[
C_d\,\text{damage} = d[C_d]_{\text{exp}} + C_d\,\text{exp-undamaged} \quad \ldots [11]
\]
\[
C_m\,\text{damage} = d[C_m]_{\text{exp}} + C_m\,\text{exp-undamaged} \quad \ldots [12]
\]

4.6.2.2 $C_d - C_l$ Plots for CFD Prediction vs. Experimental

Figure IV.7a and Figure IV.7b also show drag coefficients for the undamaged and damaged wings plotted at large scale as $C_d$-$C_l$ plots.

In general, it can be seen that the drag prediction compared to experimental for the undamaged wing [Figure IV.7a] is better than that seen for the damaged wing [Figure IV.7b] for the region of $-8^\circ$ to $+4^\circ$. Whilst, drag prediction is better for the damaged wing in the region beyond $+4^\circ$. 

73
Figure IV. 7a Aerodynamic Coefficients Comparison
2D - Undamaged CFD Prediction vs. Irwin’s Experimental Results
Figure IV.7b Aerodynamic Coefficients Comparison
2D - Damage CFD Prediction vs. Irwin's Experimental Results
The code limitation for drag prediction may clearly be identified when the drag difference is counted. The $\Delta C_d$ are very large for both undamaged and damaged wings. Indeed, it was known from the literature that Fluent’s predictions are good for the lift curve but not for drag.

A similar situation also seen in the pitching moment prediction. The moment prediction for the undamaged wing is closer and more realistic compared to the damaged wing for all incidences; whilst moment prediction for the damaged wing appears closer only in the region of $\pm 5^\circ$.

Looking at the pressure plots, this is not surprising since CFD seems to predict high negative Cps behind damage, which therefore would lead to the more negative $C_m$ as seen.

### 4.6.2.3 The increments Comparisons: $d[C_l]_{cfd-exp}, d[C_d]_{cfd-exp}, \text{and} d[C_m]_{cfd-exp}$

The coefficients of the undamaged wing in Figure-IV.7a and damaged wing in Figure-IV.7b for both CFD prediction and experimental results were subtracted from each other to define the increments $d[C_l]$, $d[C_d]$, and $d[C_m]$. The increments data for experimental results were obtained from Ref. [1]; whilst the increments for CFD prediction are defined based on the following formula:

\[
\begin{align*}
    d[C_l]_{cfd} &= C_l^{cfd\text{-damage}} - C_l^{cfd\text{-undamaged}} \quad \text{...[13]} \\
    d[C_d]_{cfd} &= C_d^{cfd\text{-damage}} - C_d^{cfd\text{-undamaged}} \quad \text{...[14]} \\
    d[C_m]_{cfd} &= C_m^{cfd\text{-damage}} - C_m^{cfd\text{-undamaged}} \quad \text{...[15]}
\end{align*}
\]

The results of each coefficient increment for both CFD prediction and experimental results plotted against incidence are shown in Figure-IV.8.

It can be seen that none of those three increments show good agreement between CFD prediction and experimental results. In $d[C_l]$, the trend of decreased lift shown in the experimental results is not in agreement with the CFD prediction for most incidences, except at $-4^\circ$ and $+8^\circ$ where the lift was decreased. At other incidences, the prediction shows the unexpected result that the lift is increased although a damage hole is present.

The opposite to expected trend shown in $d[C_l]$ was also seen in $d[C_d]$. However, relatively small disagreement between prediction and experimental results are seen between $-4^\circ$ and $+4^\circ$. The increments are positives, although the differences in $d[C_d]$ values are relatively large. Beyond $+4^\circ$, the CFD predictions show an opposite trend to the experimental values.
Figure IV.8: 2D Increments $d[C_1], d[C_d], d[C_m]$ Comparison  
CFD Prediction vs. Experimental Results
In $d[C_m]$, the shapes of plots are similar, but the trend is in the opposite direction especially in the incidences above $0^\circ$. The experimental plot seen nearly in constant values up to $10^\circ$, whilst in prediction, the plot is sharply decreased.

4.6.2.4 Pressure Distribution: $-C_p$ vs. $X/C$ Comparisons

**Undamaged Configuration: $+8^\circ$**

A comparison of wing centreline pressure coefficient ($C_p$) profiles, between experimental data and CFD results for the undamaged wing at $8^\circ$ is shown in Figure-IV.9. There are some differences between the two profiles from leading edge to about $30\%$ of the chord. A better agreement is obtained between the experimental data and CFD results at the rear.

There are two possible reasons why experimental data and CFD results are slightly different near the leading edge. First, it is from incidence setting accuracy during experiments where incidence may be larger or lower than $+8^\circ$. Second, the $C_p$ profile of the 2D experimental result is uncorrected. The tunnel walls tend to increase the speed at the upper wing surface, which in turn reduces the lift and therefore lowers the $-C_p$.

The reference pressure used for the experimental $C_p$ data was obtained from independent tunnel dynamic head pressure transducer.

**Damaged Wing Configuration: Centre-line $C_p$ at $\alpha +8^\circ$**

The $C_p$ profile of the damaged wing at the damage hole centreline is shown in Figure-IV.10. It can be explained that upstream of the damage hole, the CFD results show higher value of $-C_p$ compared with the experimental data. This occurred at about $5\%$ chord until the damage hole.

The experimental data for the upper surface downstream of the hole show that $C_p$ is almost constant up to the trailing edge as a result of a relatively small change of flow characteristics. In the CFD simulation, the 'bump' phenomenon is shown after the strong jet exit from the hole. This is believed to be due to the reverse flow underneath the strong jet, where the velocity is negative but the absolute value is relatively large and the static pressure is low which in turn produces suction, i.e. a significant difference from the experiment. This explanation is supported by the velocity contours shown in Figure-IV.11 below. On the lower wing surface, upstream of the damage hole, there is no significant difference between the experimental data and the CFD results. Downstream of the hole, the experimental profile shows a constant $C_p$ and 'parallel' to the upper surface profile while CFD results show a slight drop.
Cp vs. x/c; Incidence +8 deg.
Irwin's 2d Experimental Results vs. CFD Simulation
Undamage Configurations
20% Damage Size at c/4 - Quarter Chord

Figure IV.9 Undamaged Wings Centreline Cp vs. X/C - Experimental and CFD Simulation Comparison
Cp vs. x/c: Incidence +8 deg.

Irwin's 2d Experimental Results vs. CFD Simulation
Undamage and Damage Configurations
[20% Damage Size at c/4 - Quarter Chord]

Figure IV.10 Centreline and Spanwise Cp vs. X/C - Experimental and CFD Simulation Comparison
4.6.2.5 Experimental and CFD Cp Contours Comparison: Upper Wing Surface $\alpha + 8^\circ$

To show three-dimensional damage effects, the infinite wing '2D' experimental spanwise data from Irwin are presented in the form of pressure contours. Data from the matrix of upper surface pressure tappings are used to generate contours lines with the contour intervals of 0.05. The contour is superimposed on top of a digitised flow-visualisation photograph.

The Cp contours from the experimental data and CFD results for incidence of $+8^\circ$ are shown in Figure-IV.12. Pressure contours are seen to follow the general direction in the region upstream of the hole. Forward separation -- determined by its curved line that follows the shape of hole at a distance of approximately one-third of hole diameter upstream of the hole -- is almost similar in Cp values in the region of between -0.7 to -0.9. In addition, the counter-rotating vortex pair is formed almost at the same place as can be seen from both experimental data and CFD results. However, in the downstream region, Cp contours looks very different in shape. In the CFD, the 'light-blue' region [Cp = -0.91] forms an elongated loop around the damage hole, whilst there is no such region in the experimental data.

There is also a loop [Cp=-1.0] downstream of the damage hole in the CFD simulation at around X/C = 0.6 [red dashed-line]. In experiments, the loop is at around X/C=0.8 and for Cp = -0.65.

In conclusion, the CFD results for Cp are in broad agreement with the experimental data.
Figure IV.12: Experimental vs. CFD [Spalart-Allmaras]: Upper Surface Alpha +8 deg Cp Contour
4.6.2.6 Flow Visualisation Comparison
['Weak-jet' and 'Strong-jet' Simulations vs. Experimental]

Some experimental results for the weak and strong jet flow structure and characteristics are required to confirm the simulation. Hence, typical weak and strong jet characteristics taken from '2D' experimental flow visualisation [1] are used as a benchmark in the following analysis.

> 'Weak-jet' Flow Structures

Figures-IV.13a from the CFD simulation of a typical airfoil with a damage hole at +4° incidence shows a 'weak-jet' flow structure. The damage hole wake at the jet boundary measured from the chord-line is approximately 10 to 12% relative to the model chord. The averaged velocity of the jet \( V_j \) at the hole exit is much less than the freestream velocity measured beyond the jet boundary. This then produces a \( V_j/V_\infty \) ratio of [0.24 to 0.31] which is well within the criteria of \( V_j/V_\infty = 0.5 \) proposed by Margason [12]. The \( V_j/V_\infty \) ratio was determined using the ratio of jets and freestream flow colours that are typically shown in Fluent's plots [Figs.-IV.4/Figs.-IV.5]. Whilst \( V_j/V_\infty = 0.5 \) is Margason's criteria for the jet to start bending while still in the 'discharge-tube', bent over completely right above the exit, and without penetration into the freestream flow.

In Irwin's 'smoke-visualisation' side-view of 20% damage and quarter-chord damage location (Figure-IV.13b), it is indicated that at incidence of 0°, the jet flow through the damage is 'weak'. It can be seen that the flow exiting at the rear edge of the hole was immediately bent over because of the lack of jet penetration into the upper-surface flow. Here, the velocity component normal to the chord is significantly less than that of the freestream. This flow structure can also be identified in CFD simulation as shown in Figures-IV.14a [also at 'weak-jet' incidence, same damage size, and same location], where the bent-over flow structure becoming the damage wake is seen to remain attached to the upper surface, and the wake height (z-axis) is small.

The re-attachment of the damage hole jet to the upper surface [approximately at 0.62C], the relatively small wake width, the formation of CRVP near to the jet exit, and the 'forward' and 'secondary' separation lines are confirmed by the stream-traces shown in Figure-IV.13c.

Experience from experimental results [1] [11] indicated that the position of this CRVP varied with the jet strength and momentum, i.e. the higher the incidence the more forward its position around the damage hole -- and this was verified by CFD simulation.

In Fig.-IV.13c, the main feature is the laminar separation bubble (LSB) -- a region of locally separated flow on the wing [11] [39] [40], which have been described previously. However, this LSB is not related to the damage jet.
Figure IV.13a  Weak-jet Characteristics, Airfoil View at +4° incidence
[CFD Simulation, Turbulence Model: Spalart-Allmaras]

Figure IV.13b  Irwin’s Weak-jet Through Flow Smoke Visualisation
[0° 20%sc, c/4] [Ref.1]
Figure IV.13c ‘Weak-jet’ Characteristics - Planform view, at +4° incidence
[CFD Simulation, Turbulence Model: Spalart-Allmaras]

Note: Figures created by ‘Tecplot v8.0’ [41] based on imported solution data from ‘Fluent’.

Figure IV.13d Irwin’s ‘Weak-jet’ Oil-Flow Visualisation
[0° 20%w, c/4]. [Ref.1]
In the upper surface 'oil-flow visualisation' (another flow visualisation technique used by Irwin), shown in Figure-IV.13d, the indication of the surface flow pattern (taken at +4°), shows clearly the typical surface flow features i.e. 'forward' and 'secondary' separation lines, where the surface flow meets the jet which creates an adverse pressure gradient causing separation ahead of the damage hole. Flow through the hole has also been pushed forward to create a region of reverse flow situated just ahead of the damage hole.

'Velocity vectors' of the CFD simulation for the upper surface at the same incidence also show similar features. It can be seen that at the rear of the damage, two contra-rotating vortices have been formed. In this region the flow is then entrained rearwards, producing a wake that remains attached to the upper surface of the wing.

> ‘Strong-jet’ Flow Structures

In ‘strong-jet’s’ characteristics, as seen both in CFD [Fig.-IV.14a.] and ‘smoke visualisation’ [Fig.-IV.14b.], the jets were no longer immediately bent-over on exiting the hole, but instead, penetrated further into the flow above the upper surface. This means that the flow is detached, which in turn creates a separated region where the flow is three-dimensional with significant reverse flow within the wake.

It can be seen from the CFD-Experimental similarities that the size of the strong jet wake is increased in height relative to model chord-line. In CFD, the height is approx. 35% to 40% [Fig.-IV.14a] compared to only 10% to 12% for the weak jet [Fig.-IV.14a].

Figure-IV.14a also shows the large region of reverse flow beneath the detached strong jet and the wake is significantly increased in width relative to damage hole size [Fig.-IV.14c and Fig.-IV.14d]. Both also indicate the formation of two symmetric large vortices downstream of the damage hole. The formation of the forward separation line shows little change in position compared to what is seen in Fig.-IV.12c and Fig.-IV.12d; while the secondary separation line is moved to about half of the damage hole.

Surface flow visualisation for both experimental and CFD in Fig.-IV.14c and Fig.-IV.14d also shows similar greater reverse velocity at the trailing edge. It can be seen that as the surface flow is entrained forwards from the trailing edge, the velocity component along the surface is seen to be significantly reduced, which suggests the presence of an adverse pressure gradient.

In summary, it appears that the interpretations of the presented flow structure and characteristics from smoke, surface visualisations and CFD simulation were in relatively good agreement.
Figure-IV.14a ‘Strong - jet’ Characteristics – Airfoil View, at ±8° incidence
[Turbulence Model: Spalart-Allmaras]

Note: Figures created by ‘Tecplot v8.0’ [41] based on imported solution data from ‘Fluent’.

Figure-IV.14b Irwin’s ‘Strong -jet’ Through Flow Smoke Visualisation
[10°-20°ac, c/4] [Ref.1]
Figure-IV.14c 'Strong-jet' Characteristics - Planform view, at +8° incidence
[CFD Simulation, Turbulence Model: Spalart-Allmaras]

Note: Figures created by 'Texplot v8.0' [41] based on imported solution data from 'Fluent'.

Figure-IV.14d Irwin's 'Strong-jet' Oil Flow Visualisation
[1000 20%, c/4] [Ref.1]
4.6.3 Discussion of the Synergy between the CFD and Experiments

After reviewing limitations of simulation methods and 2D experimental results, it can be summarise some interesting synergy outcomes from experiments and simulations in 2D battle damaged wing.

The phenomena in Figure IV.7a and Figure IV.7b for undamaged and damaged wing lift curves show that the agreement of the CFD prediction to the experimental results is more reasonable for the undamaged wing compared to the damaged wing in the aspect of slope, $C_{\text{max}}$, and $C_{L\alpha}$. In addition, the reason why undamaged prediction of lift curve and drag are better than the damaged one is believed to be due to following:

- First, the grid boundary conditions (BC). The grid for the undamaged wing is presented as a two-dimensional BC, whilst for damaged wing it must be in three-dimensional BC because of the presence of the damage hole. So that this was like comparing 2D and 3D models, although both are simulated as infinite wings.

  The undamaged wing grid shown in Figure-IV.1 is a standard technique used in CFD prediction for infinite wing. This is usual. Whilst for the damaged wing, the case is unusual. The shape and boundary are infinite but the flow characteristics are three-dimensional -- mainly in the vicinity of the damage hole. Here are the points where the errors may occur. The above condition affects to the drag and pitching moment predictions much more.

- Secondly, the large differences between experimental and drag prediction were possibly because the Fluent's standard surface roughness of 0.5 is applied as default, whilst in the experiments, the model surface roughness is believed to be much smaller, approx. 0.01mm. Another reason is that a turbulent intensity ($T_i$) of 0.1% is applied in the simulation, which is not considered in the experimental data correction, but is broadly similar to experiments.

Another example of the synergy is the trend and values of coefficient increments of the CFD prediction shown in Figure-IV.8, which are far from a close agreement with the experimental results. Opposite trends and the expected results are seen in $d[C_L]$ and $d[C_d]$ at almost all incidences. However, the prediction of $d[C_m]$ seems more reasonable compared to $d[C_L]$ and $d[C_d]$, where values for all incidences are negative. These poor increments predictions are because the coefficients prediction shown for the undamaged wing [Figure-IV.7a] is not in very good agreement with experiments -- though they were reasonable except for $C_D$. In addition, coefficient prediction for the damaged wing [Figure-IV.7b] is worse than for the undamaged one.
The synergy of weak and strong jets flow visualisation, the most interesting phenomena were that among other things the CFD [Fig.-IV.13a] shows a small region of reverse flow and attachment of the jet a little after the hole. This was not detected in Irwin’s 2D, although other research [43] on the flat plate did show a reverse flow region.

The summary for the general explanation of why discrepancies arise between CFD predictions compared with experiments for above cases which described in detail in section 4.6.2.1 to 4.6.2.6 were as following:

First, the turbulence model limitation. The Spalart-Allmaras [SA] model is still relatively new, and no claim is made regarding its suitability to all types of complex engineering flows. Also, that in its original form, the SA-model is designed to be used with fine mesh as low-Reynolds number model, i.e. throughout the viscous-affected region.

Secondly, in term of computation, the Spalart-Allmaras model is the least expensive turbulence model of the option provided in Fluent since only one turbulence transport equation is solved. This model was chosen due to constraints on CPU time and level of solution accuracies required during author’s CFD prediction. Applying more equations solver such as K-ε, Fluent requires 50-60% more CPU time per iteration compared to SA-model. Furthermore, 15-20% more memory is needed [36] [37].

4.7 ‘Cavity – Jet in Crossflow’ Interaction Phenomena: Further Analysis on Battle Damaged Wing CFD Prediction

The field of cavity aerodynamics, even if it is restricted to aircraft application, is a very wide and complex one. The flow phenomena are described largely using available wind-tunnel test data, supported by the results of CFD calculations. In automotive application for instance, the cavity can indeed mimic structural discontinuities, such as open car roofs. In aerospace applications, cavities are seen in many conditions such as weapons bays with doors and the cavity of the extended undercarriage bay. In the field of computational fluid dynamics, the cavity problem is probably the most widespread benchmark test.

The purpose of this section is to show that there is a cavity phenomenon in the damage hole. Two different things interact in the damage hole i.e. cavity flow and the jet flow. Unfortunately, this cavity-JICF interaction phenomenon can only be identified and assessed in CFD simulation, since the experimental results gathered from smoke or oil flow visualisations did not allow the interaction to be identified.
In the literature on cavity aerodynamics \cite{52}, one of the typical flows is commonly called 'open flow'. Flow over open-flow cavities is a test case for several problems of industrial interest. The literature usually assumes for open flow in circular cavity, when $\frac{\text{height}}{\text{dia}} > 0.04$. Recent work at Loughborough University -- UK \cite{56} also suggests that $\frac{\text{height}}{\text{dia}} > 0.1$ is a better criterion at low speed.

Considering battle damaged wings, the cavity as shown in Figures-IV.15a and IV.15b is the damage hole with 40mm in diameter ['cavity diameter, $D$'] and ~24mm high ['cavity height, $h$']. However, not as usually happens in a cavity where one side of the cavity is open, here instead, the top and the bottom walls are both open.

It shows that the hole is not fully occupied by cavity flows due to the presence of jet; i.e. the physical size of jet increases with incidence, so therefore it reduces the size of 'effective cavity'. At 'weak-jet' incidence [Fig.IV.15a], the cavities -- upper and lower cavity flow at the plane of symmetry occupying almost three-quarters [-75%] of the damage hole, while at 'strong-jet' incidence [Fig.IV.15b], the effective cavity became smaller -- occupying approximately 67% of the damage hole.

The range of these cavity cases for both weak and strong jets can be calculated as follows:

**Weak-jet:** $\frac{\text{height}}{\text{dia}} = \frac{24}{0.75 \times 40} = 0.8$; and

**Strong-jet:** $\frac{\text{height}}{\text{dia}} = \frac{24}{0.67 \times 40} = 0.855$;

So therefore, based on circular criteria, flow is expected to be open. However, battle damage cavity is a complicated form of cavity flow.

> **'Cavity – Jet in Crossflow' Interaction Phenomena:**

> **Interpretation of the Flow Structure in the Cavity**

The question of how this cavity-JICF interaction affects battle damaged wing flow structure at the wing surface, the wake, and the jets-in-crossflow boundary are explained by the flow mechanism and development at the hole walls. The case considered is for an infinite wing with the damage hole located at 25%c.

