

© Copyright 2021

Jer-Yo Lee

Important Issues for Designing High Lift Common Research Model

Jer-Yo Lee

A thesis

submitted in partial fulfillment of the
requirements for the degree of

Master of Science in Aeronautics and Astronautics

University of Washington

2021

Reading Committee:

Dana Dabiri, Chair

Mohammed Reza Soltani

Program Authorized to Offer Degree:

Aeronautics and Astronautics

University of Washington

Abstract

Important Issues for Designing High Lift Common Research Model

Jer-Yo Lee

Chair of the Supervisory Committee:

Professor Dana Dabiri

William E. Boeing Department of Aeronautics and Astronautics

Accurate and reliable aerodynamic data even for civil aircraft are often kept by their manufacturers and are not available to the public. Moreover, researchers would like to further investigate about the performance of this vehicle or to evaluate their newly developed CFD code. To fulfil this gap, NASA decided to design and built a model (HL-CRM) of a representative of a transonic civil aircraft and make it available to all researchers. This model has been and will be tested in various wind tunnels as well as the CFD development community. All data will be published, hence everybody who is interested in this field can use the data to improve the understanding of aerodynamics. The University of Washington cooperates with Boeing to build a HL-CRM model to serve as a platform for innovative research and to provide educational courses to students. The first stage of building the model is to consider all the potential functions for our HL-CRM. 17 important issues have been listed by the experts in UW and Boeing. Hence, the propose of this paper is to investigate the 17 conceptual design considerations for designing HL-CRM. Since it is difficult to study all the issues on the model, the priority of the issues is discussed eventually.

TABLE OF CONTENTS

List of Figures	1
List of Tables	11
Chapter 1. Introduction	13
1.1 High Lift Common Research Modal.....	13
1.2 Previous Work Related to HL-CRM.....	14
1.2.1 EUROLIFT Research Programme.....	14
1.2.2 JAXA High Lift Program.....	15
1.2.3 NASA HL-CRM.....	17
1.3 Objectives	18
Chapter 2. Important issues.....	20
2.1 Cruise, Takeoff, and Landing Configurations	20
2.2 Adjustable Leading and Trailing Edge Brackets	26
2.3 Configurations with and without Nacelle/Pylon.....	32
2.4 Noise and Main Landing Gear.....	43
2.5 Removable Horizontal and Vertical Tails.....	50
2.6 Flow Visualization and Off Body Flow Field Studies.....	63
2.7 Boundary Layer Studies.....	74
2.8 Considerations for Multi-Element Flaps.....	97
2.9 Interchangeable Pressure-Instrumented Wingtips	88
2.10 Ice Shape Testing.....	88

2.11	Active Flow Control	117
2.12	Low Reynolds Number Computation Fluid Dynamics Validation.....	128
2.13	Capabilities for Hinge Moment Measurements	131
2.14	Airframe Noise Testing.....	140
2.15	Capabilities for Leading and Trailing Edge Positioning Studies.....	150
2.16	Wakes Studies	158
2.17	Transition Studies	167
Chapter 3. Conclusions and Future Studies		176
3.1	Conclusions.....	176
3.2	Future Studies	179
Bibliography		180
Appendix A.....		195

LIST OF FIGURES

Figure 1-1 (a) AFC model (b) K3DY model [2].....	15
Figure 1-2 JSM high lift model [11].....	16
Figure 1-3 Components and actual photographs of the NASA HL-CRM at NASA Langley Research Center [23][28].....	18
Figure 2-1 Total fuel burn per kilogram and total fuel consumption rate at each altitude for takeoff, landing, and cruise [36].....	20
Figure 2-2 Leading and trailing edge device positioning parameters [37].	21
Figure 2-3 Breakdown of the noise components at takeoff and approach [41].....	26
Figure 2-4 Noise component of airframe noise at different condition [42].....	26
Figure 2-5 Computed surface flow pattern at $\alpha = 10^\circ$ [43].....	28
Figure 2-6 Measured surface flow pattern and comparison of aerodynamic forces [43].	28
Figure 2-7 Total pressure distributions at $\alpha = 10^\circ$ [43].....	28
Figure 2-8 Configuration of the full-span model, the original and modified brackets [44].	29
Figure 2-9 Comparison of static pressure distribution on the flap [44].....	30
Figure 2-10 Results of the farfield SPL directivity map [44].	30
Figure 2-11 (a) Sketch of a three-element HLD in landing configuration with the slat track system (b) Positions of typical cross sections [45].	31
Figure 2-12 Contours of instantaneous pressure fluctuations with dipoles along the trailing edge [45].....	31
Figure 2-13 Contours of instantaneous pressure fluctuations with dipoles along cusps [45].....	31
Figure 2-14 (a) Flow structure at the engine-wing junction area during pre-stall conditions from CFD results (b) Comparison of VHBR and UHBR engine attached at the same wing [46]. (c) Photograph of a nacelle chine on a Boeing aircraft	33
Figure 2-15 Configurations and aerodynamic results of different testing stages [8].....	35
Figure 2-16 InfraRed flow visualization results for the effect of nacelle chines at different angle of attacks [8].....	36

Figure 2-17 Predicted maximum lift coefficient from the EGO methodology and the selected points for experimental testing [12].	37
Figure 2-18 PIV results for the effect of nacelle chines at different views [12].	37
Figure 2-19 Analyzed geometry and position of the actuators for numerical simulation [46].	38
Figure 2-20 Results of the RANS computations of different cases with different blowing momentums [46].	39
Figure 2-21 Schematic of the wind tunnel model with actuators and fluidic AFC actuator [47].	39
Figure 2-22 (a) Aerodynamic results of different actuators with different blowing momentum (b) Tuft flow visualization for the baseline flow and actuated flow [47].	39
Figure 2-23 Configurations of two types of nacelle and their aerodynamic difference [16].	40
Figure 2-24 Surface flow pattern at post-stall conditions for both configurations [16].	41
Figure 2-25 C_p contours on inboard side of initial and final nacelle [48].	41
Figure 2-26 Color mapping of the normalized turbulent kinetic energy with the nacelle in undisturbed flow (without gust) and in disturbed flow (with gust) [49].	42
Figure 2-27 Spectra comparison of the noise landing gear emissions in the side direction between noise prediction models (Guo & Fink) and ALLEGRA wind tunnel experimental results [51].	43
Figure 2-28 Source localization maps for a Gulfstream G550 aircraft at gears up/down configuration at different frequencies [53].	44
Figure 2-29 (a) 2D PIV measurement of the velocity magnitude and direction in inline wheels (b) Zoomed in view of the vortex between the wheels [54].	45
Figure 2-30 (a) Actual figure of the high-fidelity landing gear (N2D) (b) Actual figure of the low-fidelity landing gear with simplified braces and lock links (N3C) (c) Sound pressure level measured from different landing gear models [56].	46
Figure 2-31 Velocity contours for both mean and RMS values for the complex configuration (a), simplified configuration (b) [57].	46
Figure 2-32 (a) Configurations of the testing models (a fully extended landing gear with a sealed cavity (Case 1)/partially exposed cavity (Case 2)/fully exposed cavity (Case 3)) (b) Results of the mean velocity magnitude through the same section that cuts through the strut and the fuselage centerline (c) Results of the turbulent kinetic energy through the same section as (b) [58].	47