The interpretation of the flow structure in the cavity will compare two pairs of figures, i.e. Figures-IV.15a and IV.15b with Figures-IV.16a and IV.16b.
Symmetry at the Damage Hole Centreline

Figure-IV.15 [a]: Plane of Symmetry at the Damage Hole Centreline

- 'Weak-jet' at 4°: Three-fourth portion cavity, one-fourth portion on JICF;
  - $D_{hole}, 40\text{mm}; \text{at 25\% Chord}$

Figure-IV.15 [b]: Plane of Symmetry at the Damage Hole Centreline

- 'Strong-jet' at 8°: Almost equal Portion for cavity and JICF;
  - $D_{hole}, 40\text{mm}; \text{at 25\% Chord}$

92
Figures-IV.15a and IV.15b show the plane of symmetry at the of damage hole centreline. There are three main features which may be interpreted, i.e. the cross flow, the cavity and the damage jet. The feature that needs to be described here is the cavity -- why it happened; its interaction with jets in cross flow and how this interaction affect battle damaged wing characteristics.

Close examination of these figures show there are two contra-rotating vortices in the cavity for each jet -- effectively two cavities flows one above the other. The 'upper contra-rotating vortices' is the upper surface cavity and the 'lower contra-rotating vortices' for the lower surface cavity.

Figures-IV.16a and IV.16b show detail of stream-traces in the lateral side and at the upstream walls of the cavity. In Figure-IV.16a, vortices are indicated in these cavity sidewalls. The formation of these vortices is believed due to the pressure and velocity gradients that develop in the region near the jet entry, jet exit and within the cavity. The pressure and velocity gradients change due to wing incidence and Reynolds number. The upper-cavity-sidewall-vortices seem to flow to the upper wing surface and combined with strong reverse flow downstream of the cavity, contribute to the initiation of CRVP. Here, the upper wing surface crossflow spills out sideways upon leaving the damage hole that enhances the CRVP structure.

Meanwhile, the lower-cavity-sidewall-vortices are seen rolling-up flowing to the upper wing surface and then combined with upper-cavity-sidewall-vortices with the characteristics as described above. At the upstream wall of the cavity [Figure-IV.16b], the jet layers flows split and go to both the lower and upper wing surfaces.

The sketch shown in Figure-IV.17, can then explain the interpretation of the flow structure in the cavity. The sketch is defined by combining the information that gathered from Figures-IV.15a/15b and Figures-IV.16a/16b.

The impingement line shown in Figure-IV.16b [on the front surface of the hole at 0°] suggests that there are two pairs of vortices [one pair on the right-hand side of the hole, and the other on the left]. It is believed that the plane-of-symmetry vortex and the upper wall vortex are connected with flow gradually feeding into the horseshow vortex.

For the interaction with the jet in battle damage wing case, this can be concluded therefore that features seen in Figure-IV.15a/15b and Figs.-IV.16a/16b are completely different and more complicated form of open-flow cavity characteristics described on literature.
Figure IV.16a. Stream-traces detail in Lateral/Side wall of Cavity [Case: Infinite Wing, Hole located at 25% C, Incidence +8 deg.]

Figure IV.16b. Stream-traces Detail in Upstream wall of Damage Hole [Case: Infinite Wing, Hole located at 25% C, Incidence +8°]

Figure IV.17. Interpretation of the flow structure in the cavity
Chapter 5:

**CFD Simulation of Battle Damaged Finite Aspect Ratio Wing**

Given the results of the 2D CFD predictions, a better and further case needs to be made for the 3D case. This chapter describes the CFD prediction for a finite aspect ratio damaged wing. It is limited to AR8 only, since this was the chosen aspect ratio for the experimental flow visualisation and pressure measurements. Emphasize and concentration was made to the CFD flow structures.

### 5.1 The Grid and the Solver

Fine grids for each undamaged wing and damaged wing configuration were created. For the spanwise location of the damaged wing, the hole size was 20% chord -- similar to the '2D' case, but located at 50% chord. The typical grid for the damaged wing of AR8 is shown in Figure-V.1. The grid difference between 2D and 3D both for the undamaged wing was that in 2D undamaged, the grid can be configured in a two-dimensional boundary condition [BC] - see Fig.IV.1; whilst in 3D undamaged, although no damage hole, the BC must be three-dimensional since the wing is finite.

In this finite wing grid, the boundary condition [BC] was not represented as AAE's tunnel condition but as freestream with a very large distance from the wing tip to the 'roof' and from upper/lower wing to the 'tunnel walls'. The floor was represented as an airplane body with 'symmetry' BC [Fluent's terminology] of right and left hand wings.

In the ‘pressure-far-field’ BC [Fluent’s terminology], the turbulent intensity [77] was set to 0.1% [based on AAE’s new wind tunnel Ti data], and a turbulent length scale ['TLS'] of 1.25mm. This TLS 1.25mm value was required as a parameter in Fluent’s solution and was selected according to the squared-shaped ‘mesh’ of the AAE’s new wind tunnel screen. This approach is normally used to simulate and approximate the turbulent length.

The same turbulence model, Spalart-Allmaras [SA] used in 2D was also used for the damaged finite aspect ratio wing CFD prediction.
Typical Grid for Finite Wing AR8: [4.0x4.0x8.0] M computational domain, 395,000 cells [Basic]

Figure-V.1- Close-up: Denser mesh at the damage hole area and in the region close to the floor
5.2 Aerodynamic Coefficients Prediction:

All the cases with different incidences ranging from \(-4^\circ\) up to \(+12^\circ\) were computed using grids for undamaged [shown in Table-V.1] and damaged [Fig.-V.1] AR8 wing configurations. The use of pre-defined parameters is also shown in Table-V.1. Using Spalart-Allmaras [SA] model, all the solutions were converged within 262 to 344 iterations.

<table>
<thead>
<tr>
<th>Reference</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wing Area AR8 [m²]</td>
<td>0.1624</td>
</tr>
<tr>
<td>Density [freestream] [kg/m³]</td>
<td>1.176675</td>
</tr>
<tr>
<td>Model chord [Cₘₚ][mm]</td>
<td>200</td>
</tr>
<tr>
<td>Pressure [Pascal]</td>
<td>101325</td>
</tr>
<tr>
<td>Temperature [°K]</td>
<td>288.0</td>
</tr>
<tr>
<td>Velocity [U₁][m/s]</td>
<td>40.81757</td>
</tr>
<tr>
<td>Viscosity [kg/m-s]</td>
<td>1.7894e-05</td>
</tr>
</tbody>
</table>

Table-V.1: Reference Values and Undamaged Grid for Computing Forces and Moments

Predictions for all aerodynamic coefficients are shown in Figure-V.2. The figure illustrates the magnitudes and characteristics of the 3D prediction by plotting the undamaged wing coefficients against the damaged wing coefficients at midspan location. The Reynolds number for both predictions is the same, i.e. at 5.5x10⁵.

It can be seen that lift curve slopes of the damaged wing \(\frac{\delta C_L}{\delta \alpha} = 0.0708\) is slightly below the undamaged wing slope of \(\frac{\delta C_L}{\delta \alpha} = 0.08288\). The lift-curve plot is incomplete since stall is not shown for the wing. At the incidence of \(12^\circ\) for instance, the \(C_L\) of the undamaged wing of 1.1354 is, as expected, higher than the damaged value of 1.10215. The incidence of \(14^\circ\) was done intentionally to show the limitation of the prediction. The solution found during the predictions showed that the \(C_L\) was still increasing even after it reached the known stall incidence.

The zero-lift-incidence, \(\alpha_o\) of the undamaged wing is \(-2.95^\circ\), which is just slightly higher than the published 2D data \(\alpha_o = -2.8^\circ\), whilst for the damaged wing, the \(\alpha_o\) is shifted to \(-3.75^\circ\). This shift is believed to be due to the presence of the damage hole. This shift may also be why the slope of the damaged wing is lower than that of the undamaged one.

The prediction of drag coefficients also seems unreasonable. The drag of the damaged wing at high positive incidences [above \(5^\circ\)] is lower than the undamaged wing. This situation is also seen at the lowest negative incidence [-\(4^\circ\)]. However, in the region from \(0^\circ\) to \(5^\circ\) the drag of the damaged wing is higher than for the undamaged wing. There is no particular change seen in pitching moment coefficient.
3D Aerodynamic Coefficients - CFD Simulation
Undamaged vs. Damaged
AR8 - Midspan

Figure V.2:
Aerodynamic Coefficients Prediction [3D_{Undamaged} vs. 3D_{Damage}]
d[CL], d[CD] and d[CM] vs. Incidence - CFD Prediction
Configuration AR8
Midspan Damage Location [450mm from the Root]
5.2.1 The increments Predictions: \( d[C_L]_{\text{efd}} \), \( d[C_D]_{\text{efd}} \), and \( d[C_M]_{\text{efd}} \)

The increments of the coefficients \( d[C_L]_{\text{efd}} \), \( d[C_D]_{\text{efd}} \), and \( d[C_M]_{\text{efd}} \) plotted against incidence for both undamaged and damaged wings are shown in Figure-V.3. These increments were defined by subtracting the coefficients of damaged wing to undamaged wing as shown in earlier equations.

The unrealistic features shown in Figure-V.3 -- where the 'prediction of stall' by the code show lift is increasing further and not seen as declining curve, may be explained and verified by the plots shown in Figure-V.3.

In \( d[C_L]_{\text{efd}} \), it can be seen that for increased incidence of lift-curve [Figure-V.2] the differences are getting larger. Both undamaged and damaged wings are seen beginning to stall just after reaching \( 10^\circ \) -- seen in Figure-V.3 as a sudden decrease in lift-loss \( d[C_L]_{\text{efd}} \). This may happen regardless of whether the wings are close to stall since there is no relation between stall and increments.

The trend of the \( d[C_L]_{\text{efd}} \) itself looks reasonable, starting from -1\(^0\) the wing begins to lose its lift then increased up to -0.02. This magnitude is constant from +1\(^0\) up to 6\(^0\). Beyond this incidence, the lift-loss continued to decrease. The drag-rise \( d[C_D]_{\text{efd}} \) seen in the range from +1\(^0\) up to 6\(^0\). Outside of these incidences, there is no drag-rise whether at negative or positive incidences. Beyond +6\(^0\), the drag-rise is expected to increase significantly, but the prediction seen is in the opposite direction. It can also be seen that both drag-rise and lift-loss are approaching close to each other and reached almost the same value at +10\(^0\); and after this incidence showed similar trends.

In general, it may be concluded that the trends in the magnitude of the aerodynamic coefficients shown in 3D predictions on the graphs of both undamaged and damaged wings are agree reasonably -- particularly at low incidences of lift-curve, but predicted to be poor beyond stall incidences. A Similar situation is also seen in the drag prediction. Illogical solutions are indicated at the incidence of +6\(^0\) and beyond where the drag of the undamaged wing is higher than the drag of the damaged wing. In the increments, the trend of lift-loss seems reasonable but not fully supported by the trend of drag-rise. The change of pitching-moment coefficient is seen to be insignificant by an indication of constant values in the region from -1\(^0\) up to 10\(^0\).

The ability of CFD to predict the increments is very important for survivability analyses, however, since these CFD predictions are not promising, the prediction was then continued to look at pressures and flow visualisation. These are described in the next sub-chapters; while the assessment of the similarity of the above increments trends to those seen in 2D increments is described in Section 5.5.1.
5.3 Prediction of Pressure Distributions [Cp vs. X/C]:
Root, Mid, and Tip Damage Locations

Following the pressure tapping panel availability and pressure measurement test programme
described in Chapter-3, and the pressure distribution plots presented in Chapter-4 [Infinite wing
‘2D’ prediction], the plots here are presented based on the damage hole locations.

For the undamaged configuration, these are nine plots for three incidences. However, not all
profiles are plotted and presented in this chapter. The idea is to compare the 3D predictions with
the available experimental data [See Chapter-7]. The selected Cp vs. X/C cases for damage
located at the root, mid and at the tip are as follows:

1. Undamaged wing vs. Damage hole centreline profiles [Case: Alpha +10°]
3. Incidence effects on Damaged wing at Alpha +4°, +8°, and +10° [Case: centreline]
4. Damaged wing on Inboard and Outboard profiles [Case: Alpha +8°]

5.3.1 Cp vs. X/C: [Case-1]
Undamaged Wing vs. Damaged Wing Centreline Profiles

The predicted Cp profiles comparison between the undamaged wing [at root, mid and tip] and
the damaged wing centreline [also at root, mid and tip] at 10° are shown in Figures V.5a to V.5c.
This 10° incidence is chosen to show the extreme difference of the pressure profiles between the
undamaged and damaged configurations. It can be seen that for both the undamaged wing and
the damaged wing centreline, the peaks of suction are gradually decreased from the root to the
tip. For the undamaged wing, the change of suction from root to mid location [Δ-Cp = 0.15] is
much less than the change from mid to tip location [Δ-Cp = 0.35]. The magnitudes of changes
are the same for the damaged wing. This is due to the effect of typical spanwise load distribution
of a finite wing, where the load at the root is only changed slightly up to the middle then
gradually decreased up to zero at the wing tip.

The overall general trends for damaged configurations is that for the upper surface immediately
forward of damage hole, the Cp is reduced where the flow decelerated and significantly
increased after the damage hole. For every damage location, the load distribution looks very
different. The peak value of the damaged wing is lower in [Figure-V.5a] than that of the
undamaged wing but its magnitude is greater in Fig.-V.5b and Fig.-V.5c. The peak locations are
different too. It is at 5% chord for the damaged wing, while for the undamaged wing it is slightly
upstream, i.e., at 3% chord [in Fig.-V.5b and Fig.-V.5c], and not as in Fig.-V.5a.

At the lower wing surface, there is little change except in the immediate vicinity of the damage
hole and the stagnation point has moved rearwards. These trends are similar to the 2D prediction
shown in Fig.-IV.9 and Fig.-IV.10; only that there the incidences are at 8°.
Figure V.5a [Root], V.5b [Mid], and V.5 c [Tip]:
Alpha +10° Undamaged vs. Damage Centreline
5.3.2 Cp vs. X/C: [Case-2] - Spanwise Variations

The predicted pressure profiles at different spanwise locations of the damaged wing at 4° for a damage hole located at root, mid and tip are shown in Figures V.6a to V.6c. There is no particular reason why this incidence is chosen to show the profiles in spanwise variations except that the incidence represents the weak-jet flow structure; whilst in case-1 the incidence was for a strong jet.

The spanwise locations are the ‘centreline [CL]’, ‘1R’, ‘2R’ and ‘3R’, i.e., centreline, 20mm, 40mm, and 60mm from the damage hole centreline. Undamaged wing profiles are not plotted here.

It can be seen that for all damage hole locations, upstream of the damage hole, the –Cp values at different spanwise locations are almost the same [from leading edge up to 30% chord]. After this location the shape of the profiles are different as the hole region is approached.

As shown in Figures-IV.6a to 6c, downstream of the damage hole, the profiles with the damage hole at the root, mid and tip show a similar shape. At 4°, the ‘bump’ of the profiles at ‘centreline-CL’ and ‘1R’ locations decreases going downstream and reaches the same –Cp values as those at ‘2R’ and ‘3R’ at 80% chord. This is the location where the surface velocities are the same. Beyond this point, the profiles at four spanwise locations are similar. As previously described in Chapter-IV, ‘bump’ is a phenomenon of the reverse flow underneath the weak and the strong jets – which is not seen in experiment. This bump is genuine, though seems to be erroneous.

The ‘1R’ location is exactly at the edge of the hole. The characteristics of the profile at this location are similar at different damage hole locations. It does realise however that the Cps right at the edge of the hole are likely to have large errors. This is because the pressure at this point is actually a combination between the pressure at the wing surface and the pressure at the hole sidewall.

In general, there is almost no difference between ‘2R’ and ‘3R’ profiles for all damage locations, except a very small difference in the area between 30% up to 60% chord. This is due to the ‘2R’ location of 40mm from the damage hole centreline being affected slightly more than the location of ‘3R’. The profile shapes at ‘2R’ and ‘3R’ are similar to the shape but not in magnitude of the undamaged midspan profiles shown in Figure-IV.5.

At the lower wing surface, both upstream and downstream of the damage hole, and at all damage hole locations, the four profiles show the same shape including at the leading edge area.
Figure V.6a [Root], V.6b [Mid], and V.6c [Tip]:
Alpha +4° - Spanwise Cp [INBOARD Rows]

Cp vs. x/c +4 deg.
Single Damage - ROOT - Inboard Rows
CFD Prediction

Cp vs. x/c +4 deg.
Single Damage - MID - Inboard Rows
CFD Prediction

Cp vs. x/c +4 deg.
Single Damage - TP - Inboard Rows
CFD Prediction

Damage Centreline - e4 Root
1R - 20mm from Centreline - e4 Root
2R - 60mm from Centreline - e4 Root
3R - 60mm from Centreline - e4 Root

Damage Centreline - e4 MID
1R - 20mm from Centreline - e4 MID
2R - 60mm from Centreline - e4 MID
3R - 60mm from Centreline - e4 MID

Damage Centreline - e4 Tip
1R - 20mm from Centreline - e4 Tip
2R - 60mm from Centreline - e4 Tip
3R - 60mm from Centreline - e4 Tip

Upper Surface at 0°
Lower Surface at 0°
5.3.3 Cp vs. X/C: [Case-3]

Effect of Incidence of 4°, 8°, and 10° at the Damage Hole Centreline

The pressure distribution at the damage hole centreline for root, mid and tip locations at incidences of 4°, 8° and 10° are shown in Figure-V.7a to Fig.-V.7c.

The objective is to see whether changes of incidence have significant effects on the pressure distribution, mainly downstream of the damage hole, since the incidence changes could influence the damage hole wake strength, wake size and the re-attachments.

It can be seen that for all damage hole locations, upstream of the damage hole, the profiles of all the three incidences have a similar shape although the actual values are different. However, the peaks of suction do not occur at the same chordwise location. At the root, all peaks occur at approximately 6% chord, while at the mid location the peaks occur at 5% chord, and at the tip at 4% chord.

Downstream of the damage hole, all three profiles with the damage hole at the root, mid and tip have slightly different shapes. At the root, the profiles are similar with slightly different peak chordwise locations. At 4°, the peak is at 65% chord, at 8° the peak is at 66% chord, and at 10°, the peak is at 68% chord.

When the damage hole is at the mid location, the peak locations are similar to those with the hole at the root. When the hole is at the tip location, the profiles for all incidences are slightly different from those with the hole at the root and mid locations. The suction peaks at 4° and 8° are in the same position at 64% chord, while at 10° the suction peak is at 70% chord.

At the highest incidence of 10°, the 'bump' covers a range from 60% to 84% of the chord. While at the other two incidences of 4° and 8°, the bumps cover a shorter range from 60% to 70% chord. The reason is that at the highest incidence of 10°, the strongest jet exits from the hole, which strengthens the peak suction and hence there is a longer 'bump' range. The –Cp peak value is low because at this highest incidence, the tip-trailing vortex may affect the flow characteristics near the damage hole, resulting in less suction.

At the lower wing surface, no significant differences are seen. At both upstream and downstream sides of the damage hole, and at all damage hole locations, except at the leading edge area where the stagnation point at the incidence 4° occurs earlier than at the incidence at 8° and 10°. Otherwise, the three profiles are very similar.
Figure V.7a [Root], V.7b [Mid], and V.7c [Tip]:
Effect of Incidence at [4°, 8°, and 10°] [Damage Centreline]
5.3.4 Cp vs. X/C: [Case-4]
Profiles Asymmetrically [Inboard vs. Outboard Rows]

8° was the chosen incidence for case-3. This is just to differentiate the other two previous cases and to represent the incidence of the weak-to-strong-jet flow structure. Pressure distribution comparisons between inboard and outboard spanwise locations of ‘1R’ and ‘3R’ for all damage hole locations and at incidence of 8° are shown in Figures-V.8a to V.8c. The idea is to determine whether there is a significant asymmetry due to the location of the damage hole.

It can be seen that upstream of the hole, the profiles at the root for both inboard and outboard locations are quite similar with a slight difference in peak values at the mid location and some noticeable difference in peak values at the tip.

This indicates that loading is uniform at the root, with a slight change at the middle and more change towards the tip. At the tip location, the lowest peak is at the ‘3R’ outboard row, and the highest peak is at the ‘3R’ inboard row. Again, this is due to the typical characteristics of loading distribution as explained previously, i.e. the more inboard, the higher the loading is. However, looking at the root there appears to be more asymmetry in the pressure distributions than for the tip. This is believed to be due to the floor effects since it is particularly notable that there is no asymmetry for the midspan.