Figure 2-33 Estimation of the contribution ratio of model components to the noise emission and noise sources of a test model [59]	48
Figure 2-34 Conceptual design of fairings and the effects of porous fairings on landing gears [59].	49
Figure 2-35 Effect of air curtain on landing gears [59].	50
Figure 2-36 Effect of plasma actuators on the cylinder [59].	50
Figure 2-37 Concept of the OEI situation [64].	52
Figure 2-38 (a) Lift coefficients of the horizontal tail for tail alone and tail affected by fuselage (b) for tail affected by the wing only and wing-fuselage [61].	53
Figure 2-39 CFD results of flow through a complete model and wing tail combination [61].	53
Figure 2-40 C_p on the horizontal tail [62].	53
Figure 2-41 Concept of the (a) body effect (b) wing effect on vertical tail [63].	55
Figure 2-42 (a) Effect of the body effect of the CFD model comparing to different tailplanes (the taper ratio of the CFD model is 0.55) (b) Effect of the wing at various aspect ratios and positions (A_v represents different taper ratio of the vertical tail) [63].	55
Figure 2-43 Testing geometry and numerical results compared with wind tunnel data of the sidewash at the vertical tail (IU: inboard, OU: outboard) [64].	56
Figure 2-44 Conceptual design of synthetic jet actuators [66].	57
Figure 2-45 Location of the SJA and the measurement domain of the SPIV [67].	57
Figure 2-46 (a) Overall effect of spanwise jet on vertical tail with different jet momentum injection (b) Effect of the spanwise jet location [67].	58
Figure 2-47 Results from the SPIV measurements [67].	58
Figure 2-48 Comparing the surface streak patterns colored by wall shear-stress magnitude [68].	59
Figure 2-49 Views of streamlines around root/inboard separation [68].	59
Figure 2-50 (a) Conceptual design of sweeping jet actuators (b) Effect of different numbers of sweeping jet actuators installed [69].	60
Figure 2-51 Effect of different AFC methods on side force [70].	61
Figure 2-52 Comparison of yawing moment due to sideslip of different bird wings [71].	62
Figure 2-53 Different visualization methods used in JAXA study [72][11].	64
Figure 2-54 Visualization of boundary layer transition on the nacelle model [73].	65

Figure 2-55 Smoke wire visualization of the flow (a) around the airfoil (b) within the separation bubble (top view) (c) within the separation bubble (side view) at $\alpha = 8^\circ$ [74].	66
Figure 2-56 (a) Original oil film interferometry results (b) Corrected oil film interferometry results [75].	67
Figure 2-57 Predictions of the velocity and turbulent stress for wing junction flow: solid line (WMLES simulation), dashed-dot line (FUN3D RANS), symbols (experiment) [77].	68
Figure 2-58 (a) Generic organization of a junction flow (b) Photograph of the NASA Junction Flow model (c) Photograph of the PIV system installed at the trailing edge of the JF model [79].	68
Figure 2-59 Infrared images of the fuselage and of the upper and lower wing surface [78].	69
Figure 2-60 Oil flow visualizations in the trailing edge corner [78].	69
Figure 2-61 Aeroacoustic results for the effect of slat cove fillers (SCF) and PIV flow visualization around slat region for the cases w/ and w/o slat cove fillers [81].	70
Figure 2-62 (a) Results from the FLEET signal scan (left: original velocity map, right: interpolated velocity map) (b) Measurement region [82].	71
Figure 2-63 PIV results (velocity vectors and magnitude contours) of airfoil in simulated flutter condition [83].	72
Figure 2-64 Aerodynamic results of airfoil in simulated flutter condition [83].	72
Figure 2-65 (a) Schematic of the BOS measurement setup (b) vertical displacement field of an airfoil at $\alpha = 4^\circ$ under steady freestream flow condition [84].	73
Figure 2-66 Displacement field around airfoil trailing edge under unsteady flow condition at $\alpha = 4^\circ$ [84].	73
Figure 2-67 Main features of the boundary layer on a low Reynolds number	74
Figure 2-68 Surface pressure distribution with pressure tapes at 60m/s and $\alpha = 12^\circ, 14^\circ, 16^\circ$ [11].	76
Figure 2-69 Surface pressure distribution on the wing using PSP at 60m/s and $\alpha = 10^\circ$ [11].	77
Figure 2-70 Surface flow visualization using (a) tuft (b) china clay (c) oil flow at 60m/s and $\alpha = 10^\circ$ [11].	78
Figure 2-71 Velocity distributions with PIV [11].	79
Figure 2-72 Results of the hot film signals (hf3) and R [4].	80
Figure 2-73 Infrared visualization results on an airfoil [4].	81

Figure 2-74 (a) Smoke visualization of smoke-wire at the upstream (b) Surface pressure distributions for $\alpha = 5^\circ$ (c) Separation, transition, reattachment, and roll-up locations (d) Side view visualization of vortex shedding (e) Sequence visualization of vortex shedding [86].	83
Figure 2-75 (a) Photograph of the LDV system in the junction model (b) Locations of the static pressure ports (red) and pressure transducers (blue) on the model [80].	85
Figure 2-76 Locations of LDV measurements at the leading edge (a) and trailing edge (b) [80].	85
Figure 2-77 Mean-velocity profiles along a line located at $z = 98.05\text{mm}$ near the wing leading edge [80].	85
Figure 2-78 Reynolds shear-stress profiles in the corner-flow region of the wing with leading-edge extension at $x = 2892.6\text{mm}$ and $\alpha = 5^\circ$ [80].	86
Figure 2-79 Results of the unsteady pressure measurement at the wing trailing edge region [80].	86
Figure 2-80 Boundary layer profiles of mean streamwise velocity obtained from CTA measurements at different chord stations [87].	87
Figure 2-81 Boundary layer profiles of fluctuation velocity obtained from CTA measurements at different chord stations [87].	87
Figure 2-82 Effect of Winglets with different dihedral [88].	88
Figure 2-83 Effect of wingtips on wingtip induced wakes [88][89].	89
Figure 2-84 Configurations of various wingtip devices [89][90][91].	90
Figure 2-85 Comparison of the velocity magnitude contour of the wingtip vortices and the surface pressure distribution of the wing surface [93].	91
Figure 2-86 (a) Comparison of peak velocity magnitude at vortex core (b) Comparison of static pressure at vortex core [93].	91
Figure 2-87 Comparison of the (a) axial velocity (b) C_p in the vortex core [94].	92
Figure 2-88 Comparison of vortex core location (a) from experiment (b) from SVV simulation (c) from previous LES model [94].	92
Figure 2-89 Pressure distributions in the wing-winglet concave region [95].	93
Figure 2-90 Mean steady roll rate at different angle of attacks [92].	94
Figure 2-91 Photograph of the test aircraft in the wind tunnel [98].	95
Figure 2-92 Effect of folding wingtips on aircraft stability in straight flight [98].	95

Figure 2-93 Effect of folding wingtips on aircraft stability during sideslip condition [98].	95
Figure 2-94 (a) Drag and WRBM increments of wing with large wingtip to the reference plan flight wing shape (b) comparison of the rigid and flexible assessment of tip fence and large wingtip [99].	96
Figure 2-95 (a) Lift performance between different flaps (b) Mechanism of multi-element flap [101].	98
Figure 2-96 OASPL and pressure fluctuations results on the lower surface [105]	101
Figure 2-97 Concept of wake bursting over a high-lift multi-element airfoil [107].	102
Figure 2-98 (a) Lift coefficient comparison of the designed model and the original model (b) Velocity contour of the designed model from CFD results [108].	102
Figure 2-99 (a) Velocity contours of the micro flap from CFD results (b) Lift coefficient dependence on angle of attack of various sealed micro flap deflections [109]	103
Figure 2-100 (a) Schematic sketch of airfoil with spoiler droop (b) Effect of airfoil with spoiler drop [110].	103
Figure 2-101 Schematic sketch of the plain flap design [103]	103
Figure 2-102 Comparison of ice accretion between numerical simulations (LEWICE3D) and experimental results [111].	105
Figure 2-103 (a) Ice roughness features (b) Effect of roughness on lift [113]	105
Figure 2-104 (a) Geometry of horn ice shape 2 (b) Comparison of horn ice shape from numerical simulations (LEWICE3D) and experimental results (IRT) [113].	106
Figure 2-105 (a) Velocity vector field downstream of the horn ice [112] (b) Effect of horn ice on lift [113].	107
Figure 2-106 Configurations of the subcategories of horn ice [113].	107
Figure 2-107 Difference of a separation bubble between 2D and 3D simulation horn-ice accretion [112].	108
Figure 2-108 Geometry of streamwise ice on different wing locations from IRT ice accretion and LEWICE simulations [113].	109
Figure 2-109 Effect of streamwise ice on aerodynamics (LEWICE: LS-SC5, LR-SC5) [113].	109
Figure 2-110 (a) Schematic drawing of spanwise ridge ice accretion (b) Flow streamlines over spanwise ridge [112]	110

Figure 2-111 (a) Classification of different spanwise ice shapes [113]. (b) Effect of spanwise ice shape on a NACA 23012 airfoil for $Re = 1.8 \times 10^6$ [112]. (c) Effect of spanwise ice shape on a GLC-305 swept airfoil for $Re = 1.8 \times 10^6$ [113].	110
Figure 2-112 Velocity magnitude around cleaned and iced airfoils [116].	111
Figure 2-113 (a) Configuration of the 2D smooth low fidelity ice shape (b) Photograph of the applied 2D smooth ice shape with grit roughness (c) Comparison of real ice shape and high-fidelity artificial ice shape (d) Effect of ice shape fidelity of maximum scallop on lift (e) Effect of ice shape fidelity of maximum scallop on drag (f) Effect of ice shape fidelity of maximum scallop on moment coefficient [117].	113
Figure 2-114 Comparison of the tuft visualization results and pressure distribution between the high-fidelity and low-fidelity cases [117].	114
Figure 2-115 Effect of Re on maximum lift coefficient on an airfoil with different leading edge roughness [103].	115
Figure 2-116 Final ice buildup on engine fan model with different spinners [118].	116
Figure 2-117 (a) Schematic of the SJA (b) Photograph of two SJA installed in the wing model (c) Experiment setup of the PIV measurements [123].	118
Figure 2-118 Effect of SJA arrays on airfoil [123].	119
Figure 2-119 (a) Time-averaged (b) Instantaneous velocity contours with and without SJA arrays [123].	119
Figure 2-120 (a) Schematic of the DBD plasma actuator (b) Effect of DBD actuators on the airfoil at different flap angles ($Re = 240k$) [124].	121
Figure 2-121 (a) Streamlines of the baseline and plasma control airfoil (b) Time-averaged normalized vorticity of the baseline and plasma control airfoil [124].	122
Figure 2-122 (a) Conceptual design of SWJ [69] (b) Setup of the experiment [125].	123
Figure 2-123 Effect of SWJ on lift ($Re = 750k$) and mean vorticity contours from PIV on the flap ($Re = 500k$) with $\delta_f = 40^\circ$ [125].	123
Figure 2-124 Concept of testing configurations [30].	124
Figure 2-125(a) Effect of AFC on simple hinged airfoil (b) Effect of AFC on simple hinged airfoil compared to conventional flap [30].	125
Figure 2-126 Total pressure field on airfoil w/ & w/o AFC actuation [126].	126
Figure 2-127 Effect & flow features of AFC w/ different operation frequencies [128].	127

Figure 2-128 Impacts of CFD from Boeing [133].	129
Figure 2-129 Experimental and numerical results of surface flow patterns [137].	130
Figure 2-130 Separation in the inboard wing fuselage region from experimental and numerical results [137].	131
Figure 2-131 Illustration of the relationship between hinge moment and control system [140].	132
Figure 2-132 Comparison between measured and calculated C_h (Datcom, Cobalt) [141].	132
Figure 2-133 Effect of (a) angle of attack & flap deformation (b) Mach number (c) flap gap on hinge moments [141].	134
Figure 2-134 (a) Photos of the CAHMMS system (b) Comparison of the results [139].	135
Figure 2-135 (a) Principle of using Moiré interferometry (b) Setup of the experiment [142].	136
Figure 2-136 Interferograms of the testing model [142].	136
Figure 2-137 Comparison of Moiré's result to force balance and strain gauge [142].	137
Figure 2-138 (a) Potential distribution of the electric field intensity and the effect of moment (b) Relation between the input moment and the output voltage [143].	137
Figure 2-139 Setup and the calibration results of the piezoelectric sensor [143].	138
Figure 2-140 NACA 3415 performance with simulated ice contamination [144]	139
Figure 2-141 Detector methods output for NACA 3415 at different configurations [144].	139
Figure 2-142 Depiction of the most relevant noise sources on civil aircrafts [149].	141
Figure 2-143 (a) Concept of the slat noise mechanism (b) Comparison of the computed instantaneous velocity field to PIV measurements [151].	142
Figure 2-144 Power spectral density of the surface pressure from CFD using different grids [152].	142
Figure 2-145 Effect of plasma actuators on noise reduction [153].	143
Figure 2-146 Effect of slat extension [154].	144
Figure 2-147 Effect of slat gap filler [154].	144
Figure 2-148 Effect of slat cove filler [154].	144
Figure 2-149 Overview of the flap side-edge noise source mechanism [155].	145
Figure 2-150 Comparison of the computed surface pressure distributions and velocity components with experimental data [155].	145
Figure 2-151 Comparison of the simulated SPL spectrum with wind tunnel measurement [155].	146