The asymmetry upstream of the damage hole is also seen at the downstream side. The peaks of the ‘bump’ at the root are slightly different – although it was expected to be the same; quite similar at the mid and more visible differences at the tip.

At the lower wing surface, both upstream and downstream of the damage hole, and at different damage hole locations, all profiles are similar in shape and values.

5.4 ‘Weak-jet’ and ‘Strong-Jet’ Visualisation
-A Damage hole at the Wing Midspan

Three incidences 4°, 8° and 10° were selected with the damage hole located at the wing midspan so that the predictions can be compared with the actual 3D experimental flow visualisation [See Chapter-7].

Figure-V.9a, V.9b and V.9c show the stream-traces of jets at the damage hole centreline for those three incidences. It can be seen that at 4° [Figure-V.9a] the flow structure is recognised as the weak-jet because the wake was reattached at approximately 0.8C -- this re-attachment moves slightly rearwards compared with the 2D CFD [Figure-IV.13a].
Cp vs. x/C +8 deg.  
Single Damage - Root - Inboard vs. Outboard Rows  
CFD Prediction

Cp vs. x/C +8 deg.  
Single Damage - Mid - Inboard vs. Outboard Rows  
CFD Prediction

Cp vs. x/C +8 deg.  
Single Damage - Tip - Inboard vs. Outboard Rows  
CFD Prediction

Figure-V.8a [Root], V.8b [Mid], and V.8c [Tip]:  
Alpha +8° Cp Symmetrically [INBOARD vs. OUTBOARD]
The height of the jet wake from the wing chord line to the jet boundary is approximately 12.5% chord \( y/c = 0.125 \). These were weak-jet indications. There is also a reverse flow region on the upper surface wing downstream of the damage hole.

When the incidence of the wing has increased 8° [Figure-V.9b], the feature of the weak jet, i.e. the flow re-attachment is still present but it was much further away at approximately 0.9c. The height of the jet wake also increases to approximately 17.5%c \( y/c = 0.175 \).

It can be said that, although 8° is recognised as an incidence of strong jet in the infinite wing case, in this finite wing CFD prediction, the flow structure is still showing the features of the weak-jet. At 8°, the flow structure characteristic is in a ‘transition’ state from the weak to the strong jet. At 10° [Figure-V.9c], the jet is fully detached and the reverse flow is much higher and occupies a wider area of the upper wing surface. The wake height is almost doubled compared to the wake height seen at 4°. The height is approximately 22.5% chord \( y/c = 0.225 \) measured from chord line to the wake boundary.

Although the actual speed is not shown in these three figures, the jet-to-freestream velocity ratio, \( r = V_j/V_a \) is within the range of 0.2 up to 0.8 -- calculated from the speed of jet exit read in CFD divided by the freestream velocity of 40 m/s. At 10°, the jet is fully detached -- which indicates the strong jet, the ‘r’ value is above 0.5, i.e. \( r = 0.8 \).

Figures V.10a, V.10b and V.10c show the top view of the stream-traces on the upper surface for incidences of 4°, 8° and 10°.

The first thing to note is that, since the damage hole is located at wing midspan, which is far enough from the tip, there is no interaction seen between the trailing vortex and the damage hole. It seems that the trailing vortex has no effect on the damage wake at 4° and 8°, but this may possibly happen at 10° where the inboard shape wake entrainment is slightly affected. However, they all have same asymmetry, whether this is due to the tip vortex or spanwise pressure differences are not clear. The inboard shape wake entrainment is not affected where they all have a symmetry wake.

The next thing to identify is the wake shape. It can be seen that at 4°, Figure-V.10a shows the typical wake of a weak-jet flow structure, i.e., re-attachment and the formation of a counter-rotating-vortex-pair [CRVP]. However, there is no evidence of flow separation forward of the hole and there is no evidence of upstream reverse flow.

When the incidence of the wing is increased to 8° [Figure-V.10b], the flow structure shows some weak-jet characteristics, however, the wake size has increased and slightly curved the re-attachment point, which moves rearward as the CRVP moves forward. Therefore, it is not a complete a weak-jet, but a transitional stage.
Fig. V.9a Mid Damage Weak-jet at +4 deg.

Fig. V.9b Mid Damage Weak to Strong-jet 'Transition' at +8 deg.

Fig. V.9c Mid Damage Strong-jet at +10 deg.

Fig. V.10a Mid Damage Weak-jet at +4 deg.

Fig. V.10b Mid Damage Weak to Strong-jet 'Transition' at +8 deg.

Fig. V.10c Mid Damage Strong-jet at +10 deg. – AR8
At 10° [Figure-V.10c], the flow structure shows strong-jet characteristics. The wake is much wider and is strongly curved, and the jet is fully detached with strong reverse flows. At this incidence, the extent of the separation lines are entrained from both sides of the damage hole and end with a two large rotating vortex centres. The centres are located at approximately one-third of the distance between the damage hole and the trailing edge.

5.5 2D vs. 3D CFD Predictions Comparison

This is a considerable comparison. However, it is impossible to make meaningful analysis. All of the 2D CFD predictions were simulating a 25% chord damage location -- to make it comparable to Irwin's --, and all the 3D CFD prediction was done for 50% chord. No 2D CFD work has been done for 50% chord.

Although damage locations at 25% and 50% chord can be categorised as wing 'mid chord' damage where the increments are not significantly different [1], the jet flow structures, however, are affected by these chordwise distance differences. At the same incidence, the position of the damage hole at 25% in ‘2D’ tend to have ‘stronger’ effects on the momentum and velocity of both weak and strong jets compared to the position of the damage hole at 50% in 3D. This in turn therefore makes the flow structures completely different.

5.6. Empirical Model Application on Battle Damaged Wing

The application of the most appropriate empirical methods to the battle damaged wing condition is supposed to be determined by the availability of the experimental data taken from 2D [1] and from the current 3D experimental results. However, there were no measurements made to define the jet and freestream velocities during the experimental investigation. So therefore, vectors of velocity magnitude shown in 2D and 3D CFD predictions were used to define the jet velocity ratio, \( r \).

The wing incidence, damage hole size and damage hole location relative to wing chord; for both weak and strong jets characteristics; and for 2D and 3D CFD predictions were formulated to correlate to the chosen empirical equations. In order to apply the empirical models to the generic battle damaged wing case; damage hole size of 20% chord and 'mid-chord' [25% and 50% chord] locations were selected.
5.6.1 Jet Trajectory \([x/D]\) of Battle Damaged Wing  
[Case: Infinite Wing with Damage Hole at 25\% Chord]

> Momentum Flux Ratio and Jet Oblique Angle relative to Wing Incidence

As previously mentioned, the most relevant correlations for predicting the jet trajectory of the battle damaged wing case is based on a circular jet discharged at oblique angles into the cross flow. Therefore, equations [4], [5] and [6] shown in Chapter-2 are applicable. However, the range of jet-to-cross-flow momentum flux ratio, \(J\); the angle between the jet’s axis and the direction of the deflecting flow, \(\theta\) are different among these three equations. The equation selection can be made based on the values shown in Table-V.2, i.e. \(r\) and \(a_{\text{wing}}\). This Table-V.2 was determined by the assessment of Fig.-V.11a and Fig.-V.11b of 2D CFD prediction. These values then convert into \(J\) and \(\theta\), where \(\theta = 90^\circ - a_{\text{wing}}\). The conversion result also shown in Table-V.2.

The table shows that none of the \(J\) values is within the range of \(J\) that applied for Equation [4] and Equation [5]; whilst for Equation [6], effective velocity ratio \(V_e\) parameter is required instead of \(J\). In Equation [4], the recommended \(J\) range is from 2 to 22; whilst in Equation [5], from 12 to 1000. The oblique angle \(\theta\) range is from 45\(^\circ\) to 90\(^\circ\) in Equation [4] and from 60\(^\circ\) to 120\(^\circ\) in Equation [5].

From the assessment of those three equations, it was considered that the most appropriate empirical model to be used for generic battle damaged wing is Equation [6] by Margason. This was because the form of the equation is convenient with single valued, which applying most of the empirical correlations.

The only parameter that must be further defined to implement Equation [6] is the \(V_e\) parameter -- which is known as a function of \(V_{\text{jet}}\) and \(V_{\text{freesream}}\) or \(V_e = \frac{1}{r}\). Since \(z\) is defined as the points of the vertical axis, then the centreline of jet trajectory, \(x/D\) can be solved with the known \(D\); coefficients \(F\) and \(G\); and also exponents \(n\) and \(m\).

The results and the calculation method for this chosen equation are presented in Appendix-E:  
'Sample of Empirical Calculation for BDW Application'.
Figure-V.11[a]: Plane of Symmetry at the Damage Hole Centreline
['Weak-jet' at 4°: Infinite Wing with Damage Hole at 25% Chord]

Figure-V.11[b]: Plane of Symmetry at the Damage Hole Centreline
['Strong-jet' at 8°: Infinite Wing with Damage Hole at 25% Chord]

<table>
<thead>
<tr>
<th>Jet Conditions</th>
<th>r</th>
<th>( \theta_{\text{wing}} ) [deg.]</th>
<th>J</th>
<th>( \theta )</th>
<th>( V_e )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weak-jet</td>
<td>0.5225</td>
<td>+4°</td>
<td>0.273</td>
<td>86°</td>
<td>1.9138</td>
</tr>
<tr>
<td>Strong-jet</td>
<td>1.0225</td>
<td>+8°</td>
<td>1.045</td>
<td>82°</td>
<td>0.9779</td>
</tr>
</tbody>
</table>

Table - V.2: Momentum Flux Ratio [J] and Jet Oblique Angle [\( \theta \)] relative to Wing Incidence – 2D
To draw the jet-trajectory, Figure-IV.13.a and Figure-IV.14.a for the 2D CFD predictions taken from Chapter-4 is used as guidance for the axes of the coordinate system. The plots which show the results from Equation-4, 5 and 6 are superimposed on Figs.-IV.13.a and -IV.14.a and shown in Figs.-V.12a and – V.12b.

The position of zero coordinate [0, 0] is located exactly at the centre/axis of the damage hole and at the surface of jet exit. The distance from the centreline of the hole to the trailing edge is 150mm, so that the horizontal axis of the plot is limited to 150 mm, which located inline with the value of wing trailing edge.

Generally, it can be seen that none of those three empirical trajectories fully agree with the jet shape from CFD prediction. However, for the strong jet, all are within the jet boundary; whilst for weak-jet, the Shandorov’s trajectory is within jet for 100mm -- although completely wrong shape.

The shape of Shandorov’s trajectory for both weak and strong-jets is much less in agreement compared to Margason and Ivanov as the trajectory is seen as a straight line and there was no jet curvature. Whilst in Margason and Ivanov, the trajectories are more reasonable with slightly curved as deflected jets.

The plots of Margason and Ivanov are seen as one similar curve, but actually derived from different formula; i.e. Equation [5] -- original equation defined by Ivanov; and Eq-[6] is development of some investigator’s equation by Margason.

It is indicated therefore that applying Ivanov’s exponents [i.e. \( F=1, \ G=1, \ n=2.6 \) and \( m=3 \)] shown in Table-II.1 into Eq.-6 support Margason’s recommendation.

In conclusion, none of the predictions is particularly good. For the weak jet, all of the predicted trajectories get further away from the surface. In the wing case, the flow-field [freestream] is following the shape of the aft wing and therefore forcing the jet down. This is not the case for the flat plate.

It is understood that transition from weak to a strong jet occurs at a lower value of \( r \) than what is shown for JICF. Due to that, similar situation for the strong jet where all of the predicted trajectories also get further away from the surface -- proving that the predicted shapes are essentially for a weak jet.
Figs. V.12a: Weak Jet Trajectory of Empirical Equations – Damage at 25% Chord

Figs. V.12b: Strong Jet Trajectory of Empirical Equations – Damage at 25% Chord

Margason Equation - Ivanov's Exponents

\[
\frac{x}{D} = FV_{c}^{n} \left( \frac{z}{D} \right)^{m} + \frac{z}{D} G \cot(\theta)
\]
5.6.2 Jet Trajectory \([x/D]\) of Battle Damaged Wing

[Case: Finite Wing with Damage Hole at 50\% Chord]

The application of Margason’s formula is further applied to the current investigation; i.e. Finite Wing with Damage Hole at 50\%C. The chosen configuration is for mid-damage with the same incidence as 2D.

Vectors of velocity magnitude from the 3D CFD prediction [Fig.-V.13a and Fig.-V.13b] are used to define the jet velocity ratio, \(r\). The results are shown in Table - V.3. It can be seen that \(r\) - values for both weak and strong jets are slightly smaller than \(r\) shown in Table-V.2. Although the application of freestream velocities in weak and strong jets incidences are similar, the position of damage holes and the pressure differentials across wind are different -- which meant that the exit jets velocities were also different. This situation was recognised and had been previously described in Section 5.5.

Using the same plotting principle in 2D, the jets trajectory for 3D situation was drawn and the results shown in Fig.-V.14a and Fig.-V.14b. The plots were determined by the calculation results that are also shown in Appendix-E.

It can be seen that the horizontal axis and distance from the centreline of the hole to the trailing edge is limited to 100mm; 50mm shorter than in 2D trajectory. The empirical jet trajectory for this finite wing is similar in shape to the 2D predictions. They are not in agreement to the jet shape from CFD prediction -- the predicted trajectories are within CFD jet, but the trajectories are completely different.

The conclusion may be drawn that using the preference empirical methods formulated by previous investigators of JICF; none of them are fully applicable to the battle-damaged wing case. The assessments for both 2D and 3D, for 25\%C and 50\%C, and for both weak and strong jets did not shown satisfactory results even compared to CFD prediction.
Figure-V.13a: Plane of Symmetry at the Damage Hole Centreline

[‘Weak-jet’ at 4°: Finite Wing with Damage Hole at 50% Chord]

\[ r = \frac{V_j}{V_e} \quad J = \left( \frac{\rho_j}{\rho_e} \right) r^2 \]

\[ \theta = 90^\circ - \alpha_{wing} \]

\[ V_e = 40 \frac{m}{s} \quad V_j = \sim 19.5 \frac{m}{s} \]

Figure-V.13b: Plane of Symmetry at the Damage Hole Centreline

[‘Strong-jet’ at 8°: Finite Wing with Damage Hole at 50% Chord]

\[ \theta = 90^\circ - \alpha_{wing} \quad r = \frac{V_j}{V_e} \quad J = \left( \frac{\rho_j}{\rho_e} \right) r^2 \]

\[ V_e = 40 \frac{m}{s} \quad V_j = \sim 37.5 \frac{m}{s} \]

Values Defined based on Figs.-V.13a and V.13b

<table>
<thead>
<tr>
<th>Jet Conditions</th>
<th>( r )</th>
<th>( u_{wing} ) [deg.]</th>
<th>( J )</th>
<th>( \theta )</th>
<th>( V_e )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weak-jet</td>
<td>0.4875</td>
<td>+4°</td>
<td>0.2376</td>
<td>86°</td>
<td>2.0513</td>
</tr>
<tr>
<td>Strong-jet</td>
<td>0.9375</td>
<td>+8°</td>
<td>0.8789</td>
<td>82°</td>
<td>1.0666</td>
</tr>
</tbody>
</table>

Table V.3: Momentum Flux Ratio and Jet Oblique Angle relative to Wing Incidence – 3D
Figs.-V.14a: Weak Jet Trajectory of Empirical Equations – Damage at 50% Chord

Figs.-V.14b: Strong Jet Trajectory of Empirical Equations – Damage at 50% Chord

\[
\frac{x}{D} = F V^n \left( \frac{z}{D} \right)^m + \frac{z}{D} G \cot(\theta)
\]
Chapter 6:
An Experimental Investigation of Battle Damaged Finite Aspect Ratio Wing

6.1 Changes of Aerodynamics Coefficients due to Damage

Results for the damage model may be presented as changes in coefficients $d[C_L]$, $d[C_D]$, and $d[C_M]$, where:

\[
\begin{align*}
    d[C_L] &= C_L \text{damage} - C_L \text{undamaged} \quad \ldots[10] \\
    d[C_D] &= C_D \text{damage} - C_D \text{undamaged} \quad \ldots[11] \\
    d[C_M] &= C_M \text{damage} - C_M \text{undamaged} \quad \ldots[12]
\end{align*}
\]

The increments $d[C_L]$ -- loss of lift due to damage, $d[C_D]$ -- drag due to damage, and $d[C_M]$ -- change of pitching moment due to damage, are presented and plotted against incidence to see how these change with incidence. Increments $d[C_L]$, $d[C_D]$ and $d[C_M]$ are respectively shown in Figure-VI.1a to VI.1c for $d[C_L]$; Figure-VI.2a to VI.2c for $d[C_D]$, and Figure-VI.3a to VI.3c for $d[C_M]$ for AR6, AR8 and AR10 configurations. The idea was to show the increments and trend differences among those three aspect ratios. From the lowest to the highest possible number of damage locations. The plot also shows the prediction of the increments based on Irwin’s 2D increments [1].

An assessment of Irwin’s 2D results by Render in 2005 [42] led to the analysis that the tests in the AAE’s Low Turbulence Tunnel for 2D were under predicting the damage effects at low incidence, and over predicting at high incidences. The investigation was then developed to see whether the same trends appeared when comparing Irwin’s 2D data with finite aspect ratio results.

The conversion of Irwin’s 2D increments to 3D increments was defined by applying the increments of the 2D results at each incidence, multiplying by its respective area and dynamic pressure then dividing by dynamic pressure and the area of AR6, AR8, or AR10. The formulas are written as follows:
\[ d[C_L] = \frac{d[C_L]_{2D} q_{2D} S_{2D}}{q_{2D}(AR6, AR8, AR10) S_{2D}(AR6, AR8, AR10)} \]  
\[ d[C_D] = \frac{d[C_D]_{2D} q_{2D} S_{2D}}{q_{2D}(AR6, AR8, AR10) S_{2D}(AR6, AR8, AR10)} \]  
\[ d[C_M] = \frac{d[C_M]_{2D} q_{3D} S_{3D}}{q_{3D}(AR6, AR8, AR10) S_{3D}(AR6, AR8, AR10)} \]  

Where:

\( d[C_L]_{2D}, d[C_D]_{2D} \) and \( d[C_M]_{2D} \) are 2D increments.

\( q_{2D} \) and \( q_{3D} \) are dynamic pressures \([\frac{1}{2} \rho V^2]\) of 2D and 3D tests.

\( S_{2D} \) and \( S_{3D} \) are the area of the 2D wing and area of AR6, AR8 and AR10 respectively.

### 6.2 The Increments and Trends of AR6, AR8, and AR10

This section presents the increments for the three aspect ratios. The section is intended to highlight the differences and trends and is not intended to explain why the trends occur. The analysis and the discussion are described in the section on flow visualisation and pressure tests.

#### 6.2.1 ‘\( d[C_L] \)’: The Lift – Loss

At first sight, of Figure-VI.1a to Fig.-VI.1c show that the magnitudes of the lift-losses, \( d[C_L]\), were similar for all aspect ratios and for all damage locations. They fall between 0.0 and \(-0.025\) in the incidence below \(+6^\circ\). Beyond this incidence, the highest \( d[C_L]\) is seen for tip damage of AR6.

At the same incidence of \(+4^\circ\), the highest lift-loss appeared for tip damage at AR6, followed by mid and tip damage at AR8; whilst for AR10, the \( d[C_L]\) shows the smallest values for all damage locations. Thus, the highest lift-loss in AR10, i.e. at mid-tip, shows as the smallest \( d[C_L]\) for all aspect ratios.

For the comparison of the 2D results that predicted for 3D, all damage locations in all aspect ratios show similar trends and follow the shape of the 2D increments predictions. All are almost in line with the prediction for the incidence region of \(-5^\circ\) to \(+2^\circ\) for AR6 and from \(-5^\circ\) to \(+4^\circ\) for both AR8 and AR10.

In this particular incidence region of each aspect ratio, it can be seen that the line of the 2D prediction is close to the root and tip damage locations in AR6; almost at the same line with mid location in AR8; and very close to the mid-tip and mid-root locations in AR10.
$d[CL]$ vs. Incidence - Configuration AR6
Tip Damage
vs. Irwin's 2D $d[CL]$ predicted to $d[CL]AR6$

Figure-V1.1a: $d[CL]$ vs. Incidence – AR6
d[CL] vs. Incidence - Configuration AR8
Mid and Tip Damage
vs. Irwins 2D d[CL] predicted to d[CL]AR8

**Figure-VI.1b: d[Cl] vs. Incidence – AR8**
Figure VI.1c: $d[C_L]$ vs. Incidence - AR10

Mid-Root, Mid-Tip, and Tip Damage vs. 'Irwin's 2D' $d[C_L]$ Predicted $d[C_L]$ to AR10

Irwin's 2D Result Predicted to AR10
The plot also shows the significant difference of slope and gradient between the 2D prediction compared to the gradient of all aspect ratios and all damage locations. The sudden change was started at $+2^\circ$ in AR6 and at $+4^\circ$ in AR8 and AR10.