Figure 2-152 Effect of applying PRLE to flaps [156].	147
Figure 2-153 Effect of applying small barriers to flaps [157].	147
Figure 2-154 Comparison of isolated jet noise and jet-wing interaction noise [158].	148
Figure 2-155 Noise spectrum for engines placed at different locations [162].	149
Figure 2-156 Comparison of airframe noise from model and full-scale aircraft [163].	150
Figure 2-157 Different types of flaps and slats [164].	151
Figure 2-158 (a) Definition of the airfoil (b) Effect of slat gap on L/D and the far-field streamlines through slat slot (c) Effect of slat overhang on L/D and the Mach contours through slat slot (d) Effect of flap gap on L/D and the Mach contours through flap slot (e) Effect of flap overhang on L/D and the Mach contours through flap slot (f) L and L/D as a function of slat and flap deflection [165].	154
Figure 2-159 (a) Configurations and noise spectra for different slat configurations (b) Effect of slat overlap on noise (c) Effect of slat gap on noise (d) Effect of slat deflection on noise [166].	156
Figure 2-160 Effect of Re on slat gap (b) Effect of M on lift curves of configuration [167].	157
Figure 2-161 Side view measurement of slat wake [168].	157
Figure 2-162 Sketch of a trailing edge deflector and its effect [170].	158
Figure 2-163 (a) Effect of wake vortices on aircrafts (b) Different stages of wake [172].	159
Figure 2-164 Simulated vortex circulation vs. time [173].	160
Figure 2-165 Concept of the roll-up process and sources of various near field vortices [172].	161
Figure 2-166 (a) Concept of the vortex merging mechanism at low Reynolds number (b) Experimental images of vortex merging at high Reynolds number [172].	161
Figure 2-167 Numerical results of the secondary vortex situation and the circulations between the primary and secondary vortex [177].	162
Figure 2-168 Effect of the crosswind on wake vortices in ground from CFD simulations [177].	163
Figure 2-169 Mechanism of wake vortex in ground with obstacles and the effect of obstacle on ground wake [177].	163
Figure 2-170 Evolution of ground wake vortices encounter crosswind and obstacles [177].	163
Figure 2-171 (a) Evolution of wake vortex in HIT condition (b) Evolution of wake vortex in HIT condition with a smaller vortex separation (c) Wake vortex under different stratified	

atmosphere (d) Wake vortex in shear environment (e) Time evolutions of wake vortices at various conditions [178].	165
Figure 2-172 Concept of pulsed lidar [179].	166
Figure 2-173 (a) CFD and experiment results from Coder and Ortiz-Melendez [181] (b) CFD and experiment results from JAXA [134].	167
Figure 2-174 Concept of attachment line contamination [185].	169
Figure 2-175 (a) Profile of ACD (b) Effect of ACD at different locations on the leading edge [186].	170
Figure 2-176 Nature of laminar separation bubble on a turbine blade [187].	171
Figure 2-177 Dependence of the separation and reattachment location on different Reynolds number of a micro air vehicle [188].	172
Figure 2-178 Effect of reducing LSB by applying plasma actuators [189].	172
Figure 2-179 (a) Profile of discrete roughness elements (b) Effect of discrete roughness elements [191].	174
Figure 2-180 Effect of roughness diameter on transition location (Re_{TR} represents the transition Reynolds number, transition location x_tr/c can be obtained by scaling Re_{TR} by chord Reynolds number) [193].	174

LIST OF TABLES

Table 1 Slat and flap positioning for the HL-CRM configuration [38].	22
Table 2 Comparison among the sideforce coefficients from the CFD model and the semi-empirical methods [63].	55
Table 3 Summary of the best AFC method at different C_Y regions [70].	61
Table 4 Specific weights of different types of trailing edge flaps (unit: lb/ft^2) [101].	99
Table 5 Manufacturing cost for trailing edge flaps (for airplanes weight < 250,000~lb) [101]...	99
Table 6 AFC cartridges [30].	124
Table 7 Priorities of different issue of the HL-CRM at UW	177

ACKNOWLEDGEMENTS

First, I want to thanks Prof. Dana Dabiri and Prof. Mohammed R. Soltani for giving me a chance to work on this wonderful project. Being involved in this Boeing sponsored project to know about the design of commercial aircrafts at low speed in an aerodynamic prospective is the biggest award I've obtained during this journey in Seattle. I would also like to thank them about their useful comments, suggestions and guidance for my master thesis.

Next, I would like to thank Prof. Robert E. Breidenthal and Dr. Giovani Nino in the AA department for their guidance throughout my vortex generator side project and their invitations to marvelous events like the Northwest Test Pilot Symposium, Boeing Flight Test Tour, APS DFD Annual Meeting, and SAMPE Seattle Events. These experiences really broaden my horizon in the aerospace industry and they'll definitely be one of the best memories in my master's career.

Last, I would like to thank my parents for encouraging and supporting me emotionally and financially through these years to chase my dream. I would not have finish my degree without them.

Chapter 1. INTRODUCTION

1.1 HIGH LIFT COMMON RESEARCH MODAL

High Lift Common Research Model (HL-CRM) is a transonic transport type aircraft model created by the National Aeronautics and Space Administration (NASA). The momentum of building this model is because high quality experimental test results from relevant transport aircrafts are rare. Current research mostly falls on a relevant platform (wing or sections of high lift devices) or on a platform that does not represent commercial transportation. Actual aerodynamic data from commercial aircraft models are often held by their manufacturers. Therefore, developing a relevant high lift aircraft geometry to the public domain could serve as a platform for the benchmark of conventional high lift systems [1]. Hence, Boeing started the collaboration with the University of Washington (UW) to design, fabricate, and test a version of the HL-CRM (UW HL-CRM) in the Kirsten Wind Tunnel (KWT) at UW. The goal of this model is to serve as a platform for training, teaching, innovation, and research for undergraduate students, graduate students, and faculties from the Aero & Astro department and other UW Engineering disciplines for 10+ years. The design of the UW HL-CRM includes four phases: conceptual design, model design, fabrication, and testing. Phase 1 is conceptual model design considerations, which means to list the potential functions for our HL-CRM. Phase 2 is to provide actual model design and plans for KWT modifications. In Phase 3, the UW HL-CRM will be fabricated and KWT will be modified. In Phase 4, the UW HL-CRM will be tested and the data will be compared with other HL-CRM results. This paper listed the conceptual design considerations in Phase 1.

1.2 PREVIOUS WORK RELATED TO HL-CRM

1.2.1 *EUROLIFT Research Programme*

Aircraft manufacturers first brought out a program to study the high lift flows for transport aircrafts in the early 2000. The European high lift research project EUROLIFT included 11 European partners and was led by Airbus-Deutschland. The goal of EUROLIFT is to collect 3D configuration data at high Reynolds numbers for understanding the high lift flow phenomena, study the scale effects, and to provide accurate low speed and high lift performance prediction. Two wing models are used in this project: a 2D AFV model with slats and flaps, and a 3D KH3Y half model including the main wing, fuselage, slats, and removable flaps (Fig. 1-1). Various experiments including collecting data for CFD validation, measuring the transition positions and the separation area on the wing, testing a new high lift system, and deciding the optimum flap location have been done in the project. Ref. [2] provides an overview of the first EUROLIFT project, and ref. [3] demonstrates the comparison between numerical simulations and experimental results. Among all the experiments, the author could only find results of the transition measurements in ref. [4], complete data of the remaining experiments is not in public. This is probably because of security issues.