What may be concluded in general is that increasing the incidence results in greater lift-loss for all damage locations and aspect ratios. The method of applying the 2D predictions to the 3D case is justified which shows the closeness and agreement for the three aspect ratios. In addition, the sudden differences to the 2D are believed to be connected with the onset of strong jet flows. It may also be due to Irwin’s over prediction of the effects.

6.2.2 ‘$d[C_D]$’: The Drag Increment

The drag increment plot is shown in Figure-VI.2a to Fig.-VI.2c for all aspect ratios. Eespecially for the mid-tip and mid-root sections, AR10 shows very good agreement with the 3D prediction at all incidences. However, for AR6, the good agreement is limited to the incidence region of below $0^\circ$, whilst for AR8, the agreement is relatively good in the region between $-4^\circ$ up to $+2^\circ$. This suggests that agreement is strongly related to an approximation to the 2D conditions.

The indication of the closeness of the damage location plots was seen in a certain incidence range for all aspect ratios. The closeness seen in tip damage location for AR6 is in the region between $-5^\circ$ up to $+5^\circ$.

In AR8, the tightness between plots is also seen but limited to the shorter incidence region, i.e. between $-1^\circ$ to $+4^\circ$. Beyond this region, the drag increments are different. Mid damage is seen closer to the 3D prediction between $-4^\circ$ up to $+4^\circ$. Also shown is that tip damage produces less $C_D$ increase.

AR10 shows the best fit to the prediction for all damage locations. At incidences of $0^\circ$ up to $+5^\circ$, the three-damage locations shown diverged in the middle sections [mid-tip and mid-root]; the closeness went together with higher drag-increments in line with the 3D prediction. Whilst tip damage went together with a lower drag increment. At $5^\circ$, the two middle sections have the same magnitude of highest drag-increment, with the tip damage, again, producing less drag loss.

As shown in the lift loss [Figure-VI.1a to Fig.-VI.1c], the magnitudes of drag increment, $d[C_D]$ were seen as almost similar for all aspect ratios and for all damage locations, i.e. approximately maximum of $50dC$ in the incidence below $+8^\circ$ for AR6 and AR8, or below $+5^\circ$ for AR10. The similarity is shown in shape, gradient and magnitude in middle sections [mid-tip and mid-root damage] for all incidence ranges; whilst for tip damage, the similarity is shown only at incidences below $0^\circ$. 

124
$d[C_D]$ vs. Incidence - Configuration AR6
Tip Damage
vs. Irwins 2D $d[C_D]$ to $d[C_D]_{AR6}$

Figure-V1.2a: $d[C_D]$ vs. Incidence – AR6
**d[CD] vs. Incidence - Configuration AR8**

*Mid and Tip Damage vs. Irwin's 2D d[CD] predicted to d[CD]AR8*

![Graph showing d[CD] vs. Incidence for AR8 configuration](image)

- **Mid-Damage:**
  - d[CD]: AR8 - Damage Location 450mm from the Root
- **Tip-Damage:**
  - d[CD]: AR8 - Damage Location 650mm from the Root
- **Irwin's 2D Results predicted to AR8:**

**Figure-V1.2b:** d[CD] vs. Incidence – AR8
Figure V1.2c: d[CD] vs. Incidence - AR10
Beyond this, the shape and gradient of the graphs as well as the magnitude of drag increment, \(d[C_D]\) are different. However, there were actually very significant differences for approximately ten drag-counts [10dc] among those damage locations. The highest drag increment of above 50dc appeared for mid damage of AR8.

An assessment of \(d[C_D]\) for similarities at the same locations, e.g. tip damage, is described in the following section [Section 6.3].

Looking in more detail at the \(d[C_D]\), it did not show a clear indication of the transition region from weak-jet to strong jet; and the kink was not identifiable. The change of gradient is relatively small for all three damage locations including the 3D prediction. [See Sect. 6.3].

6.2.3 ‘d[C_M]’: The Changes of Pitching Moment.

The \(d[C_M]\) for all damage and for all aspect ratios is shown in Figure-VI.3a to Fig.-VI.3c. The increments were quite small in magnitude, so the plot has been expanded to ease the analysis.

A similar trend to the 3D predicted \(d[C_M]\) was shown for all damage locations and for all aspect ratios. In AR6, the 3D prediction line trend is almost similar to the tip damage at almost all incidences, whilst in AR8 and AR10, the prediction is always below for all the damage locations at all incidences.

No drastic or sudden change in slope is indicated for any damage location or for all aspect ratios. All damage lines gradually decreased with similar gradient to the predicted line. However, the magnitudes of \(d[C_M]\) were different where AR6 is the highest and AR10 is the lowest.

In AR6, the tip damage including the predictions were found between 0.0 up to -0.01; whilst maximum \(d[C_M]\) for AR8 and AR10 were not more than -0.005 for all damage locations and predictions.

Unlike AR6 and AR8, it can be seen that there was no negative \(d[C_{Me}]\) for all damage locations in AR10. This positive \(d[C_M]\) started from the initial incidence of -4° up to 0°. After 0°, the mid-tip damage drastically changed and dropped to maximum \(d[C_M]\). The mid-root and tip values of \(d[C_M]\) remained positive up to an incidence of +4°. After +4°, the mid-root then dropped to negative \(d[C_M]\) at +6°. Whilst, the tip \(d[C_M]\) remained positive at +6°, it then slightly dropped to a negative \(d[C_M]\) at +8°. These made the mid-tip the highest \(d[C_M]\) at +6°. The lowest \(d[C_M]\) was indicated at the tip location then followed by the mid-root as the second lowest.

In short, it may also be noted that the trend of \(d[C_M]\) at tip damage for all aspect ratios was different compared to the trends \(d[C_L]\) and \(d[C_D]\). Above 0°, and for all aspect ratios, the trend of tip lines was decreased to different magnitudes. The explanation of these differences is described in Section 6.1.6.
Figure-V1.3a: $d_{CM}$ vs. Incidence - AR6
Figure-V13b: \(d[C_M]\) vs. Incidence – AR8
d[CM] vs. Incidence - AR10

Mid-Root, Mid-Tip, and Tip Damage vs. 'Irwins 2D' d[CM] Predicted d[CM] to AR10

Figure-V1.3c: d[CM] vs. Incidence - AR10
6.3 Effect of Tip Damage on AR6, AR8, and AR10

The experiments were intended to investigate whether a constant distance of the damage hole from the tip or from the root could produce different increments by varying the aspect ratio. However, this analysis is limited only to the tip effect because root damage results were ignored. The second intention was to see whether the trailing vortex formation at the wing tip of each aspect ratio affects the transition from weak-jet to strong jet.

The results of a typical drag-increment $d[C_D]$ investigation of this tip effect are shown in Figure-VI.4a. The plot is presented in drag area increments, $\{d[C_D] \times \text{Area}\}$ for the respective aspect ratio against incidence.

For illustration, the plot is also compared with Irwin’s 3D corrected. The correction was defined by $d[C_D]_{2D} \times q_{2D} \times S_{2D}$. The results were in the same line for all three aspect ratios.

It can be seen that from $-4^\circ$ up to $0^\circ$, compared with AR6, AR8, and AR10, the 3D corrected graph shows almost the same magnitudes of drag-increment-area. At positive incidences, the 3D corrected show increased drag area increments compared to the three aspect ratios.

At the incidence of $+8^\circ$, the 3D corrected increased to the maximum value of 0.0016 [Not shown in the plot] then dropped to 0.0001 at $+10^\circ$. It can clearly be seen that, although 2D model has a smaller area, it still gives high drag-increment-areas because the aspect ratio is infinite where there were no tip effects on it.

Although considered very small, there were differences in drag-increment-area between AR6, AR8 and AR10 at incidences below $0^\circ$. Beyond this, $0^\circ$ up to $+4^\circ$, the differences were very small. What is indeed striking about this figure is that the curves look very similar regardless of aspect ratio. However, this is not true for $\{d[C_L] \times \text{Area}\}$ and $\{d[C_M] \times \text{Area}\}$ as shown in Figure-VI.4b and Figure-VI.4c.
Figure-VI.4a ‘drag-increments-area’, $d[C_D] \times \text{Area}$ of Tip Damage Effects for AR6, AR8 and AR10
Effect vs. Incidence
Configuration AR6, AR8, AR10
Tip Effect vs. Irwin's 3D Corrected to AR6, AR8 and AR10

Figure VI.4b and 4c.
d[C_{l}] x Area and d[C_{M}] x Area of Tip Damage Effects for AR6, AR8 and AR10
6.4 Single Hole Damage: Finite Wing – AR8 Flow Visualisation

6.4.1 ‘Weak-jet’ at Incidence +4°

The weak jet characteristics for the finite wing experiments are shown in Figure-VI.5a and VI.5b. The figures represent the tip and mid damage locations respectively. All are for AR8 upper wing surfaces at an incidence of 4°.

Figure-VI.5a illustrates the weak-jet pattern with the influence of ‘tip-effects’. The effect shown by the trailing vortex path (approximately) is shown as a dashed line. It shows that the vortex starts at the tip and prevents the LSB forming at ‘A’. Although laminar separation has occurred at ‘B’, turbulent reattachment is prevented, since the separated flow is entrained into the tip vortex.

Figure-VI.5a also shows the curvature of the horseshoe vortex at ‘C’. It can be observed that the horseshoe vortex forms on only one side of the damage [outboard]; and on the other side, the laminar separation of the horseshoe vortex combines with the LSB. Interestingly, this is true for all damage locations, suggesting that the tip vortex is not the fundamental cause of the asymmetry.

The preliminary assessment of what causes the asymmetry and why the horseshoe vortex appears only on one side and not the other side is as follows:

First, the effect of gravity because the model was installed vertically. There was a moment of 10 to 15 seconds after oil had been painted before the tunnel was brought up to the maximum speed. In that period, there was the possibility that the oil moved due to gravity before flowing in the freestream direction. Moreover, it is difficult to use oil flow on vertical surfaces at air speed less than 100 mph (47.7 m/s), and 150 mph (71.55 m/s) is much better [28].

Second, the effect of surface imperfection on the inboard side of the damage hole. This damaged panel was always used to represent all spanwise damage locations.

Third, the effect was believed to be a genuine feature and not due to gravity, since it appears to have a similar shape for all damage locations and this effect also shown in the CFD prediction]. This is also supported by referring to the previous student experiments [39] where the horizontal model was tested in an open jet tunnel, at a Reynolds number of approximately 400,000 and at the same incidence of +4°. The model had the same NACA profile as in the current experiments. Also it was provided with a damage hole size of 20% chord and was located in close proximity to the wing tip and at 25%c.
AR8 - Tip Effect incidence 4 deg. - Upper Surface

AR8 - Mid Effect incidence 4 deg. - Upper Surface

Figure VI.5a [Tip] and VI.5b [Mid].
Incidence of 4 deg. - Weak-jet Flow visualisation of Finite Wing AR8 at Reynolds No. 5.5x10^6
The photograph in Figure-VI.6a shows the results of the student experiments with weak jet flow at +4°. All the principle weak-jet flow characteristics are present and clear in natural transition condition; including where the freestream flow wrapped around the hole in a horseshoe formation that cut through the wing LSB. The asymmetry in the wake is visible. The dashed line through the centre of the damage hole and through the wake also shows that the line is inclined. This shows strong evidence that it is not gravity that is responsible for the wake asymmetry because this model was installed horizontally.

For the current investigation and at the inboard side of the photograph in Figure-VI.6b, as expected, the LSB still appears after the joint between tip and mid panel. The LSB is interrupted indicated by waviness because the surface at the joint was not smooth enough.

An interesting ‘kink’ of forward separation line appears in the outboard side of damage hole. Kink – is an interaction between the LSB and the horseshoe vortex. Since this kink was also seen at the mid damage, it can be said that this phenomenon is not connected to the spanwise position of the damage hole.

Figure-VI.6b shows the applied forced transition for flow visualisation tests at +4°. The laminar separation bubble on the wing was eliminated by forcing transition upstream of the separation by placing roughness at the wing surface just before the LSB. The roughness with ‘150’ grit size, 5mm width and located at approximately 10mm from leading edge was placed along the span of the mid damage panel. It can be seen that in the span where the roughness is applied, the LSB is eliminated and the damage wake remains asymmetric.

The curvature still appear on the wakes close to the trailing edge where the wake is distorted to the inboard direction but not due to gravity. In addition, the level of distortion looks similar to the student’s experiments that are shown in Figure-VI.6a.

In short, the kink outboard of the damage may be the horseshoe vortex passing through the laminar separation bubble. Hence, the LSB is seen below the horseshoe vortex as a kink.

It also appeared that in the tip damage [Fig.VI.5a], due to the wing tip trailing vortex, the damage wake centreline was slightly distorted to the inboard direction parallel to the edge of the curvature of the horseshoe vortex shown at ‘C’. This distorted wake was also seen in the mid damage [Fig.VI.5b]. Comparing Figs.VI.5a and VI.5b, the wake with damage near the tip is far more distorted. This is most probably due to the influence of the tip vortex.
**Fig. VI. 6a:** Previous Student Project [Tip Damage]. Natural Transition, No Gravity, Weak-jet 4 deg. – Model in Horizontal Position, Damage Hole at 25% chord

**Fig. VI. 6b:** Current Investigation [Mid Damage]. Forced Transition, Gravity Constraint Weak-jet 4 deg. – Model in Vertical Position, Damage Hole at 50% chord

*Figure VI.6* Symmetry Damage Wake [Natural and Forced Transition Application]
The weak-jet flow structure can also be identified in the mid-span position (Figure-VI.5b). Like the tip damage, the LSB also occurred between 50% and 65% chord. Better joints between tip and mid panels at ‘D’ and ‘E’ did not prevent the growth of the LSB outboard.

In both Figs.-VI.5a and VI.5b, at the damage hole rear exit, CRVP were formed. Since the CRVP looks similar i.e. same size, location, and distance, it suggests that there was significant change in jet strength. Downstream of the hole, the damage wake was attached to the surface in both cases.

6.4.2 ‘Transition to Strong jets’ at Incidence +8°

The investigation of strong-jet characteristics is illustrated in Figure-VI.7a and Fig.-VI.7b for the upper wing surfaces of AR8 at an incidence of +8°.

In Figure-VI.7a, for the tip effect, there are significant changes to the damage flow compared with 4°. The LSB is now at the leading edge and has no influence. Separation prior to damage is now located at the edge of the hole. A strong tip effect -- where the tip vortex has stronger with incidence -- tends to ‘twist’ the damage flow in an anti-clockwise direction, which resulted in asymmetric damage flow, particularly on the forward edge of the damage hole. The damage wake increased in size, and the flow patterns indicated completely different flow-fields to those seen in the tip weak-jets. In this Fig.-VI.7a, it is clearly seen that the flow outside the wake looks to be attached right to the trailing edge, but since there are vortices, this cannot be categorised as weak-jet. It was transitional therefore from weak to strong jets.

In Figure-VI.7b of mid damage, the LSB has also moved to the leading edge, which in turn made the forward separation take place immediately prior to the damage. The influence of a strong tip effect that tends to twist the damage flow in an anti-clockwise direction no longer exists. The wakes also ended with two large rotating vortices located at approximately one radius downstream of the damage hole.

Comparing tip and mid wakes, it is seen that the wake for the mid damage is larger and less twisted than at the tip. This is expected since the flow in the mid section is far from the possible influence of root and tip effects, which in turn allow the jet wakes to freely develop and form larger wakes. Since the wake is twisted, so the effects of the finite aspect ratio are seen.

In both figures, as was seen in a weak-jet, CRVP were formed at the damage hole rear exit. CRVP positions are approximately symmetrical about the damage hole centreline. However, these had moved forward around the edge of the damage hole from the previous weak-jet location implying that the jet is stronger.
Figure VI.7a [Tip], and VI.7b [Mid]

Incidence of 8 deg. - Strong-jet Flow visualisation of Finite Wing AR8 at Reynolds No. 5.5x10^6
The damage wake is seen detached from the surface in the mid location and attached right to the trailing edge for tip damage. The wake showed significant reverse flow in both tip and mid locations but there was much larger reverse flow for the mid location.

It can also be seen in both figures that the secondary separation line is behind the front edge of the hole. This suggests that the jet was at the back of the hole.

Figure-VI.7a [Tip] and Figure-VI.7b [Mid] also indicated that the reverse flow was greatest at the inboard side of the trailing edge. At the inboard, the vortex goes beyond the wing’s trailing edge, suggesting it is bigger or stronger than at the outboard. The strong reverse flow then fed into a strong vortex that was displaced to the inboard side of the damage. The vortex centre at the outboard side seems weak compared with the inboard vortex.

For tip location, the reason was clearly seen that the strong outboard entrainment was due to the effect of the trailing vortex, while that for the mid location was believed to be due to genuine reasons -- such as no influence of root and tip effects; and not due to surface imperfection, nor to gravity or root-tip effects. In general, there seems to be a similar degree of asymmetry for both locations.

6.4.3 ‘Strong-jet’ at Incidence $10^\circ$

The characteristics of strong jet for the current experiments were investigated further at $10^\circ$. The main objective of conducting this test was to verify that both of the jets are now strong.

In general, however, both cases clearly show that the strong jet was stronger than at $8^\circ$. The statement is supported by the criteria that there are larger and wider damage wakes and stronger reverse flow indicated by larger vortex centres there are clearer secondary separation lines in $10^\circ$ than $8^\circ$. The tip is now strong rather than transitional. More explanation that is detailed is given in the following:

Figure-VI.8a shows the influence of the tip. It can be seen that the region of trailing vortex formation was stronger and was initiated just at the leading edge of the wing tip, i.e. it covered a larger area compared to that seen in $8^\circ$.

The entrainment flow in this wing tip had resulted in a stronger vortex centre than at $8^\circ$ and the effect of the twisted anti-clockwise damage flow due to the trailing vortex is still present.

Figure-VI.8b illustrates the mid-damage strong-jet flow structure. Here, both forward and secondary separation line locations appeared similar to those seen in $8^\circ$ but the characteristics of CRVP were stronger and further forward. It can also be seen that the effect of strong flow entrained from the separated region before the trailing edge had enhanced the reverse flow and in turn made the vortex centre appear much bigger and stronger than that seen for mid damage at $8^\circ$.  

141
Figure-VL8a [Tip] and VL8b [Mid].
Incidence of 10 deg. – Strong-jet Flow visualisation of Finite Wing AR8
6.5 Finite Wing – AR8 Pressure Measurement

This section analyse the spanwise pressures at ‘centreline’, ‘1R’, ‘2R’ and ‘3R’ of Inboard and Outboard sides of damage hole; and at the incidence of +4°, +8°, and +10°.

Three things need to be noted about the analysis:
1. For the damaged wing, there were no pressure tappings installed at the hole, and therefore no centreline Cp values were measured at this point.
2. There were no pressure tappings installed in the lower wing surface.
3. Profiles always ended at 88% chord for centreline and all inboard rows; whilst for all outboard rows, profiles always ended at 90.5% chord. These were manufacturing limitations where pressure tappings could not be placed close to the trailing edge.

6.5.1 Damage Centreline and Spanwise Pressure Data
6.5.1.1 Tip Damage

a. Tip Damage: Incidence +4°

Figure-VI.9 for tip damage shows the Cp vs. X/C profiles are presented at above and below the flow visualisation photograph. The lines of spanwise rows are marked on the photograph based on the colours on the graphs. Overall, it can be seen that upstream of the hole, the centreline pressure peak position changed relative to the pressure lines of the two groups of rows -- though differences in the peak values were very small. Whilst downstream of the hole -- where the flow emerged, the profiles show significant differences between the centreline and the two groups of rows. At the leading edge, the peak suction showed the expected trend, i.e. decreasing magnitude as the row moves outwards.

- Inboard Rows. At the inboard rows [1R, 2R, and 3R], and at the region before x/c = 0.1, there were always slight reductions in the magnitudes of –Cp peaks. After the peak [at the region between x/c = 0.1 and x/c = 0.4], slight differences in –Cp values for those three rows were seen when approaching the hole. These differences became more significant just prior to the upstream edge of the hole -- a peak at the centreline was the lowest and 3R was the highest. At these inboard rows, the further the distance from the centreline, the higher the peaks. This phenomenon was clearly indicated also in the damage hole region [x/c = 0.4 up to 0.6].