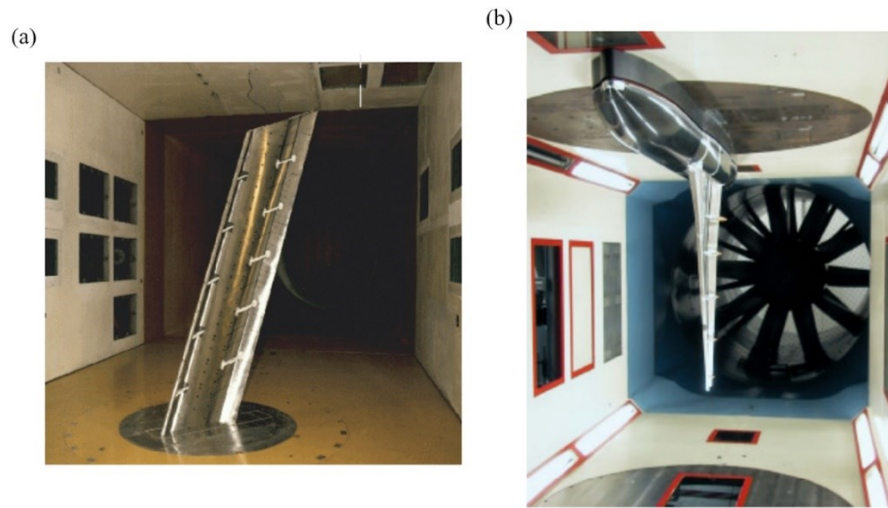


Figure 1-1 (a) AFC model (b) K3DY model [2].

The success in collecting high-quality data in EUROLIFT I leads to the operation of the following project. EUROLIFT II was brought out to study the effect of nacelle/pylon on wing separation and the application of nacelle strakes to delay separation. The KH3Y high lift model includes a leading-edge slat and a trailing edge Fowler flap. Configurations of the model with and without nacelle/pylons and nacelle strakes were tested in the wind tunnel, and the results were collected by flow visualization techniques. Ref. [5] provides an overview of the EUROLIFT II project. Ref. [6] studied the take-off performance of the model with different optimization strategies to decide the best position for the flap. Ref. [7] performs numerical studies of the geometric effects like wing deformation, and wind tunnel walls that might influence the experimental results. Ref. [8][9][10] discuss the effects of the installation of nacelle strakes.

1.2.2 *JAXA High Lift Program*

The Japan Aerospace Exploration Agency (JAXA) also developed a research program to study the aerodynamic performance of high lift configuration aircrafts. A high lift configuration aircraft model, the JAXA high-lift configuration Standard Model (JSM) represents a 100-

passengers modern class regional jet aircraft. JSM is also a half model aircraft without horizontal tails and landing gears, and it is equipped with slats, flaps, flap track fairings and a replaceable nacelle/pylon (Fig. 1-2). JAXA's high lift project focuses on using various flow visualization techniques to read the flow field, applying nacelle chines/strakes to delay separation, testing different nacelle configurations, collecting data (including acoustic data) for CFD validation, and studying geometric effects. Ref. [11] and [12] provide an overview of the goals and results of the JAXA project. Next, aerodynamic and aeroacoustic data of the baseline configuration are shown in ref. [13][14][15]. The effects of nacelle chine/strakes and different nacelle configurations on wing separations are investigated in ref. [16][17][18]. The mechanism of using nacelle chine for nacelle-wake separation delay is fully understood by main the aid of flow visualization techniques. In addition, the geometric effects like the spacer effect is studied in ref. [19], the wind tunnel interference effects is discussed in ref. [20] by CFD analysis. Lastly, CFD validation and numerical simulations are done in ref. [21][22].

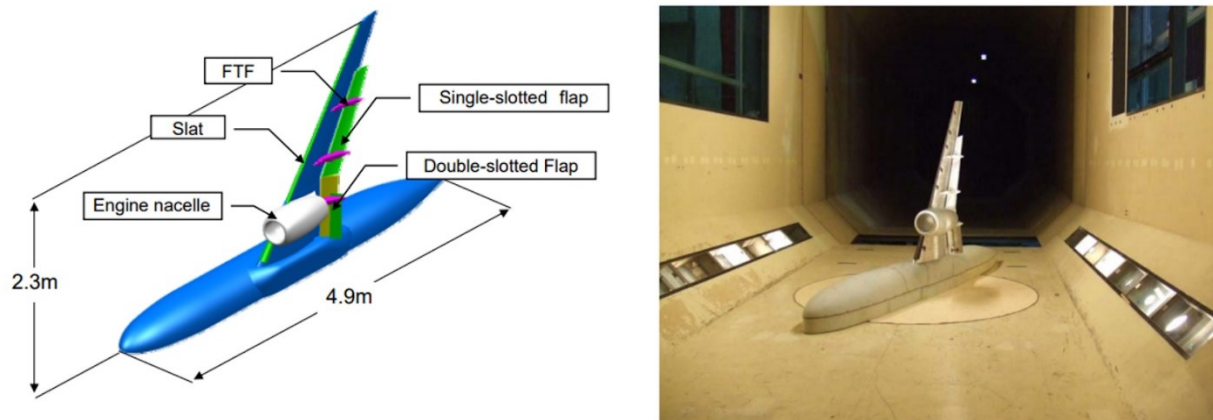


Figure 1-2 JSM high lift model [11].

1.2.3 NASA HL-CRM

The HL-CRM model developed by NASA is a half model with a fuselage, wing, a single leading edge slat, and two trailing edge flaps, as shown in Figure 1-3. The research interest in NASA HL-CRM project is to apply active flow control (AFC) techniques to simple hinge flaps to replace current flaps, to measure the noise source from the HL-CRM and try to reduce it by slat cove fillers or slat gap filler, and to collect uniform data for CFD validation. Ref. [1][23][24] talks about the goals of developing the HL-CRM and the details of the configuration. Ref. [25] gives a detailed description about the definition of the initial landing and takeoff configurations. Ref. [26][27][28][29][30] shows that initial results of applying AFCs on flaps: AFC devices could increase the flap aerodynamic performance, but the lift generated by a simple hinge flap with AFC is still not enough to replace double slotted flaps. Moreover, in ref. [31][32] NASA also studied the effect of nacelle chine on nacelles. Last, CFD simulations and validation studies are presented in ref. [33][34].

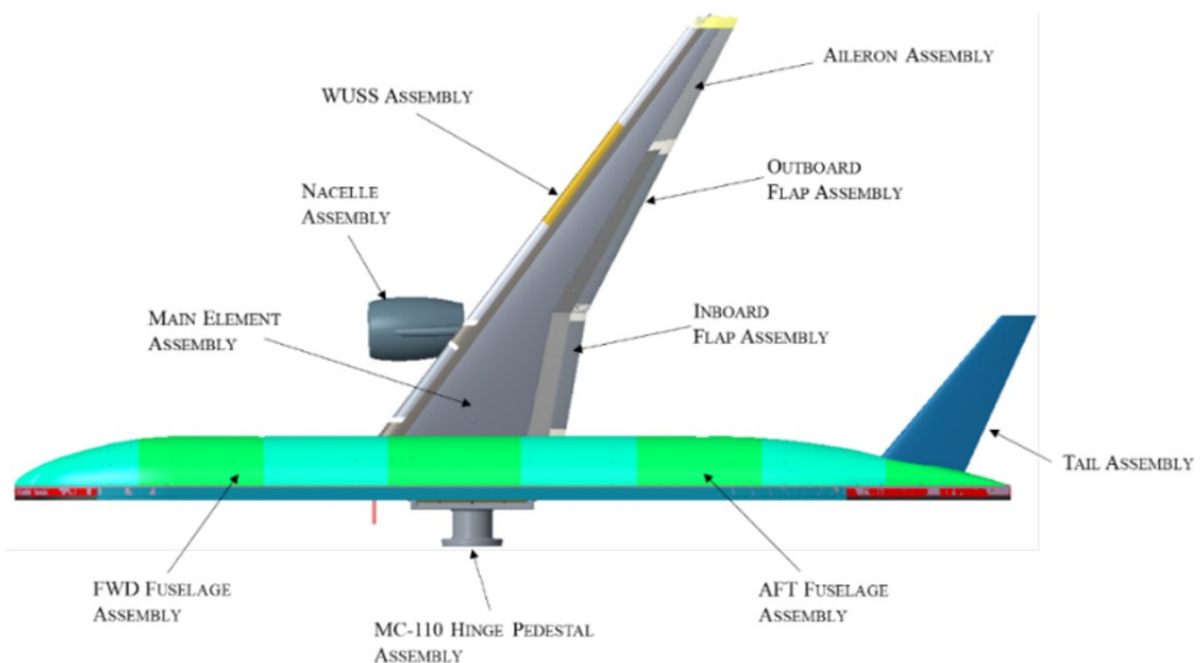




Figure 1-3 Components and actual photographs of the NASA HL-CRM at NASA Langley Research Center [23][28].

1.3 OBJECTIVES

The objective of this thesis is to make a list of all conceptual model design considerations to support future follow-on testing objectives that will influence the definition of model design and KWT facility modification requirements. In this phase, Boeing and UW listed 17 topics in aerodynamics that are related to the HL-CRM. In other words, the purpose of this research is to provide an investigation of current research interests for these 17 topics. Specific objectives include:

- Cruise, takeoff, and landing configurations
- Adjustable leading and trailing edge brackets
- Baseline configuration with defined leading-edge trims and nacelle/pylon, and full span leading edges without nacelle/pylon

- Nose and main landing gear
- Removable horizontal and vertical tails
- Flow visualization and off body flowfield studies
- Boundary layer study at various leading edge, main wing, and trailing edge locations
- Considerations for multi-element flaps
- Interchangeable pressure-instrumented wingtips
- Ice shape testing
- Active flow control (AFC)
- Low Reynolds number Computational Fluid Dynamics (CFD) validation
- Capabilities for hinge moment measurements
- Airframe noise testing
- Capabilities for leading and trailing edge positioning studies (deflections, gaps, heights, overlaps, etc.)
- Wakes studies, both steady state and transient/dynamic response
- Transition studies

Chapter 2. IMPORTANT ISSUES

2.1 CRUISE, TAKEOFF, AND LANDING CONFIGURATIONS

Short takeoff and landing configurations is one of the main targets for designing future aircrafts. In 2011, the European Commission published an agenda “Flightpath 2050 Europe’s Vision for Aviation” that listed the long-term industrial objectives for future aircrafts: 75% reduction in CO₂, 90% reduction in NO_x, and 65% in noise reduction compared to the aircraft in 2000 [35]. Fig. 2-1 shows the amount of fuel burn and fuel consumption for each stage. It is shown that although the highest fuel burned is during cruise than takeoff and landing, the highest fuel consumption per kilometer is during takeoff and landing [36]. Hence, the purpose of building the HL-CRM is to perform research on various areas to improve the performance of such an aircraft at low speed during landing and takeoff. Since the research topics listed by Boeing are all related to takeoff, landing and cruise, the following paragraphs provide an introduction of the contents of each topic, how they affect different stages of aircraft in flight, and how they fulfill the objectives of future aircrafts.

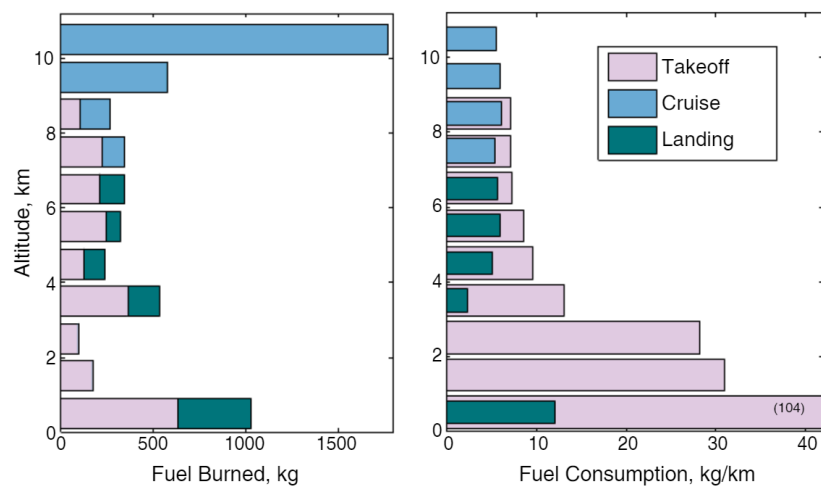


Figure 2-1 Total fuel burn per kilogram and total fuel consumption rate at each altitude for takeoff, landing, and cruise [36].

The configurations of aircraft takeoff and landing varies from the leading edge and trailing edge, aiming to provide maximum lift during takeoff and maximum drag during landing. The configurations of the airfoil during landing and takeoff for the HL-CRM were defined by Lacy and Clark [37]. The position variable of the leading edge and trailing edge on the airfoil is shown at Fig. 2-2. The slat and flap positioning are a deflection angle defined as a simple rotation started from a wing reference plane ($Z=0$). For the slat, the distance from the slat trailing edge upper surface to the wing reference plane is defined as the height h ; the gap g is the shortest 3D distance between the wing leading edge and the slat. On the other hand, for the flaps, the overlap o is defined as the X difference between the trailing edge and the flap, and the shortest 3D distance between the wing trailing edge and the flap is defined as the gap. All the parameters except the deflection will be represented in a dimensionless form by dividing the values by a reference chord (C_{ref}): the C_{ref} for the leading edge is the local trapezoidal chord without the yehudi, and the C_{ref} for the outboard flaps and inboard flaps are the local wing chord and a constant value equal to the wing chord at the wing trailing edge break. The initial rigging of the slats and flaps for the NASA HL-CRM is listed in Table 1.