Downstream of the hole, the profile of the centreline suggests that the flow in this region was accelerating at the downstream edge of the hole that made the increase of –Cp. The subsequent recovery of Cp profiles for the centreline and all spanwise inboard rows reflects the flow returning to undisturbed surface flow velocities. This indicates that the wake was still attached and that a weak-jet was present.
**Figure V1.9 AR8 – TIP DAMAGE ‘Weak-jet’ Incidence +4 deg.**

Inboard and Outboard Spanwise Pressure - Flow Visualisation Agreement
Downstream of the hole, the magnitudes of \(-\text{Cp}\) at approximately 62.5\%c was very similar for the 2R and 3R rows. Soon after this point, the centreline profile was increased; reaching the peak at 75\%c then declining with \(-\text{Cp}\) reduced to the same magnitudes as the other three rows at approximately 88\%c.

- **Outboard Rows.** Upstream of the hole, the profiles for all spanwise rows lie below the centreline profile. This was the opposite situation to the inboard rows. The centreline was the highest and the 3R was the lowest. This was also acceptable since the more outboard the rows were, the most affected by the trailing vortex. The peak of the profile at the centreline shows still higher than the other three rows. This indicated that for these profiles the trailing vortex was more significant than the presence of the hole. The photographs also indicated clearly that the effect of the trailing vortex influenced the flow up to a distance of 1R. The unsmooth profile lines at centreline, 1R, and 2R rows just at the forward hole were believed to be due to the flow mechanism that was happening in that position. The fluctuating lines indicate that the pressures were located at the forward/secondary flow separations and at the horseshoe vortex regions that were affected by trailing vortex. These are shown in the photograph. Downstream, the accelerated flow was seen in the centreline and in 1R, which also suggests that the centreline flow affects the wake up to 1R distance. At 90\%c, the centreline profile and the other three rows return to the undisturbed surface indicating that the wake was also still attached.

The asymmetry in the pressure profiles between inboard and outboard rows downstream of the hole were indicated mostly at 1R. This was consistent with the flow patterns shown in the photograph where it suggests that the flow was accelerating at 1R outboard but not at 1R inboard.

**b. Tip Damage: Incidence +8°**

When the incidence increased to 8° [Figure-VI.10], for both inboard and outboard rows, the profiles also show a reduction in \(-\text{Cp}\) between the wing leading edge and the upstream edge of the hole. The centreline profile was the lowest in the inboard rows and the highest in the outboard rows. At the damage hole region, the presence of the hole seems not to affect the 2R and 3R profiles in the inboard rows but did slightly affect the outboard. This can be said because the profiles of 1R and 2R at the inboard were not dropping as seen at the outboard. Downstream of the hole, different Cp characteristics were seen within the wake, especially for centreline and 1R profiles. The \(-\text{Cp}\) peak in the centreline wake had moved progressively rearward from the downstream edge of the hole, by approximately 16\%c [at 83\%c] compared to what was seen for 4° peaks. Peak of 1R at outboard was seen earlier than inboard which indicated that at the 1R inboard, at approximately 75\%chord, the flow was decelerated and had a velocity lower than at the outboard. This also indicates that the flow was asymmetric.

145
Figure VI.10 AR8 – TIP DAMAGE Incidence 8 deg.
Spanwise Pressure – Flow Visualisation Agreement
By the trailing edge, for all spanwise rows – inboard and outboard, the profiles no longer returned to the values seen in 4°. This indicates that the jet has changed from weak (4°) to strong jets (8°); although from the flow visualisation 3R-Inboard and outboard are shown to be clearly attached. However, since the centreline-CL and 1R’s did not meet other rows at the trailing edge, strong-jet is indicated.

c. Tip Damage: Effect of Incidences on ‘1R’-Inboard/Outboard

The objective is to see whether changes of incidence have significant effects on the profile shapes, mainly downstream of the damage hole. Downstream, the incidence would have a significant effect on damage wake strength, damage wake size and the re-attachments. These wake characteristics would certainly affect the pressure distribution profiles.

Figure-VI.11 shows the incidence effect at 4° and 8° in the tip damage. The figures show the combination of 1R inboard/outboard that was taken from the same data at 4° and 8° analysed previously.

In tip damage, it can be seen that at the weak-jet incidence of 4°, as expected, the 1R-inboard -Cp peak at the leading edge was slightly higher than the outboard. However, soon after this peak, both inboard and outboard had very similar values. This continued until the region of the damage hole centreline, except for the small region [30% - 36%c] where the 1R-outboard had a decelerated profile – this decelerated 1R outboard pressure is possibly due to ‘pressure-leaking-since there is a quite abrupt change. This was not seen in 1R-inboard. This pressure leaking was not a faulty measurement but possibly due to tube leakage inside the model.

At downstream, after the jet exit region, 1R-Inboard flow accelerated to be slightly higher than outboard at 62.5%c, but then decelerated to the undisturbed attached flow surface. Whilst in 1R-outboard, after 62.5%c, the flow accelerated to the peak at 75%c. The flow then also returned to the same value of 1R-Inboard at the trailing edge.

At a strong-jet incidence of 8°, 1R-inboard –Cp peak at the leading edge was significantly higher than the outboard, and after the peak, 1R-Inboard continued to remain higher than outboard until just at the upstream edge of the hole [40%c]. It appeared that the trailing vortex affected the pressure in the outboard upstream regions. By contrast in the hole region, 1R-Inboard always had a lower –Cp than outboard, implying that the trailing vortex has accelerated the exited strong jet in 1R-Outboard and decelerated the flow at 1R-Inboard. The flow visualisation photograph shows where the wake at the outboard region is twisted in the inboard direction.

Downstream, whilst the 1R-Inboard flow indicates fluctuating acceleration, probably due to experimental error, and is detached at the trailing edge, the 1R-Outboard continue to decline and returned to the undisturbed attached flow surface.
Spanwise Pressure Measurement - AR8
Effect of Incidences - Alpha 4deg. and Alpha 8deg.

[TIP Inboard and Outboard - TIP]

Figure-VI.11 Effect of Incidences of 4° and 8° at TIP Damage
[1T Inboard vs. Outboard]
6.5.1.2 Mid-Damage

a. Mid-Damage: Incidence +4°

Figure-VI.12 for mid damage shows the Cp vs. X/C profiles at +4°. The plots are presented at above and below the flow visualisation photograph.

The overall trends starting from the leading edge to the hole regions and then up to the trailing edge, show that the shape of the three spanwise profiles at both inboard and outboard rows are different.

The most visible differences are seen in –Cp peaks where inboard spanwise are higher than outboard. On the hole centerline region, the inboard spanwise rows, except 1R, are less affected by the hole; whilst outboard, all spanwise location profiles i.e. 1R, 2R, and 3R are dropped to almost the same value. The pressures that are increased or decreased by the presence of the hole can be indicated by the shape or change in profile line compared to the known undamaged profile.

Downstream of the hole, the 1R-inboard is seen to be slightly accelerated compared to sudden increased acceleration at 1R-outboard. At inboard spanwise, the 2R and 3R are decelerated in relatively small differences along the downstream region and do not return to the same positions at the trailing edge. Whilst at outboard spanwise, the 2R and 3R are also decelerated but in very close and almost at the same values up to the trailing edge. The 1R-outboard is also very close to the point where 2R and 3R meets. These agreed to the photograph of flow asymmetry and wake re-attachment close to the trailing edge.

In the region of 30% - 34% C inboard, there is a significant spike in 3R, small spike in 2R, and gradual deceleration in 1R. Whilst in outboard, the 2R has dropped to the same value as 1R. The 3R-outboard has also dropped in this region. These features however were not explainable in term of flow physics but are believed to be due to either the rough surface or inaccurate pressure reading at the time of measurement.

As seen in the weak-jet incidence of tip damage, the centreline profile suggest that at the downstream hole the flow is always accelerating just at the hole exit causing the increase of –Cp. In addition, the subsequent recovery of Cp profiles reflects the flow returning to undisturbed surface flow velocities, and indicates that the wake was still attached and a weak-jet present.

The overall trends seen in pressure profiles for this mid damage are consistent with those seen for tip damage at the same incidence at +4°.
INBOARD ROWS

Suspect Tapping error

Hole Centrelne

Suspect. This tapping reads low

Hole Edge upstream

Hole Edge downstream

Figure-VI. 12 AR8 – MID DAMAGE ‘Weak-jet’ Incidence +4 deg.
Inboard and Outboard Spanwise Pressure - Flow Visualisation Agreement
b. Mid-Damage: Incidence +8°

The pressure distributions for 8° characteristics are shown in Figure-VI.13. In the centreline and for all inboard spanwise profiles, the \(-\text{Cp}\) peaks at the leading edge are almost all concentrated at the same \(-\text{Cp}\) value. This indicated that the difference of lift distribution were insignificant between spanwise rows at the inboard. This phenomenon was not seen in outboard rows where the \(-\text{Cp}\) peaks at the leading edge are different. This is reasonable for a finite aspect ratio wing where the flow and the loading were asymmetric at either side of hole. The condition was consistent with the flow visualisation of mid damage. Significant wake asymmetry between inboard and outboard wakes seen in the photograph made the profiles dissimilar. Also, the addition of dotted lines to the flow visualisation photograph helps to emphasise the asymmetry in the flow.

There was a difference in the profiles seen in the damage hole region. At downstream, peaks of IR profiles were seen to have a different shape between inboard and outboard, and the magnitudes of \(-\text{Cp}\) at 75%c for all spanwise rows were different. 1R-Inboard flow indicates fluctuating acceleration, which is suspected to be a measurement error, and is detached at the trailing edge. Whilst the 1R-Outboard continued to decline and was seen returning to the undisturbed attached flow surface. This is similar to the 1R-inboard/outboard positions at 4°.

The downstream hole centreline \(-\text{Cp}\) had a different magnitude to the other three spanwise inboard rows; and at outboard, the \(-\text{Cp}\) magnitude for 1R was also slightly higher. The velocity difference between these two spanwise rows was believed to be due to the reverse flow that flows from outboard to inboard, i.e. asymmetry. In addition, it appeared that the finite aspect ratio wing affected the pressure in the outboard upstream regions, i.e. there was a decreasing pressure distribution along the span.

c. Mid-Damage: Effect of Incidences on ‘1R’-Inboard/Outboard

In Figure-VI.14, in the upstream region, at both incidences of 4° and 8°, both profile pairs seem very similar despite the slightly higher leading edge peaks of 1R-Inboard. Approaching the hole up to the hole damage region, 1R-Outboard of 8° is always higher than 1R-Inboard, possibly due to measurement error, where in the hole centreline region, 1R-Inboard was the lowest compared to the other three profiles. This implied that although mid damage was no longer affected by trailing vortex and there was no root influence, the 1R inboard/outboard local pressure at 8° was not similar. This was believed to be due to the vertical position of the model. This indication was supported by the wake asymmetry seen in the photograph. At downstream, as was seen in the tip damage, the 1R-Inboard was detached at the trailing edge, whilst and the other three were almost returned to the undisturbed surface positions.
Figure VI.13 AR8 – MID DAMAGE, Incidence 8 deg.
Spanwise Pressure - Flow Visualisation Agreement
Spanwise Pressure Measurement - ARB
Effect of Incidences - Alpha 4deg. and Alpha 8deg.
[Inboard and Outboard - MID]

Figure-V1.14 Effect of Incidences of 4° and 8° at MID Damage
[1'R Inboard vs. Outboard]
6.6 Mid Damage AR8 vs. Irwin’s-2D

6.6.1 Flow Visualisation Results Comparison

The general comparison is focussed between mid-span damage in the finite wing and the infinite-wing 2D visualisations. This comparison is made because mid-span, due to its position, is considered as ‘quasi-2D’ in terms of flow structure characteristics. However, despite some similarities, the flow structures also show some significant differences.

The similarity of weak-jet flow structure at the mid damage shown in Figure-VI.15b is clearly and fully identified in Irwin’s -2D additional tests ‘weak jet’ visualisation with the damage hole also located at 50% chord [shown in Figure-VI.15a]. Both were taken at 4°. It shows that both LSB occurred almost at the same location i.e. at 50% wing chord -- though in Irwin’s 2D, LSB occurred slightly earlier and is seen to be interrupted at the portside. The two forward/secondary separation lines were similar though the separation lines in 2D were not as sharp as those seen in 3D. In addition, the distance of the separation lines ahead of the damage hole appeared slightly more forward in 2D than in 3D.

At the downstream hole where the CRVP formed, the vortex pairs in 2D wing Fig.-VI.15a appeared slightly stronger than in 3D [Figure-VI.15b] and the vortex pairs’ location was little bit more backward in 3D than in 2D. This is believed to be due to the known condition of finite and infinite wings. Finite wing required more angle of incidence than infinite wing to have the same flow characteristics. These wing type differences also affected the pattern of wakes. The wake for the finite wing shows purely weak jet where the wake was ‘un-divided’ and asymmetric, whilst for the infinite wing the wake was divided and symmetric. In 2D, the wakes already show an indication of strong jet. Both wakes, however, remain attached to the upper surface of the wing.

The mid damage strong-jet phenomenon that is seen in Finite wing AR8 [Figure-VI.16b] can also be identified in the 2D wing visualisation [Figure-VI.16a]. However, it can be observed that the wake due to strong jet in AR8 was not as large as the 2D strong jet. In the 2D strong jet, the wake size was much wider and stronger. In addition, the flow entrainments were similar between right-hand and left-hand sides of the damage hole. Whilst for finite wing strong jet, the wakes were unsymmetrical. The size of the wakes in 3D is seen to be almost similar to the weak size in 2D wake jet [Fig.-VI.15a].

Both photographs in Figure-VI.16, show the pair of counter-rotating-vortices located at the edge of the damaged hole but in both cases the vortices had moved further forward around the damaged edge from what has seen in the weak-jet of both finite and infinite wings.
Figure-VL.15a: Irwin's 2D +4° – Upper Surface
Figure-VL.16a: Irwin's 2D +8° – Upper Surface

Figure-VL.15b: Mid Damage – AR8 +4° – Upper Surface
Figure-VL.16b: Mid Damage – AR8 +8° – Upper Surface

[Taken from Ref. 1 for Top Left Fig.]  
[Taken from Ref. 1 for Top Right Fig.]
From both weak-jet and strong-jet observations, it may be concluded that the flow structure of surface visualisations undertaken at 50\%C mid damage in Finite Wing of AR8 despite the genuine asymmetric indication, also show some similarities to those identified at the 50\%C damage location in 2D. In addition to that, an ‘early strong jet’ indicated in Finite Wing -AR8 at 8° was also indicated in 2D but at the incidence of 4°.

The differences indicated in both weak and strong jet flow structures in both 2D and 3D above also support the big differences seen in Irwin’s 2D increments at higher values of incidence as discussed in the previous section.

### 6.6.2 Pressure Analysis Based on Flow Structure Visualisation

At 8° of mid damage for the finite wing — AR8 shown in Figure-VI.17b was identified as a strong jet. Irwin’s 2D was also identified as a strong jet but Figure-VI.17a shows that the jet is running fuller, stronger, and more extensively. Significant differences in jet flow occurred.

Although Irwin’s 2D damage location [25\%C] is different from the current investigation [50\%C], the issue here is about the different characteristics of both strong-jets. The jet is running fuller, stronger, and more extensively in Irwin’s 2D is believed to be more due to the hole damage location. Quarter chord [25\%c] is the aerodynamic centre region and centre of pressure. Holes in this region will affect the momentum of the exit jet and in turn influence the flow structure and characteristics, including pressure. In a finite wing with a hole at 50\%c, it showed strong jet characteristics but the momentum at the exit is weaker.

The most likely reason for this is that the pressure distributions were different. A typical pressure distribution difference is shown in Figure-VI.17. The graph for the damage hole centreline, is only for 8° since no -Cp data was available for +4° in Irwin’s 2D analysis. In addition, the pressure measurement comparison shown here is for different damage hole locations since there was no pressure measurement for 2D at 50\% wing chord damage.

As already seen in all Cp profiles for the finite wing, the comparison can only be made for the upper surface. In addition, the actual peak for the finite wing could not be identified because the tappings were sparse with no tappings near and at the leading edge.

In conclusion, the most significant and interesting observation is that behind the hole there is far more acceleration for the finite aspect ratio wing than for the 2D. In addition, the pressures near the leading edge are almost identical.

156
Figure VI.17a Irwin's 2D: Alpha +8° Upper Surface [Ref.1]

Pressure Distribution Comparison
Irwin's 2d vs. Mid Damage - AR8 at Alpha 8 deg.
[Pressure at Hole Centreline]

Figure VI.17b Mid Damage Finite Wing – AR8 Alpha +8° Upper Surface
:Pressure Distribution Comparison Based on Alpha +8° Flow Structure Visualisation
Chapter 7: Comparison Analysis of 3D CFD Prediction and 3D Experimental Investigation

7.1 Introduction

This chapter shows the comparison between the 3D CFD predictions of the battle-damaged finite aspect ratio wing that has been described in Chapter-V and the experimental results in Chapter-VI. The chapter is based on midspan damage of AR8 and is intended to show whether Fluent® can predict the effects of battle damage on an aircraft wing in all aspects of measurements.

7.2 The Increments Comparison:

\[ d[C_L]_{3D-cfd}, d[C_D]_{3D-cfd}, \text{ and } d[C_M]_{3D-cfd} \text{ vs. } d[C_L]_{Mid-A8-Exp}, d[C_D]_{Mid-A8-Exp}, \text{ and } d[C_M]_{Mid-A8-Exp} \]

Although limited resources [e.g. CPU and file size] and time were available for the CFD prediction for a finite wing during the study, it is very important to define the aerodynamic coefficients’ increment. The finite wing increment, \( d[C_L] \), \( d[C_D] \), and \( d[C_M] \) comparison between CFD and experiment is one of the essences of the current investigation.

The results of each coefficient increment for both the 3D CFD prediction and experimental results of midspan damage - AR8 plotted against incidence are shown in Figure-VII.1. The general trend is that the \( d[C_L] \) increment shows relatively good agreement in the incidence region of \( 0^0 \) to \( +6^0 \), but not the \( d[C_D] \) and \( d[C_M] \) increments. Incidences prior to and beyond this region are far from agreement.

The greatest agreement among those increment comparisons was indicate in \( d[C_L] \), though Fluent® shows unrealistic prediction for the incidences below \( 0^0 \). However, the good agreement of the prediction covered the weak to strong jets incidences. The overall \( d[C_L] \) of the prediction was in the correct trend; continue declining up to close to the stall incidence.

The agreement for \( d[C_D] \) is not as good as for \( d[C_L] \) but covered almost the same incidence range -- e.g. \( d[C_D]_{3D-cfd} \) is twice the value of \( d[C_D]_{3D-Exp} \). The highest disagreement of approximately 0.006 within this incidence range is seen at \( +2^0 \). This disagreement is far too large.
Figure VII.1: Increments $d[C_1]$, $d[C_2]$, $d[C_3]$ Comparison
3D CFD Prediction vs. Mid Dam – AR8 Experiment
The increased $d[C_D]$ trend of the prediction is seen as being limited up to $+6^\circ$, and then declining whilst in the experiment, the trend tended to increase. In 2D, the $d[C_D]$ is seen to stop increasing in $+8^\circ$. In $d[C_M]$, the shape of plots is almost similar in the incidence region of $0^\circ$ to $+6^\circ$, but the trend is in the opposite direction, especially in the incidences below $0^\circ$. The code showed and predicts the pitching moment increment as negative below $0^\circ$, whilst the experiment showed it to be positive and constant in the same incidence region. As with the prediction, the experimental plot is seen to be almost constant for values up to $8^\circ$.

It may be said that for predicting the increments $d[C_L]$, $d[C_D]$, and $d[C_M]$ of the finite wing, the code can be used in the limited positive incidences up to prior to stall. At close-to-stall-incidences, the flow is beginning to separate at the upstream hole, and there is large turbulence at the downstream hole. Fluent's solution using SA is not suitable and tends to give less accuracy to solve the problems of very large turbulent flows. This limitation of the prediction was not limited to a particular airfoil type. For all increments, the limitation of the predictions was seen almost at the same incidence range.

### 7.3 Pressure Profiles Comparison

The comparison is made only for selected cases. The sequence of the analysis is first, describing the profiles’ comparison of weak and strong jets incidences at the centreline of mid damage. Next is the analysis of the spanwise distance of ‘1R’ at the inboard and outboard sides of the hole. This will assess the asymmetry. The last is the analysis of weak and strong jets profiles at the tip, mid and root damage locations.

#### 7.3.1 Profiles Comparison at Weak and Strong Jets Incidences

**Mid Damage Centreline - AR8**

The profiles comparison of the CFD prediction and the pressure measurement for damage centreline $[CL]$ located at the midspan at $4^\circ$ and $8^\circ$ is shown in Figure-VII.2. The first impression is that similar profile features are seen, but CFD under predicts leading edge peak suction. A good agreement is seen after the leading edge to the upstream edge of the damage, whilst downstream from the damage significant differences appeared. The upstream hole shows that, despite the peak at the first small chord percentage from the leading edge up to $40\%$ chord where there are no experimental data point, $-C_p$ values are almost the same for both CFD and experiments. However, at the $10$ to $12\%c$ at $8^\circ$ incidences, the experimental points were slightly higher than the predictions, but still showed that the shapes are relatively similar.

In the CFD prediction at both incidences, downstream of the damaged hole, the CL ‘bumps’ indicated that the jet was accelerated to its maximum speed and reached the peak values between $65$ and $67\%c$. After this, the flow speed decreased and the profile gradually declined up to the trailing edge.
Pressure Measurement CFD vs. Experiments
Effect of Incidences - Alpha 4 and 8 degrees
Centreline Damage - MID AR8

Figure VII.2 Profiles Comparison at Weak and Strong Jets Incidences - 3D CFD vs. Mid Dam - AR8 EXP.
There were also large differences between the peaks of CFD and experiment where the peak of CFD prediction occurred earlier than the experiment. In addition, the experiment shows bigger differences between weak and strong jets than CFD.

In general, the graphs indicated that in CFD simulation, the emerged strong jet produced more suction or more acceleration than the experiment, but the detached flow acceleration on CFD was much lower than the experiment. Also, essentially, both incidences showed greater leading edge peaks in the experiment than in the prediction and large differences in the rear –Cp. This implies that there was a significant difference in the flow behaviour such as jets velocity/accelerations and reattachment/detachment.

7.3.2 Spanwise Distance Profiles Comparison of 1R -Inboard/Outboard Asymmetrically [Case: Mid Damage]

Profiles comparison of CFD vs. experiment for ‘1R’ inboard/outboard spanwise distance at midspan damaged with incidence of 4° and 8° are shown in Figure-VII.3a and VII.3b. Upstream of the hole, the impression was that the two figures of CFD/experiment at inboard/outboard in both incidences starting from leading edge up to 50%c were quite similar. However significant differences in –Cp peaks occurred at the leading edge area. At the downstream region [starting from 50%c], the differences between CFD and experiments and between inboard and outboard at all incidences were not as extreme as those seen in the centreline profiles [Figure-VII.2]. Downstream hole characteristics were similar to what seen in centreline profiles, but the characteristics were slightly different between 1R-inboard and 1R-outboard at both incidences. ‘Double peaks’ phenomena showed in inboard at 8°. This can be explained by the first peak being due to the influence of the accelerated jet exited from hole that was still having an affect until the distance of 1R. The second peak showed a sudden decline that was believed to be due to experimental error -- where the tapping reads low. This phenomenon is not seen in outboard results.

In terms of spanwise distances, consistency was shown in the CFD prediction with no indication of genuine asymmetry. This also adds support to the attribution of error in the experiments as being due to tapping.

Overall, it may be concluded that both incidences at both inboard/outboard showed greater leading edge peaks in the experiment than prediction, and large differences in the rear –Cp where peaks of both predictions are higher than for the experiments. CFD profiles tend to show the strong influence of a strongly accelerated jet with almost no influence of reverse flow from the trailing edge. However, in the experiment, the profiles showed no drastic change in velocity and pressure after the jet exited region, but showed strong influence of reverse flow from the trailing-
Pressure Measurement CFD vs. Experiments
Effect of Incidences - Alpha 4, 8 degrees

'1R' - Inboard - MID AR8

Figur-VII.3a [Inboard] CFD vs. Experiments for Spanwise Distance of '1R' - Inboard / Outboard at Incidence 4° and 8°
Pressure Measurement CFD vs. Experiments
Effect of Incidences - Alpha 4, 8 degrees

'1R' - Outboard - MID AR8

Figure-VII.3b|Outboard| CFD vs. Experiments for Spanwise Distance of '1R' – Inboard /Outboard at \( \alpha 4^\circ \) and \( \alpha 8^\circ \)
edge. These are indicated by the shape of the profile at the trailing edge in both CFD and experiment. Also, there is no indication of asymmetry in the CFD, whilst in the experiment, the asymmetric wake between inboard and outboard were clearly identified.

7.3.3 Profiles Comparison for Tip and Mid Damage Case: Centreline damage

Profile comparisons of CFD vs. experiment for centreline ['CL'] at tip and mid damage with the chosen incidence at 8° are shown in Figure-VII.4a and VII.4b.

Differences in \(-C_p\) a peak that occurred at leading edge and is the downstream regions were clearly seen in both CFD and experiment. At tip and mid damage, the leading edge peaks in the experiment were higher than the CFD and the experiment peaks always occurred earlier than those in the CFD. Soon after the peaks until the upstream edge, the trend of both CFD and experiment were almost at the same magnitude. In this upstream region, the slightly higher \(-C_p\) in experiment indicated that the trailing vortex at 8° affected the local pressure at the tip the more so in the CFD than in the experiment.

In the downstream region, the magnitudes of \(-C_p\) peaks difference between CFD and experiments were clearly indicated in all damage locations. In contrast to what was seen at leading edge peaks, the downstream peaks at tip and mid damage from the CFD prediction were higher and occurred earlier than in the experiment. It was seen that at the location where the CFD peak occurred, the experiment point was at the lowest position. This indicated that the CFD predict that the jet accelerated from the exit and reached the peak position at \(X/C\) 0.65, whilst in the experiment, the jet at the downstream edge remained decelerated and the speed was lower than at the upstream edge.

At this downstream region, both profiles indicated that the flow characteristics were different. The nature of the differences can then be determined from the flow visualisation shown in the following section [Section 7.4]. Also, in the downstream region, the CFD peak trend of tip and mid damage was also seen in experiment.

At the mid damage, the peaks of CFD and experiments were higher and located at the same chord position. In CFD, the region of reverse flow in all damage locations were limited to a region between \(X/C\) 0.6 to \(X/C\) 0.8, whilst in experiment, they covered the region of \(X/C\) 0.75 up to \(X/C\) 0.9. The wake velocity in CFD also reduced much more quickly than in experiment. The entrainment flow from the trailing edge in CFD was not as strong as in the experiment.

For all pressure cases shown, the summary is as follows:
- Forward of hole, reasonable agreement although CFD under-predicts leading edge suction peak.
- Downstream of hole, poor agreement.
**CFD vs. Experiment Profile Comparison**

**Tip Damaged - Alpha 8 deg.**

**Centreline - Damage**

![Graph showing CFD vs. Experiment Profile Comparison for Tip Damaged - Alpha 8 deg.](image)

**CFD vs. Experiment Profile Comparison**

**Mid Damaged - Alpha 8 deg.**

**Centreline - Damage**

![Graph showing CFD vs. Experiment Profile Comparison for Mid Damaged - Alpha 8 deg.](image)

**Figure-VII.4a [Tip] and VII.4b [Mid]:** CFD vs. Experiments for Centerline Damage at 8°

166
7.4 Weak and Strong Jets Visualisation Comparison

Case: Mid Damage – AR8

The characteristics of weak and strong jets are compared for CFD and experiment for the finite wing – AR8 configuration at 4°, 8° and 10°; with damaged hole located at the midspan. Figures VII.5a to VII-5c show the stream-trace visualisation of CFD predictions; and Figure-VII.6a to VII-6c show the flow visualisation of the experiments. To make the comparison easier, figures are arranged as seen. Both figures are at freestream Reynolds number of 5.5x10^5.

At the 4° CFD prediction [Figure-VII.5a], the flow structure is recognised as weak-jet, and for comparison with experimental results, the photographs shown in Figure-VII.6a was based on the flow visualisation at the same damage location and incidence.

In both figures, some recognised characteristics of weak jet can be identified such as the straight, symmetrical, narrow wake and its re-attachment and the formation of the counter-rotating-vortex-pair [CRVP]. Some significant differences between prediction and experiments can also be identified.

The CRVP distances relative to one another are about the same, but the intensity and the magnitude of the pairs were different. The size of the CRVP cores in the experiment is seen to be small and rather weak, whilst in the CFD prediction, cores or centres appeared to be relatively big with strong intensity. Also, the outboard CRVP in CFD is seen bigger than the inboard are.

The upstream hole reverse flow and the formation of a laminar separation bubble [LSB] were not seen in the CFD prediction, whilst in experiment, the LSB is clearly formed on both sides of the damage hole. In CFD, forward separation is not formed, it is only seen as deflected freestream flow and it has not separated. The secondary separation lines can still be identified though formed at about half of the size of the hole -- formed as a combination of the rolled-up jet flow at the lateral wall [Figure-IV.12b] with the deflected freestream flow. In experiment, both separation lines are clearly formed in front of the damage hole.

The re-attached flow is clearly identified in both figures, but the shape of the wakes is different. In CFD, the wake is narrow, almost straight, and symmetrical, and seems purely originated from the wake of damage that exited from the hole. Whilst, in the experiment the wake formed as a single wake at the inboard side and was slightly curved. Although the experiment was installed in the vertical position, this phenomenon has been assessed as a genuine effect. In addition, the wake in the CFD prediction seems less wide compared to the experiment.
Figure VII-5a. Weak-Jet Prediction - Alpha +4°.

Figure VII-5b. 'Transition'-Jet Prediction - Alpha +8°.

Figure VII-5c. Strong-Jet Prediction - Alpha +10°.

Figure VII-5d. Weak-Jet Experiments - Alpha +4°.

Figure VII-5e. 'Transition'-Jet Experiments - Alpha +8°.

Figure VII-5f. Weak-Jet Experiments - Alpha +10°.

Figure VII-5 Flows Structure Visualisation Comparison - Mid Damage - AR8
3D- CFD Prediction vs. Experiments
The differences seen in this flow visualisation support and explain the differences in Fluent’s results seen in the pressures and force coefficients increments.

Despite the similarity and differences shown in the weak-jet flow structure, the comparisons in the ‘transition’-jet at +8° [Figure-VII.5b and VII.6b] and in strong jet at +10° [Figure-VII.5c and VII.6c] are easier.

In CFD simulation Figure-VII.5b], -- although the incidence of +8° is recognised as an incidence of strong jet [in 2D tests], the features of a weak jet are still present. This can be indicated by the re-attachment of the flow -- slightly moved rearward compared to 4°. Here, it can be said that at 8°, the flow structure characteristic is in a ‘transition’ condition from weak to strong jet. However, all known characteristics of strong jet were also present in both CFD and experiment [Figure-VII.6b].

At +8°, the main differences were that the rather strong reverse flow that entrains from inboard and forms asymmetric vortices in the experiment is not shown in the CFD prediction. Indeed, there asymmetric vortices were also seen in CFD, but the size of the vortices seems bigger in the experiment, with more entrainment and a much larger wake behind the hole.

In figure-VII.5c and VII.6c that show both CFD and the experiment at +10°, the strong-jet characteristics were clearly identified. Flow in both figures was detached, there was a strong reverse flow, and the formation of large vortex centres. Both showed the asymmetry and size of the vortex centres, but in CFD, the pair of vortex centres was smaller than in the experiments. The location of these vortex centres was also different. In experiment, the centres were slightly more backward, whilst in CFD, the centres were rather close to the damage hole.

In conclusion, it can be said that the prediction of flow visualisation is in relatively agreement with what is seen in experiment but there are significant differences, particularly for weak jet.

The important point is that Fluent is poor for pressures especially downstream of damage and poor in coefficients increments. Observation of this flow visualisation characteristics account for Fluent’s relatively poor performance.
7.5 JICF, Empirical Model, and Cavity Phenomena into CFD and Experimental Results of Battle Damaged Wings

7.5.1 Introduction

This section is intended to summarise and incorporate the information gained in published work of jets and obstacle in crossflow [Ch.-2.3], the JICF empirical modelling applied to battle damaged wings [Ch.-2.3.1 and Ch.-5.6], and the JICF-cavity interaction phenomena [Ch.-4.7] into the results of CFD simulations and experiments of battle-damaged wings. The radial diagram relation of these five subjects that need to be incorporated is shown in Fig.-VII.6.

Discussion will be separated into two inter-related sub-discussions, i.e. summary discussion of JICF published work, empirical modelling/application, 2D/3D CFD simulation and JICF-Cavity interaction in one discussion, whilst discussion on 2D/3D CFD simulation and 2D/3D experiments already explained in detail in previous chapters. The relation between these two sub-discussions will then be incorporated into the final discussion of battle-damaged wings.

7.5.2. JICF, Empirical Modelling and JICF-Cavity Phenomena on Battle Damaged Wings

- JICF Published Work.

The very close similarity between battle damage and a ‘Jet in Crossflow’ [JICF] where the jet exiting from the damaged wing is interacting with the freestream flow is understandable as the main principle of the JICF. Several published work that have been undertaken to examine the characteristics of JICF that show and described the main feature of battle damaged wings were as follows: Margason [12]
JICF studies emphasized the low-\(r\) and high-\(r\) experimental cases, which were taken from Andreopoulos and Rodi in 1982 [13]. These low-\(r\) and high-\(r\) clearly represents ‘weak-jet’ and ‘strong-jet’ characteristics on battle-damaged wings. The most obvious feature of these two flow features is the mutual deflection of both jet and crossflow; and it is clear how these two flows differentiated. Crabb et.al in 1980 [46] investigated the velocity characteristics in the upstream and downstream regions. Velocities ratio \(r\) of 2.3 and 1.15 were used. These are categorised ‘strong-jet’ velocities ratio in battle damage term. From Margason [12] and Morton et. al. in 1996 [14] published work, it was confirmed that in battle damaged wings characteristics, deflected jets must necessarily contain embedded pairs of vortices of modest strength with vortex axes approximately parallel to the curved jet axis and occupying most of the jet cross section.

The water tunnel experiments by Haven et. al. of NASA in 1997 [15] examined the effect of hole exit geometry on the nearfield characteristics of crossflow jets. Haven found that the jet from the high-aspect-ratio holes, with increased separation distance between the sidewall vortices, stays attached to the surface for higher velocity ratios than for the low-aspect-ratio holes. The results of this published work was taken as an important reference to understand the separation and vortices characteristics of round hole geometry in battle damaged wings.

About the vortex-structure of the JICF investigation, Fric and Rosko in 1989 [55], worked with transverse jets of air in a wind tunnel. They identified and seen as well in the battle damaged wings flow visualisations [CFD and experiments] about four types of vortical structures [i.e. jet shear layer, vortex pair, horseshoe vortexes, and wake vortex street] that exist in the JICF near field.

In a number of published work cases, some deal with a jet expanding into a stream of fluid at an angle to it. Abramovich [47] described the mechanism as that of a jet of fluid entering into a flow, which is moving at an angle to the jet’s axis, is bent, and becomes curved. This was the actual condition of the battle-damaged wings where range of incidence was always applied during CFD simulations or experiments. The bent or curve of flow is depend on the wing incidence -- whether positive or negative, hi or low incidences, the JICF of battle damaged wings always characterise into two terms i.e. weak-jet and strong-jet.

Gustaffson [54] in 2001 conducted detailed three-component velocity measurements in the wake of an inclined jet in a cross-flow. Gustaffson found that in the wake, there were two counter-rotating foci close to the wall downstream of the jet that could be observed an indication of a 3D separation. In battle damaged wings flow visualisations, whether CFD and experiments the pairs of counter rotating vortex [CRVP] were always becomes a dominant feature.
JICF- Battle Damaged Wings Empirical Model Application.

Battle damaged wings that apply the JICF theory where jet discharged at oblique angles into the cross flow is relatively new research. There were some experimental works done but not published before 1985 [10]. So far, no development or application of empirical model into battle damaged wings until the present investigation.

From the earliest studies and published work of JICF empirical model, i.e. Shandorov [1957], Abramovich [47] in 1963, Vakhramov [1964], Visel & Mostinski [1965], Pratte and Baines [1967], Margason [1968], Ivanov [1971], Fearn and Weston in [45] in 1979, Margason [12] in 1993, and Demuren [48][49] in 1985/1994, empirical models were developed to correlate experimental data obtained under various idealized conditions, and most of which consider the jet trajectory. However, Margason [12] in 1993 expressions that compiled and reassessed from some of above investigators identified the best correlation for empirical model. It was based on this correlation, the empirical model applied into battle-damaged wings.

Since there were no measurements made to define the jet and freestream velocities during the experimental investigation therefore, vectors of velocity magnitude shown in 2D and 3D CFD predictions were used to define the jet velocity ratio, \( r \). The empirical model was applied to the generic battle damaged wing, i.e. damage hole size of 20% chord and the location at ‘mid-chord’ [25% and 50% chord].

For infinite [2D] battle damaged wings with damage hole at 25% chord, the conclusion was that none of the predictions is particularly good. For the weak jet, all of the predicted trajectories get further away from the surface. It is understood that transition from weak to a strong jet occurs at a lower value of \( r \) than what is shown for JICF. Due to that, similar situation for the strong jet where all of the predicted trajectories also get further away from the surface -- proving that the predicted shapes are essentially for a weak jet. Whilst, the application of empirical model to finite wing [3D] with damage hole at 50% chord concluded that the empirical jet trajectory for this finite wing is similar in shape to the 2D predictions. They are not in agreement to the jet shape from CFD prediction -- the predicted trajectories are within CFD jet, but the trajectories are completely different.

‘Cavity – Jet in Crossflow’ Interaction Phenomena

As the purpose of this discussion is to review that there is a cavity-JICF interaction in battle-damaged wings, therefore there is no direct connection to JICF empirical model. However, as described previously that the application of empirical model to battle damaged wings could only applied in CFD simulation, same situation for cavity-JICF interaction where CFD
Simulations were also used to identify and assess the phenomena. The reason was that smoke or oil flow visualisations experiments did not allow the interaction to be identified. So, the incorporation of JICF published work to empirical model and cavity in battle-damaged wings are correlated through CFD simulations.

For the battle-damaged wings, the cavity is the damage hole where the top and the bottom walls are both open. The range of the cavity cases for both weak and strong jets terms in battle-damaged wings can be calculated using criteria seen on the literature. So therefore, based on circular criteria, flow is expected to be open. However, battle damage cavity is a complicated form of cavity flow.

For the interpretation of the flow structure in the cavity and how this cavity-JICF interaction affects battle-damaged wing flow structure at the wing surface, the wake, and the jets-in-crossflow boundary are explained by the flow mechanism and development at the hole walls. The essence of the interaction can be concluded that features seen in the phenomena are completely different and more complicated form of open-flow cavity characteristics described on literature.

### 7.5.3 JICF, Empirical Model, and Cavity Correlation into CFD and Experimental Results of Battle Damaged Wings

This section discusses the inter-relation between section 7.5.2 into the results described in CFD simulations and the experiments throughout the thesis. Detail discussion of how the infinite damaged wings experiments done previously [1] compares with 2D CFD simulations is related to the comparison of 3D CFD simulations and finite battle-damaged wings experiments can be correlated as following:

1. Discussion of JICF published work experiments to empirical model and its application to cavity-JICF interaction phenomena in battle-damaged wings are incorporated through CFD simulations.

2. The results of the infinite battle-damaged wing experiments and 2D CFD simulations that described in Chapter-4 helps to gain and guide the understanding of the 3D CFD simulations and finite wing experiments [described in Chapter-7.1 to 7.4]. Here, the CFD simulation is the linking and the incorporate aspects.

3. From those two points above, the published work on JICF, empirical modelling and its application, and JICF-Cavity interaction phenomena are linked closely and incorporated into the CFD simulations and experiments throughout the thesis. Overall are concluded in the following Chapter.
Chapter 8
Conclusions

1. From the results of the CFD simulation of Irwin’s 2D quarter-chord [25%e] single-hole damage, the following conclusions were drawn:

   - Two chosen turbulence models i.e. K-e and Spalart-Allmaras [S-A] models work reasonably well in predicting jet flow structures, pressure distributions and Cp contours. In all cases, S-A performed better than the K-e model.
   - The comparison with Irwin’s 2D experimental results showed that the experimental Cp values on the damage centreline was always higher in the CFD simulation. It also appears that the interpretations of the presented flow structures, and the characteristics of the smoke/surface flow visualisations were all reasonable compared with the CFD prediction. However, CFD prediction was poor on coefficients increments.