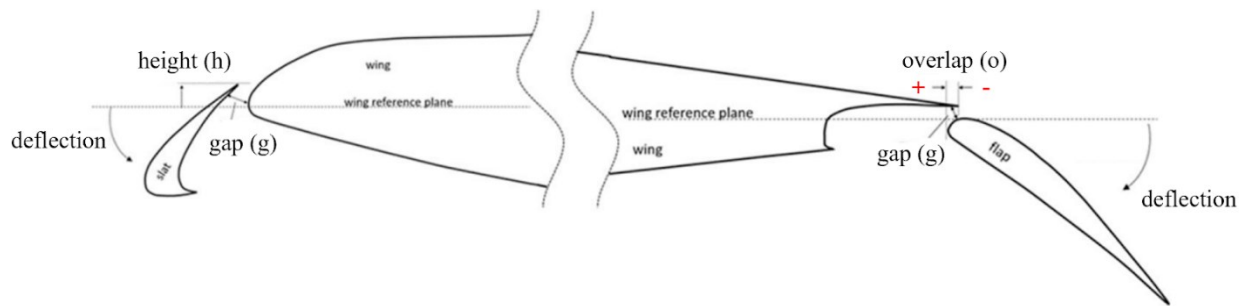


Figure 2-2 Leading and trailing edge device positioning parameters [37].

Configuration		Conventional (Baseline)		Simple Hinged Flap	
		Nominal	Range (Estimation)	Nominal	Range (Estimation)
Slat Deflection	Landing	30°	27° to 33° (3° increment)	30°	27° to 33° (3° increment)
	Takeoff	22°	19° to 25° (3° increment)	22°	19° to 25° (3° increment)
Slat Gap	Landing	1.0% C_{ref}	0 to 3% C_{ref}	1.0% C_{ref}	0 to 3% C_{ref}
	Takeoff	0.3% C_{ref}	0 to 3% C_{ref}	0.3% C_{ref}	0 to 3% C_{ref}
Slat Overhang	Landing	-0.8% C_{ref}	-1 to 2% C_{ref}	-0.8% C_{ref}	-1 to 2% C_{ref}
	Takeoff	1.5% C_{ref}	-1 to 2% C_{ref}	1.5% C_{ref}	-1 to 2% C_{ref}
Flap Deflection	Landing	37°	34° to 43° (3° increment)	50°	40° to 60° (10° increment)
	Takeoff	25°	10° and 25°	25°	10° and 25°
Flap Gap	Landing	1.25% C_{ref}	0 to 1.5% C_{ref}	N/A	N/A
	Takeoff	0.9% C_{ref}	0 to 1.5% C_{ref}	N/A	N/A
Flap Overhang	Landing	1.2% C_{ref}	1 to 4% C_{ref}	N/A	N/A
	Takeoff	3.3% C_{ref}	1 to 4% C_{ref}	N/A	N/A

Table 1 Slat and flap positioning for the HL-CRM configuration [38].

The research topics can be categorized into three main research areas, defined by Radespiel and Heinze [39] as the fundamentals of high lift for future commercial aircrafts. The first research area is “Flight Dynamics”. The goal of this research is to study flow structure based on the flight dynamics of commercial aircraft in order to increase lift capability at low-speed flight, and to provide a high-quality aerodynamic database for flow simulation.^s For the NASA HL-CRM project, numerical investigations and CFD validation tests using flow visualization were done to understand the transition and separation locations during landing and takeoff before applying flow control techniques. The purpose of doing these experiments is because the understanding of boundary layer is not enough, and the ability to predict transition and separation is insufficient. Hence, the project aims to build a uniform database for the public to improve current simulation accuracy, since previous research is either based on different aircraft models, or not open to the public. More details related to boundary layer study, transition, fluid visualization methods, and CFD on high lift are discussed in Chapter 2.7, 2.17, 2.06, 2.12.

Besides the topics included in the NASA HL-CRM project, other issues mentioned by Boeing that are related to flight dynamics are brackets (Chap. 2.2), multi-element flaps (Chap. 2.8), and winglets (Chap. 2.9). Brackets that support high-lift devices might cause flow separation on the wing, reducing the efficiency of the aircraft and providing additional noise source, but previous research does not focus a lot on brackets. Chapter 2.8 talks about the development from multi-element flaps to single-element flaps: although multi-elements flaps provide more lift during flight, the manufacturer and maintenance cost is way higher than single element flaps. Hence, aircraft manufacturers hope to use single element flaps to replace multielement flaps for future aircrafts. Chapter 2.9 mentioned the advantages and disadvantages of winglets and introduces a new type of wingtips, the folding wingtip, that has the potential to increase aerodynamic and aeroelastic performance during landing takeoff and cruise needed to be fully studied.

Aircraft wake plays an important role during takeoff and landing: wing wake may endanger the aircrafts following behind. Hence, the FAA have established regulations for the following aircraft to maintain a safety distance from a landing aircraft up ahead of them. With a strong growth rate in air traffic, these outdated rules will limit the capacity of airports. Current separation standards are based on largely empirical equations. Therefore, the research interest in wake studies is to build a predict system that can predict when and how long the corridor is free of vortices is needed and to try and reduce the wake from various parts of the aircraft in order to reduce drag and improve the performance. Aircraft flying with icing hazards is very dangerous, an icing wing will reduce the maximum lift, increase drag, and stall at a lower angle of attack. Current understanding in the shapes and mechanism of ice accretion are based on 2D airfoils. The understanding of 3D icing is rare, because previous studies focused on the overall effect only, neglecting the research in the flow-field. Chapter 2.10 presents current achievements of different

ice shapes on aircrafts, latest research interest in ice shapes, and current methods to prevent icing. Last, Chapter 2.13 intimate the purpose to study hinge moment: the hinge moment for aircraft design uses an estimated value since hinge moment cannot be measured for small aircraft models. The estimated hinge moment is not accurate, it will lead to an overestimated control surface actuator, causing additional weight for the aircraft during takeoff, landing, and cruise.

The second research area is “Efficient High Lift”. The goal of this research is to provide more lift during landing and takeoff [39]. Applying AFC on aircrafts provides a potential to increase pitching moment, delay boundary layer separation, and improve aerodynamic performance by changing the flow field using pulsating jets [40]. The interesting locations of installing AFC devices on the aircraft are the trailing edge of airfoils, nacelles, and the vertical tail. Since the current high-lift system includes a complex system of levers, linkages, and gears, which increases the weight. Aircraft manufacturers hope to use a simplified high lift system with AFC that could reduce the complexity of the mechanical systems, lower weight, reduced actuation power, and lower maintenance costs for future aircrafts. However, the latest results from NASA shows that replacing conventional flaps by simple hinge flaps with AFC is still unavailable (Chap. 2.11).