2. From the assessment of the 2D and 3D CFD simulations, it was observed that there is a cavity and jet-in-crossflow interaction phenomena in the damage hole. Based on circular cavity criteria, the flow was expected to be open. However, in reality battle damage is a more complicated form of cavity flow.

3. Applying empirical models derived from jet-in-crossflow are not fully applicable to the battle-damaged wing case. The assessments for both 2D and 3D with weak and strong jets did not show satisfactory results. For the jet trajectory, none of the predictions was particularly good with the predicted trajectories being further from the wing surface than predicted by CFD.

4. For the application of Irwin’s 2D data to AR6, AR8, and AR10, the following conclusions were drawn:

   - An assessment of Irwin’s 2D force results led to the analysis that the tests in the AAE’s Low Turbulence Tunnel for 2D were under-predicting the damage effects at low incidence, and over-predicting at high incidences. At AR8 in particular, and at low incidences, the values of d[C_l] and d[C_d] were similar for mid damage and Irwin’s 3D prediction. However, as incidence was increased, Irwin’s data was significantly larger. This therefore suggested that Irwin’s 2D could not be used immediately to predict 3D.
From both weak and strong jets observations, the following conclusions were drawn:

I. The flow structure of surface visualisations undertaken at 50% c mid damage of AR8 shows that, despite the genuine asymmetric indication, some similarities to those identified in 2D also at 50% damage location exist.

II. The 3D flow structure asymmetry was believed to be due to three possibilities, i.e.: first, the effect of gravity because the model was installed vertically. Secondly, the effect of surface imperfection on the inboard side of the damage hole. This damage panel was always used to represent all spanwise damage locations. Third, was believed to be a genuine feature, neither due to gravity nor due to surface imperfections since it appears to be a similar shape for all damage locations.

III. For tip location, the reason of asymmetry was clearly identified as being due to the effect of the trailing vortex.

IV. An ‘early strong jet’ was indicated in both Infinite and Finite Wing-AR8. The only difference was that for Finite Wing-AR8 it happened at ‘transition incidence’ of 8°, whilst for Infinite wing an early strong jet indicated at ‘weak-jet incidence’ of 4°.

5. From the experimental results of half-chord [50% c] single-hole damage and from damage locations tested on a Finite Wing, the following conclusions were drawn:

For the lift-loss $d[C_l]$, drag increment $d[C_D]$, and pitching moment changes, $d[C_M]$ for AR6, AR8 and AR10, the conclusions were drawn as follows:

I. As damage moves out towards the tip, the lift-loss $d[C_l]$ is decreased, the drag increment, $d[C_D]$ also decreased, and the pitching moment changes, $d[C_M]$ indicate more to positive values at all incidence ranges and for all aspect ratios.

II. Increasing the incidence, results in greater magnitudes of $d[C_l]$ and $d[C_D]$ for all damage locations and aspect ratios. The highest $d[C_l]$ appeared in tip damage of AR6, whilst the highest $d[C_D]$ appeared in mid damage of AR8.

At the weak jet incidence 4° of AR8 and in all of three damage locations, the main characteristics of the weak-jet were illustrated clearly: forming an attached wake with minimal localised $C_p$ changes. The coefficient changes $d[C_l]$, $d[C_D]$, and $d[C_M]$ were relatively small. Whilst at 8°, the flow structure was indicated as transitional to stronger-jet; i.e. through-flow penetrated further into the crossflow; development of a wake region.
of separation, and reverse flow; but with the wake still reattached. At $10^6$, the flow structures were verified as strong jet.

- Strong-jet coefficient effects indicated greater $d[C_L]$, $d[C_D]$, and $d[C_M]$ than those seen for the weak jet; but not as significant as seen for 2D. Lift losses resulted primarily from the reduction in pressure peak upstream of the damage, whilst the $C_p$ distribution within the detached wake region resulted in a significant drag increment $d[C_D]$.

- The $C_p$ profiles of centreline damage at weak, 'transition' and strong jet incidences of midspan AR8 were consistent with the photographs where weak, and strong jets were present upstream and downstream of the hole. Also, the $C_p$ profiles in the vicinity of the damage hole were consistent with the flow patterns shown in the flow visualisation photographs.

6. The chosen Spalart-Allmaras solver in Irwin's 2D case was applied further for the prediction of half-chord [50%] single-hole damage in Finite Wing – AR8. This CFD prediction was then compared with the experimental results. The following conclusions were drawn:

- The magnitude of the aerodynamic coefficients shown in the predictions of both undamaged and damage wings were reasonable in linear part of the lift-curve, but predicted poor close to and beyond stall incidences. A similar situation was also seen in the drag and pitching moment predictions.

- In the coefficients changes prediction $d[C_L]$, $d[C_D]$, and $d[C_M]$ of a finite wing – which will feed into aircraft survivability analysis, the solver predicts the trend of $d[C_L]$ reasonably well but is not fully supported by the trend of $d[C_D]$ and $d[C_M]$. It also seems that the code can only be fully utilised in the limited positive incidences prior to stall.

- The solver [Fluent Spalart-Allmaras] was also assessed for the prediction of pressure distributions and flow visualisation. The conclusions were that predictions of $C_p$'s in centreline damage and spanwise distances showed general trends, i.e. well-predicted and good agreement at upstream and in the vicinity of the damage hole, but showed poorly at downstream. At both weak and strong jet incidences, the solver always predicts the $C_p$ lower at upstream and higher at downstream.

- CFD predicts weak, transition and strong jet visualisations for each case of a finite wing. The conclusion was that prediction of flow structures and characteristics were in relatively good agreement for the case of transition and strong jet incidences but slightly poor for weak jet.
From the comparison of the CFD prediction and experimental results, the following conclusions were drawn:

A. For the finite wing coefficients changes $d[CL]$, $d[C_D]$, and $d[C_M]$ comparison between CFD and experiment, the conclusion were drawn as follows:

I. In general trend, the three increments $d[CL]$, $d[C_D]$, and $d[C_M]$ were not in particularly good agreement. In the positive incidences up to $+6^\circ$, the increments were relatively good but at incidences prior to and beyond this, they were far from agreement. Fluent® also showed unrealistic predictions for negative incidences.

II. The greatest agreement among those increment comparisons were indicated for $d[CL]$, and covered the weak to strong jet incidences. The agreement in $d[C_D]$ was seen to be not as good as for $d[CL]$, but covered almost the same incidence range. The disagreement was considerable.

B. In showing weak-strong jet flow structures and characteristics, in CFD prediction, some characteristics show similarity. The experimental results validated the predictions reasonably.

C. For pressure profile comparisons, the conclusions were drawn as follows: Upstream of the damage for centreline, both inboard and outboard rows there was reasonable agreement, apart from the leading edge peak region. Downstream of the damage, CFD and experiment produce completely different shaped curves.
Chapter 9:
Recommended Further Work

Due to the complexity of the damage flow, it is recommended that further work be undertaken to improve the approaches made during CFD prediction and to validate them during the detailed experimental investigation. Further works are described in the following:

A. Further CFD Work

1. For 2D and 3D CFD assessments, using current damage shape, location and geometry: [Round, 20% c size, 50% c location] and the available CFD Code: Fluent
   - Grid Refinement and Mesh sizes Sensitivity: Current damage configuration [50% c location] was not simulated in 2D configuration, therefore 2D with more grid refinement [mainly in the leading edge area] and mesh size sensitivity should be performed. The results could confirm what was found in the 3D prediction [Chapter-5]. Grid refinement could also re-confirm the finding that the magnitude of the Cp values for damaged centreline and at spanwise 0.5r was always higher in CFD simulation compared to experimental results. Grid refinement could also improve the analysis of Cp contours.
   - RANS-Fluent Turbulence Models Sensitivity. Previous model studies were limited to K-e and SA where demonstration of jets flow structures, Cp contours, and pressure profiles were reasonably well represented by the Spalart-Allmaras [SA] model. Other turbulence models such as K-omega are worth investigating using the current geometry. The idea whether S-A is still good for all those above studies should be tested.

2. For 2D and 3D CFD assessments, vary and modify the existing damage shape, location, and geometry: [Round, 20% c size, 50% c location] using the available CFD Code: Fluent.
   - Modify to ‘star’ shape, unequal entry and exit hole size, damage sizes, variation in penetration angle, etc.
   - Rework on the grid refinement, mesh sizes and turbulence model sensitivities to define better and more accurate aerodynamic coefficients increments.
B. Further Experimental Work

1. Design and manufacture a new 2D model for use in the AAE wind tunnel for better accuracies for force, pressure, visualisation and detailed flow-fields with more advanced techniques such as traversing hot-wire or pressure probes.

This new 2D model [approximately 600mm] chord, can also be used for Reynolds number sensitivity, may also be required for further and better analysis of the upstream flow, internal cavity and damage wakes. These could only be achieved by a larger model provided with more dense pressure tappings, especially at upstream damage on both wing surfaces. This model should also be provided with removable panels that could accommodate the change of damage shape, size, unequal entry and exit, and variation in penetration angle.

Those possible further works are required based on the previous and current investigations and assessment. Testing with different damage shapes by maintaining the same percentage area lost, it would be possible to gain complete mapping of data applicability.

For further works on damage size, where the results have shown that the effects of damage change significantly with size, the situation where the damage hole has an exit hole larger than the entry hole should be considered.

Throughout, the previous and current investigations have shown that damage was modelled using the basic assumption that the penetration angle, i.e. direction of attack, would be at $90^\circ$ to the chord line. Tests should be undertaken to investigate combinations of damage with entry and exit holes at different chordwise locations.

An individual model of those modifications could be made for testing at AAE’s small Low Turbulence Wind Tunnel for this particular test objective.

2. For development of the experimental research into battle damage on a finite aspect ratio wing, a new finite half wing should be made but limited to a single aspect ratio e.g. AR8. The design and the concept of this 3D model should follow the objectives or tests possibilities defined for the 2D model. In addition, damage should be extended to include damage separated in a spanwise direction because surface pressure data has shown that spanwise influences may be extensive.

The base of the model may be configured as high aspect ratio sweep with constant and taper chords, different section profiles, and wing tip variations. Wing thickness variations could also give more realistic wing configurations.
References


14. Morton, B. R., Ibbetson, A., 'Jets Deflected in a Crossflow', Department of Mathematics, Monash University, Melbourne, Australia, Department of Meteorology, University of Reading, Reading, England


27. ESDU, ‘esdupac A9514 v1.0’, Fortran Source Code for Ref. [34], October 1995.


32. ESDU 74035, ‘Subsonic lift-dependent drag due to the trailing vortex wake for wings without camber or twist’, ESDU, April 1996.

181

34. Public Domain Aeronautical Software [PDAS] – P.O. Box 1438 Santa Cruz CA 95061 USA; Website : Free Download Stuff http://www.pdas.com/

35. Gambit v.2.0 Documentations: Fluent Europe, Sheffield, July 2002

36. Fluent v.5.4. Documentations: Fluent Europe, Sheffield, July 2002

37. Fluent v.6. Documentations [updated version]: Fluent Europe; September 2002


42. E-Mail Communications P.M.Render and M.Samad-Suhaeb, Period August-October 2005.


49. Domuren, A.O., 'Modelling Jets in Cross Flow', University of Karlsruhe, Germany, 1985


Appendix-A:
Airfoil Geometry and Published Data

NACA 641-412
[Applied Coordinates, Chord 200mm, Span 1000mm]

<table>
<thead>
<tr>
<th>Upper Surface</th>
<th>Lower Surface</th>
</tr>
</thead>
<tbody>
<tr>
<td>Station</td>
<td>Ordinate</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>0.68</td>
<td>3.50</td>
</tr>
<tr>
<td>1.14</td>
<td>4.20</td>
</tr>
<tr>
<td>2.10</td>
<td>5.02</td>
</tr>
<tr>
<td>4.52</td>
<td>6.40</td>
</tr>
<tr>
<td>9.48</td>
<td>8.40</td>
</tr>
<tr>
<td>14.46</td>
<td>9.26</td>
</tr>
<tr>
<td>19.50</td>
<td>10.12</td>
</tr>
<tr>
<td>24.54</td>
<td>10.56</td>
</tr>
<tr>
<td>49.62</td>
<td>15.62</td>
</tr>
<tr>
<td>59.68</td>
<td>16.28</td>
</tr>
<tr>
<td>69.76</td>
<td>16.62</td>
</tr>
<tr>
<td>79.84</td>
<td>16.60</td>
</tr>
<tr>
<td>89.92</td>
<td>16.20</td>
</tr>
<tr>
<td>100.00</td>
<td>15.40</td>
</tr>
<tr>
<td>110.06</td>
<td>14.36</td>
</tr>
<tr>
<td>120.12</td>
<td>13.06</td>
</tr>
<tr>
<td>130.16</td>
<td>11.56</td>
</tr>
<tr>
<td>140.18</td>
<td>9.86</td>
</tr>
<tr>
<td>150.18</td>
<td>8.04</td>
</tr>
<tr>
<td>160.16</td>
<td>6.14</td>
</tr>
<tr>
<td>170.12</td>
<td>4.12</td>
</tr>
<tr>
<td>180.12</td>
<td>2.18</td>
</tr>
<tr>
<td>190.06</td>
<td>0.20</td>
</tr>
<tr>
<td>200.00</td>
<td>-1.93</td>
</tr>
</tbody>
</table>

Source: NACA Report No. 824: Summary of Airfoil Data, pp 446


184
Appendix-B:
General Assembly of Experimental Rig:

Mounting Structure and Wind Tunnel Model

<table>
<thead>
<tr>
<th>NO.</th>
<th>PART NAME</th>
<th>MATERIAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
<td>Pin for positioning Model Panels</td>
<td>Alloy St. Dia 5mm</td>
</tr>
<tr>
<td>15</td>
<td>Wind Tunnel Model</td>
<td>Composite</td>
</tr>
<tr>
<td>14</td>
<td>Trailing Rod</td>
<td>A356 St. Rod</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Dia 8 mm</td>
</tr>
<tr>
<td>13</td>
<td>Lead Rod</td>
<td>A356 St. Rod</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Dia 14 mm</td>
</tr>
<tr>
<td>12</td>
<td>Turntable Cover Screws</td>
<td>8d. Screws M2</td>
</tr>
<tr>
<td>11</td>
<td>Turntable Cover</td>
<td>Thckwood 25mm</td>
</tr>
<tr>
<td>10</td>
<td>M8 and M10 Tapped</td>
<td>-</td>
</tr>
<tr>
<td>9</td>
<td>Block-Plate Dowel</td>
<td>Alloy St. Dia 8</td>
</tr>
<tr>
<td>8</td>
<td>Block-Plate Bolts</td>
<td>M10</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Materia Grade 29</td>
</tr>
<tr>
<td>7</td>
<td>Interface Block</td>
<td>A356 Plate 10mm</td>
</tr>
<tr>
<td>6</td>
<td>Upper Plate</td>
<td>A356 Plate 10mm</td>
</tr>
<tr>
<td>5</td>
<td>Main Structure Web</td>
<td>A356 St Plate 3mm</td>
</tr>
<tr>
<td>4</td>
<td>Main Structure s</td>
<td>A356 St. Plate</td>
</tr>
<tr>
<td></td>
<td></td>
<td>10 mm</td>
</tr>
<tr>
<td>3</td>
<td>Balance-Plate Bolts</td>
<td>Materia Grade 29</td>
</tr>
<tr>
<td></td>
<td></td>
<td>A356</td>
</tr>
<tr>
<td>2</td>
<td>Balance-Plate Dowel</td>
<td>Alloy St. Dia 10</td>
</tr>
<tr>
<td>1</td>
<td>Lower Base Plate</td>
<td>Steel A356</td>
</tr>
<tr>
<td></td>
<td></td>
<td>ASTM Grade</td>
</tr>
</tbody>
</table>
APPENDIX-C:
Hardware Arrangement and Schematic for Data Acquisition System
APPENDIX-D:
Applied Formulas and Equations for Wall Corrections

D.I Analytical Assessment: Blockages Corrections

D.I.1 Barlow's Method on Solid Blockage

D.I.1.1 Solid Blockage (due to model volume)
Blockage corrections were created by the reduction in the cross-sectional area of the tunnel, due to the presence of the model and its wake. To maintain continuity in the airflow, the constricted flow must increase in velocity in the region of the model. This blockage results in aerodynamic forces greater than those found in free air conditions because these forces are proportional to the square of the surrounding fluid velocity.

The solid blockage correction for three aspect ratios (AR6, AR8 and AR10) of the model was based on the philosophy that the wing was represented by a source-sink distribution and was contained in the tunnel walls by an infinite distribution of image. The solid blockage velocity effect for a wing,

\[ \Delta V_{sb} = \frac{V_{s1}(Wing Volume)}{K_{ij} C^{3/2}} \]

where \( K_{ij} \) is the wing shape factor of NACA 641-412, \( r_{ij} \) is a factor of AAE’s rectangular wind tunnel cross section and wing span-to-tunnel-width ratio.

In AAE’s balance position and test section situation, the wing was installed vertically, and consequently, the height (1.32m) re-oriented to ‘width’ and width (1.92m) re-oriented to ‘height’.

D.I.2 Maskell’s Methods on: Wake Blockage, Dynamic Pressure, and Velocity Corrections

D.I.2.1 Wake Blockage (due to wake shed from the model).
This effect is a result of the finite size of the wing wake. The magnitude of the correction for wake blockage increases with an increase of wake size, which corresponds to an increase in drag. Maskell has reported, as cited by Barlow that the necessity of considering the momentum effects outside the wake when separated flow occurs. The wake blockage for separated flow is then: Maskell added a term to account for the increased velocity outside the wake and its
consequently lowered pressure. Dividing the total drag coefficient into a constant amount $C_{DO}$, one proportional to $C_{2}$, and one due to separated flow $C_{Dn}$, Maskell obtains the total wake blockage correction $\epsilon_{wb}$, as

$$\epsilon_{wb} = \frac{S}{4C} C_{DO} + \frac{5S}{4C} (C_{Dn} - C_D - C_{DO})$$

...[2]

Where $S$, was wing area based on aspect ratios. The total blockage correction was given as

$$\epsilon_T = \epsilon_{ib} + \epsilon_{wb}$$

...[3]

It was realised that the blockage correction were required to produce the correct dynamic pressure that was used to calculate all coefficients, and for that, Maskell defined the formula as

$$\frac{q_c}{q_u} = 1 + \frac{S}{2C} C_{DO} + \frac{5S}{2C} (C_{Dn} - C_D - C_{DO})$$

...[4]

Equation 4 may then be simplified as equation 5 and velocity correction as equation 6.

$$q_c = q_u (1 + \epsilon_T)^2$$

...[5]

$$V_c = V_u (1 + \epsilon_T)^2$$

...[6]

D.II Analytical Assessment: Upwash Corrections

D.II.1 ‘ESDU’s Methods on Upwash Interference
(Subsonic Linearised Theory)

As known, the wall-induced upwash is uniform, or nearly so, in the region occupied by the wing, which will effectively changes the local wing incidence by a constant amount over the planform. As the upwash interference is concerned, the flow about the test wing in the wind tunnel is then the same as that about the test wing in free air at the 'corrected' value of incidence. The corrected incidence was given by,

$$\bar{\alpha}_{corr} = \bar{\alpha} + \Delta \bar{\alpha}.$$

...[7]

where $\bar{\alpha}$, geometric incidence at the wing chord.
The effective additional incidence of datum chord was obtained by,

$$\Delta \alpha = \delta \frac{SC}{c}$$

and $\delta_{oa}$ mean value of upwash interference parameter ($\delta$) over wing planform.

Lift, drag and moment coefficients, corrected for upwash interference, were obtained by taking the equivalent free-air undisturbed stream to be inclined upwards at the angle $\Delta \alpha$ to the $x$-axis of the corresponding wind tunnel axes.

$$c_{corr} = c_{L} - c_{D} \Delta \alpha_{L}$$

$$c_{Dcorr} = c_{D} + c_{L} \Delta \alpha_{D}$$

$$c_{Mcorr} = c_{M}$$

The incidence correction related to lift, $\Delta \alpha_{L}$ and the incidence correction related to drag $\Delta \alpha_{D}$ were defined by,

$$\Delta \alpha_{L} = \delta \frac{SC_{L}}{c}$$

$$\Delta \alpha_{D} = \delta \frac{SC_{L}}{c}$$

All of the upwash interference parameters, $\delta_{oa}, \Delta \alpha_{L}$, and $\Delta \alpha_{D}$ were numerically obtained by the computer program ‘A9514 (Version 1.0 ESDUpac A9514v10for)’ [34] which also provides results for a number of upwash-interference related parameters that have been evaluated by the methods described above. The code and data were modified slightly to incorporate the AR6, AR8 and AR10 plan-forms respectively.

D.III ‘AGARD-336’s Methods on Wing Lift Interference (Finite-span Horseshoe Vortex)

For a small wing model and small upwash angle, additional upwash at the model location due to the walls requires corrections to angle of attack and drag — due to the change in effective stream direction at the model location. The corrected $C_{D}$ and $\alpha$ analytically were then calculated as,
\[ C_{Dcorr} = C_{Duncorr} \cos \Delta \alpha + C_{Luncorr} \sin \Delta \alpha \equiv C_{Duncorr} + C_{Duncorr} \Delta \alpha \] \[ \alpha_{corr} = \alpha_{uncorr} + \left( -\delta_0 + \frac{c}{2 \beta H} \right) \frac{S_{Cluncorr}}{C} \] ...[15]

where \( \Delta \alpha, \alpha \) was evaluated at the wing centre of lift (the wing 0.25 chord location), defined as

\[ \Delta \alpha = \delta_0 \frac{S}{C} Cluncorr \] ...[16]

and \( \delta_0 \) as average interference parameter at the centre of the lift; \( \delta_1 \) as streamwise curvature interference parameter, \( c \) as m.a.c, \( \beta \) as Prandtl-Glauert compressibility factor, and \( H \) as tunnel height.

Alternatively, due to the streamwise gradient of the upwash interference, the correction was required to pitching moment and lift or pitching moment and angle of attack. As above, the corrected \( C_L \) and \( C_M \) were then calculated as,

\[ C_{Lcorr} = C_{Luncorr} \cos \Delta \alpha - C_{Duncorr} \sin \Delta \alpha \equiv C_{Luncorr} \] ...[17]

\[ \Delta C_M = \delta_1 \frac{c}{16 \beta H} \frac{S_{Cluncorr} \partial C_L}{C \partial \alpha} \] ...[18]

where \( \frac{\partial C_L}{\partial \alpha} \), a slope that was defined from Irwin’s 2d experimental results.
APPENDIX-E:
Sample of Battle Damaged Wing Empirical Calculation

The centreline of the jet trajectory is defined by empirical methods of Margason [with Ivanov’s exponents] that shown in Chapter-5. The equation [6] is:

\[ \frac{x}{D} = F V_e^{n} \left( \frac{z}{D} \right)^{m} + \frac{z}{D} G \cot(\theta) \]

Where:

- \( x \) and \( z \) are the coordinate points of horizontal and vertical axes of the jet.
- \( D \) is the diameter of the hole = 40 mm
- \( \theta \) is the angle between the jet’s axis and the direction of the deflecting flow. \( \theta = 90^\circ - \alpha_{wing} \);

For typical ‘weak-jet’ \( \theta = 86^\circ \) and for ‘strong-jet’ \( \theta = 82^\circ \)

\( F, G \) are jet trajectory coefficients; whilst \( n, m \) are jet trajectory exponents;

From Table-II.1: Jet Trajectory equations terms for arbitrary Jet angle of Ivanov:

\( F = 1; \ G = 1; \ n = 3.6; \) and \( m = 3. \)

\( V_e \) is effective velocity ratio \( V_e = \frac{1}{r} \) \([r \text{ values defined from Eq.1; Table-V.2 and Table-V.3}].\)

\( V_{\mu e} \) and \( V_{\text{freesam}} \) were defined by assessing the ‘colour scale’ of Figure-IV.11[a] and Fig.-IV.11[b] for infinite wing [damage at 25% chord] and Figure-IV.13[a] and Fig.-IV.13[b] for finite wing [damage at 50% chord]. Both represents weak and strong jets trajectory.

Using the equation [6] above, the sample of the calculation for weak-jet trajectory \([x/D]\) of infinite wing by taking \( z \) value started at e.g 3 mm above the \([0,0]\) is shown below; and for strong-jet trajectory, the \( z \) value started at e.g. 5 mm. The same method is used for finite wing.

The complete results of trajectory lines for both cases are shown in Table – E1[a] and E1[b] for infinite wing and Table – E2[a] and E2[b] for finite wing.

Sample of calculation for weak and strong jets trajectory for infinite wing:

<table>
<thead>
<tr>
<th>( V_{\text{e}} ) (mm/s)</th>
<th>( V_{\mu e} ) (mm/s)</th>
<th>( V_{\text{freesam}} ) (mm/s)</th>
<th>( F )</th>
<th>( G )</th>
<th>( n )</th>
<th>( m )</th>
<th>( x ) (mm)</th>
<th>( z ) (mm)</th>
<th>( D ) (mm)</th>
<th>( \beta ) (deg)</th>
<th>( \alpha ) (deg)</th>
<th>( \cot \theta )</th>
<th>( x/D )</th>
<th>( X ) (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>20.8</td>
<td>48</td>
<td>8.10754</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>3</td>
<td>40</td>
<td>0.036</td>
<td>0.078</td>
<td>0.781</td>
<td>0.883</td>
<td>1.192219</td>
<td>0</td>
</tr>
<tr>
<td>20.8</td>
<td>48</td>
<td>8,10754</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>3</td>
<td>40</td>
<td>0.036</td>
<td>0.078</td>
<td>0.781</td>
<td>0.883</td>
<td>1.192219</td>
<td>0</td>
</tr>
</tbody>
</table>

Sample of calculation for weak and strong jets trajectory for finite wing:

<table>
<thead>
<tr>
<th>( V_{\text{e}} ) (mm/s)</th>
<th>( V_{\mu e} ) (mm/s)</th>
<th>( V_{\text{freesam}} ) (mm/s)</th>
<th>( F )</th>
<th>( G )</th>
<th>( n )</th>
<th>( m )</th>
<th>( x ) (mm)</th>
<th>( z ) (mm)</th>
<th>( D ) (mm)</th>
<th>( \beta ) (deg)</th>
<th>( \alpha ) (deg)</th>
<th>( \cot \theta )</th>
<th>( x/D )</th>
<th>( X ) (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>19.6</td>
<td>48</td>
<td>8.675391</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>3</td>
<td>40</td>
<td>0.036</td>
<td>0.078</td>
<td>0.781</td>
<td>0.883</td>
<td>1.58616844</td>
<td>0</td>
</tr>
<tr>
<td>19.6</td>
<td>48</td>
<td>8.675391</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>3</td>
<td>40</td>
<td>0.036</td>
<td>0.078</td>
<td>0.781</td>
<td>0.883</td>
<td>1.58616844</td>
<td>0</td>
</tr>
</tbody>
</table>

\( \beta \) is effective velocity ratio \( \beta = \frac{1}{r} \) \([r \text{ values defined from Eq.1; Table-V.2 and Table-V.3}].\)

The complete results of trajectory lines for both cases are shown in Table – E1[a] and E1[b] for infinite wing and Table – E2[a] and E2[b] for finite wing.
JET TRAJECTORY OF INFINITE WING FOR DAMAGE HOLE AT 25% CHORD

Margason Equation - Ivanov's Exponents

\[ X = \frac{FV^e}{G} \left( \frac{z}{D} \right)^e + \frac{z}{D} \cot(\theta) \]

WEAK JET - Alpha +4deg.

<table>
<thead>
<tr>
<th>Vj (m/s)</th>
<th>Vr (m/s)</th>
<th>Ve exp n</th>
<th>F</th>
<th>G</th>
<th>n</th>
<th>m</th>
<th>y or z (mm)</th>
<th>D (mm)</th>
<th>y/D or z/D</th>
<th>Theta</th>
<th>Cot Theta</th>
<th>x/D</th>
<th>X (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>20.9</td>
<td>40</td>
<td>5,40724</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>0</td>
<td>40</td>
<td>0</td>
<td>86</td>
<td>1,559168944</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>20.9</td>
<td>40</td>
<td>5,40724</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>3</td>
<td>40</td>
<td>0.075</td>
<td>86</td>
<td>1,559168944</td>
<td>0.119219</td>
<td>4,768754</td>
</tr>
<tr>
<td>20.9</td>
<td>40</td>
<td>5,40724</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>6</td>
<td>40</td>
<td>0.16</td>
<td>86</td>
<td>1,559168944</td>
<td>0.252125</td>
<td>10,08499</td>
</tr>
<tr>
<td>20.9</td>
<td>40</td>
<td>5,40724</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>9</td>
<td>40</td>
<td>0.225</td>
<td>86</td>
<td>1,559168944</td>
<td>0.412405</td>
<td>16,45619</td>
</tr>
<tr>
<td>20.9</td>
<td>40</td>
<td>5,40724</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>12</td>
<td>40</td>
<td>0.3</td>
<td>86</td>
<td>1,559168944</td>
<td>0.613746</td>
<td>24,54985</td>
</tr>
<tr>
<td>20.9</td>
<td>40</td>
<td>5,40724</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>15</td>
<td>40</td>
<td>0.375</td>
<td>86</td>
<td>1,559168944</td>
<td>0.858036</td>
<td>34,79343</td>
</tr>
<tr>
<td>20.9</td>
<td>40</td>
<td>5,40724</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>18</td>
<td>40</td>
<td>0.45</td>
<td>86</td>
<td>1,559168944</td>
<td>1.194381</td>
<td>47,77443</td>
</tr>
<tr>
<td>20.9</td>
<td>40</td>
<td>5,40724</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>21</td>
<td>40</td>
<td>0.525</td>
<td>86</td>
<td>1,559168944</td>
<td>1.601008</td>
<td>64,94033</td>
</tr>
<tr>
<td>20.9</td>
<td>40</td>
<td>5,40724</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>24</td>
<td>40</td>
<td>0.6</td>
<td>86</td>
<td>1,559168944</td>
<td>2.103485</td>
<td>84,13861</td>
</tr>
<tr>
<td>20.9</td>
<td>40</td>
<td>5,40724</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>27</td>
<td>40</td>
<td>0.675</td>
<td>86</td>
<td>1,559168944</td>
<td>2.715419</td>
<td>108,6168</td>
</tr>
<tr>
<td>20.9</td>
<td>40</td>
<td>5,40724</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>30</td>
<td>40</td>
<td>0.75</td>
<td>86</td>
<td>1,559168944</td>
<td>3.450556</td>
<td>138,0222</td>
</tr>
<tr>
<td>20.9</td>
<td>40</td>
<td>5,40724</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>33</td>
<td>40</td>
<td>0.825</td>
<td>86</td>
<td>1,559168944</td>
<td>4.322564</td>
<td>172,9028</td>
</tr>
<tr>
<td>20.9</td>
<td>40</td>
<td>5,40724</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>35</td>
<td>40</td>
<td>0.9</td>
<td>86</td>
<td>1,559168944</td>
<td>5.345153</td>
<td>213,3052</td>
</tr>
</tbody>
</table>

Table - Ell[a]: Calculation of Equation [6] Parameters for Weak Jet Trajectory—Damage at 25% Chord

STRONG JET - Alpha +8deg

<table>
<thead>
<tr>
<th>Vj (m/s)</th>
<th>Vr (m/s)</th>
<th>Ve exp n</th>
<th>F</th>
<th>G</th>
<th>n</th>
<th>m</th>
<th>y or z (mm)</th>
<th>D (mm)</th>
<th>y/D or z/D</th>
<th>Theta</th>
<th>Cot Theta</th>
<th>x/D</th>
<th>X (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>40.9</td>
<td>40</td>
<td>0.94379</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>0</td>
<td>40</td>
<td>0</td>
<td>82</td>
<td>1,558601809</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>40.9</td>
<td>40</td>
<td>0.94379</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>5</td>
<td>40</td>
<td>0.125</td>
<td>82</td>
<td>1,558601809</td>
<td>0.196669</td>
<td>7,866743</td>
</tr>
<tr>
<td>40.9</td>
<td>40</td>
<td>0.94379</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>10</td>
<td>40</td>
<td>0.25</td>
<td>82</td>
<td>1,558601809</td>
<td>0.404397</td>
<td>16,17589</td>
</tr>
<tr>
<td>40.9</td>
<td>40</td>
<td>0.94379</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>15</td>
<td>40</td>
<td>0.375</td>
<td>82</td>
<td>1,558601809</td>
<td>0.634246</td>
<td>25,36893</td>
</tr>
<tr>
<td>40.9</td>
<td>40</td>
<td>0.94379</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>20</td>
<td>40</td>
<td>0.5</td>
<td>82</td>
<td>1,558601809</td>
<td>0.827775</td>
<td>35,89099</td>
</tr>
<tr>
<td>40.9</td>
<td>40</td>
<td>0.94379</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>25</td>
<td>40</td>
<td>0.625</td>
<td>82</td>
<td>1,558601809</td>
<td>1.204544</td>
<td>48,18174</td>
</tr>
<tr>
<td>40.9</td>
<td>40</td>
<td>0.94379</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>30</td>
<td>40</td>
<td>0.75</td>
<td>82</td>
<td>1,558601809</td>
<td>1.567113</td>
<td>62,68451</td>
</tr>
<tr>
<td>40.9</td>
<td>40</td>
<td>0.94379</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>35</td>
<td>40</td>
<td>0.875</td>
<td>82</td>
<td>1,558601809</td>
<td>1.996042</td>
<td>79,84169</td>
</tr>
<tr>
<td>40.9</td>
<td>40</td>
<td>0.94379</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>40</td>
<td>40</td>
<td>1</td>
<td>82</td>
<td>1,558601809</td>
<td>2.502392</td>
<td>100,0957</td>
</tr>
<tr>
<td>40.9</td>
<td>40</td>
<td>0.94379</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>45</td>
<td>40</td>
<td>1.125</td>
<td>82</td>
<td>1,558601809</td>
<td>3.097222</td>
<td>123,8889</td>
</tr>
</tbody>
</table>

Table - Ell[b]: Calculation of Equation [6] Parameters for Strong Jet Trajectory—Damage at 25% Chord
## JET TRAJECTORY OF FINITE WING FOR DAMAGE HOLE AT 50% CHORD

Mergason Equation - Ivanov's Exponents

\[ \frac{x}{D} = F V \frac{z}{D} \left( \frac{z}{D} \right)^n + \frac{z}{D} G \cot(\theta) \]

### WEAK JET - Alpha +4deg.

<table>
<thead>
<tr>
<th>Vj (m/s)</th>
<th>Vepsilon (m/s)</th>
<th>Ve exp n</th>
<th>F</th>
<th>G</th>
<th>n</th>
<th>m</th>
<th>y or z (mm)</th>
<th>D [mm]</th>
<th>y/D or z/D</th>
<th>Theta</th>
<th>Cot Theta</th>
<th>x/D</th>
<th>X (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>19.5</td>
<td>40</td>
<td>6,475391</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>0</td>
<td>40</td>
<td>0</td>
<td>86</td>
<td>1,559168944</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>19.5</td>
<td>40</td>
<td>6,475391</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>3</td>
<td>40</td>
<td>0.075</td>
<td>86</td>
<td>1,559168944</td>
<td>0.119669</td>
<td>4,756779</td>
</tr>
<tr>
<td>19.5</td>
<td>40</td>
<td>6,475391</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>6</td>
<td>0</td>
<td>40</td>
<td>0.15</td>
<td>86</td>
<td>1,559168944</td>
<td>0.25573</td>
<td>10,22919</td>
</tr>
<tr>
<td>19.5</td>
<td>40</td>
<td>6,475391</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>6</td>
<td>9</td>
<td>40</td>
<td>0.225</td>
<td>86</td>
<td>1,559168944</td>
<td>0.424572</td>
<td>18,58287</td>
</tr>
<tr>
<td>19.5</td>
<td>40</td>
<td>6,475391</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>9</td>
<td>0</td>
<td>40</td>
<td>0.3</td>
<td>86</td>
<td>1,559168944</td>
<td>0.642586</td>
<td>25,70345</td>
</tr>
<tr>
<td>19.5</td>
<td>40</td>
<td>6,475391</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>12</td>
<td>16</td>
<td>40</td>
<td>0.375</td>
<td>86</td>
<td>1,559168944</td>
<td>0.926184</td>
<td>37,046569</td>
</tr>
<tr>
<td>19.5</td>
<td>40</td>
<td>6,475391</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>18</td>
<td>0</td>
<td>40</td>
<td>0.45</td>
<td>86</td>
<td>1,559168944</td>
<td>1.291696</td>
<td>51,56784</td>
</tr>
<tr>
<td>19.5</td>
<td>40</td>
<td>6,475391</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>21</td>
<td>0</td>
<td>40</td>
<td>0.625</td>
<td>86</td>
<td>1,559168944</td>
<td>1.765573</td>
<td>70,22292</td>
</tr>
<tr>
<td>19.5</td>
<td>40</td>
<td>6,475391</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>25</td>
<td>0</td>
<td>40</td>
<td>0.625</td>
<td>86</td>
<td>1,559168944</td>
<td>2.127555</td>
<td>85,10219</td>
</tr>
</tbody>
</table>

Table - E2 [a]: Calculation of Equation [6] Parameters for Weak Jet Trajectory for Damage at 50% Chord

### STRONG JET - Alpha +8deg.

<table>
<thead>
<tr>
<th>Vj (m/s)</th>
<th>Vepsilon (m/s)</th>
<th>Ve exp n</th>
<th>F</th>
<th>G</th>
<th>n</th>
<th>m</th>
<th>y or z (mm)</th>
<th>D [mm]</th>
<th>y/D or z/D</th>
<th>Theta</th>
<th>Cot Theta</th>
<th>x/D</th>
<th>X (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>37.5</td>
<td>40</td>
<td>1,1827</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>0</td>
<td>40</td>
<td>0</td>
<td>82</td>
<td>1,558601809</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>37.5</td>
<td>40</td>
<td>1,1827</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>3</td>
<td>5</td>
<td>40</td>
<td>0.125</td>
<td>82</td>
<td>1,558601809</td>
<td>0.197135</td>
<td>7,885408</td>
</tr>
<tr>
<td>37.5</td>
<td>40</td>
<td>1,1827</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>6</td>
<td>0</td>
<td>40</td>
<td>0.25</td>
<td>82</td>
<td>1,558601809</td>
<td>0.40813</td>
<td>18,32521</td>
</tr>
<tr>
<td>37.5</td>
<td>40</td>
<td>1,1827</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>15</td>
<td>0</td>
<td>40</td>
<td>0.375</td>
<td>82</td>
<td>1,558601809</td>
<td>0.646845</td>
<td>25,87379</td>
</tr>
<tr>
<td>37.5</td>
<td>40</td>
<td>1,1827</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>20</td>
<td>0</td>
<td>40</td>
<td>0.5</td>
<td>82</td>
<td>1,558601809</td>
<td>0.927138</td>
<td>37,08554</td>
</tr>
<tr>
<td>37.5</td>
<td>40</td>
<td>1,1827</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>25</td>
<td>0</td>
<td>40</td>
<td>0.625</td>
<td>82</td>
<td>1,558601809</td>
<td>1.262871</td>
<td>50,51485</td>
</tr>
<tr>
<td>37.5</td>
<td>40</td>
<td>1,1827</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>30</td>
<td>0</td>
<td>40</td>
<td>0.75</td>
<td>82</td>
<td>1,558601809</td>
<td>1.667903</td>
<td>65,71612</td>
</tr>
<tr>
<td>37.5</td>
<td>40</td>
<td>1,1827</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>35</td>
<td>0</td>
<td>40</td>
<td>0.875</td>
<td>82</td>
<td>1,558601809</td>
<td>2,156093</td>
<td>88,24373</td>
</tr>
<tr>
<td>37.5</td>
<td>40</td>
<td>1,1827</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>38</td>
<td>0</td>
<td>40</td>
<td>0.95</td>
<td>82</td>
<td>1,558601809</td>
<td>2.494689</td>
<td>99,78757</td>
</tr>
<tr>
<td>37.5</td>
<td>40</td>
<td>1,1827</td>
<td>1</td>
<td>1</td>
<td>2.6</td>
<td>40</td>
<td>0</td>
<td>40</td>
<td>1</td>
<td>82</td>
<td>1,558601809</td>
<td>2.741302</td>
<td>109,6521</td>
</tr>
</tbody>
</table>

Table - E2 [b]: Calculation of Equation [6] Parameters for Strong Jet Trajectory for Damage at 50% Chord