The application of AFCs on nacelles is to delay nacelle-wake separation. Nacelle-wake from current VHBR (very high bypass ratio) engines have been successfully delayed and understood by nacelle strakes. Yet an UHBR (ultra-high bypass ratio) engine for future aircrafts will generate more nacelle-wakes, and the previous method that uses nacelle strakes does not have the ability to suppress the separation. Hence, the potential of using AFCs to delay the separation of nacelle-wakes have been studied, and the mechanism is still unclear. More details of nacelle wakes are discussed in Chapter 2.3. The last application of AFCs is on vertical tails. Current design

for vertical tails is to provide enough yawing moment during an OEI (one-engine inoperative) condition, which means that the vertical tail is oversized during normal operation, increasing the drag, and weight in flight. NASA studies the effect of applying AFCs on vertical tails to provide additional yawing moments, and the results are positive. However, the research is still in the early stage since other effects such as the interference effects from wing/fuselage and operating at contamination environments have not yet been studied. Current progress of using AFC to reduce the size of the vertical tail has been mentioned in Chapter 2.5.

The last research area is: Fundamentals of Aeroacoustics, which develops an integrated aeroacoustic and aerodynamic design methodology [39]. The research area aims at reductions of airframe noise sources by investigating fundamental acoustic and aerodynamic effects, collecting high-quality experimental data for CFD validation, and studying the potential of noise source reducing methods on airframe and noise absorbing surfaces. The reason to reduce airframe noise is because with the invention of bypass turbofan engines, noise orienting around the airframe during takeoff and landing is more important than engines, as shown in Fig. 2-3 [41]. Major airframe noise for current civil aircrafts can be characterized into two groups: high-lift systems (slats and flaps) and landing gears, as shown in Fig. 2-4 [42]. Noise from high-lift systems originates from flow flows through slat and flaps. These flows are hard to predict due to their complexity, for instance, various phenomena like vortex shedding, edge scattering, secondary vortex happens inside the slat cave. For landing gears, the prediction of landing gears is still inaccurate due to the negligence of details such as struts, joints, and dressing in previous research. Additionally, most noise reduction technologies are still in the laboratory. Hence, the purpose of studying high-lift landing gear noise is to develop a methodology to predict acoustic accurately and to perform better designs that generate lesser noise.

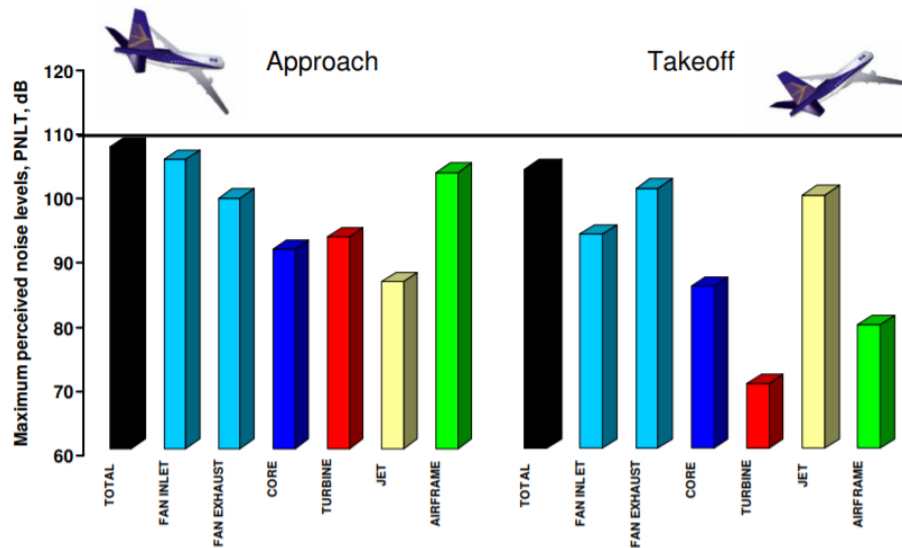


Figure 2-3 Breakdown of the noise components at takeoff and approach [41].

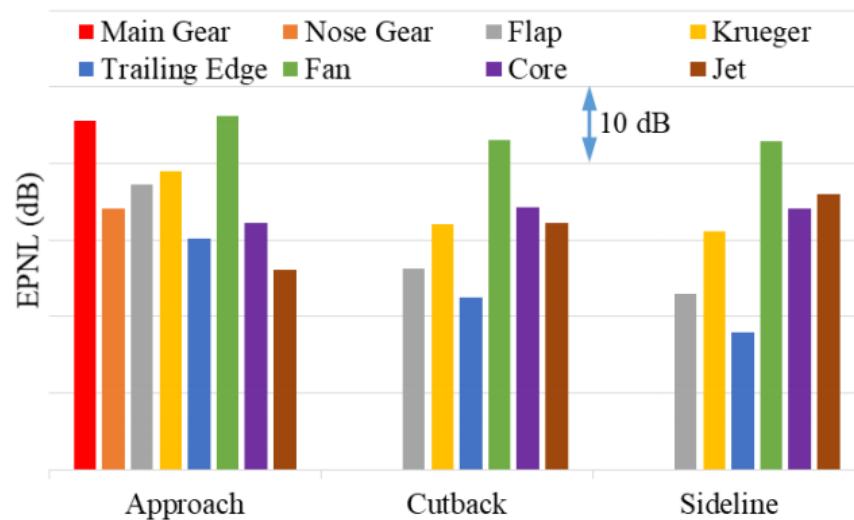


Figure 2-4 Noise component of airframe noise at different condition [42].

2.2 ADJUSTABLE LEADING AND TRAILING EDGE BRACKETS

Leading edge slats and trailing edge flaps are common high lift devices on the aircraft wing that increases the amount of lift produced by the wings. Supporting devices such as slat tracks, flap brackets, and flap track fairings (FTF) are used to guide the movement of panels and to support high lift devices. Despite the fact that the installation of high lift devices could provide more lift

to the aircraft, these supporting brackets might cause flow separation on the wing, thus reducing the efficiency of the aircraft, and providing a contribution to the noise source.

In JAXA's high lift program, the effect of supporting brackets was considered and three numerical models were built: one clear model without FTF and supporting brackets, a half model without FTF but included supporting brackets, and a full model with FTF and supporting brackets to study the overall effect of supporting brackets. Computational results of surface flow patterns, total pressure distributions, and aerodynamic forces are analyzed. Computed surface flow patterns at $\alpha = 10^\circ$ are shown in Fig. 2-5. First, Fig. 2-5(a) shows the flow separation near the flap trailing-edge. By comparing Fig. 2-5(a) and Fig. 2-5(b), the model with brackets shows that the wake induced by the brackets increases the separation on the flap trailing edge and influences the flow on the main wing. For the model with FTF and bracket supports, the local flow separation is more severe on the flap trailing edge due to the presence of FTF. Fig. 2-6 shows the experimental and computational results of the aerodynamics forces and an experimental surface flow pattern results of a full model including FTF and supporting brackets to compare with the numerical case. The lift coefficient of the clear model and half model didn't not show large changes at the moderate angle of attacks. The full model, however, shows larger changes in the lift coefficient, it is probably caused by the large separation on the outboard flap. Moreover, by comparing the surface flow pattern from the experimental and numerical results, flow separation near the wingtip on the main wing was observed in both figures, showing a good agreement between the experimental and computational results. In Fig. 2-7, the pressure distribution shows that the main pressure loss comes from the nacelle pylon, and the disturbed wakes from the FTFs and brackets affect the flow separation [43].

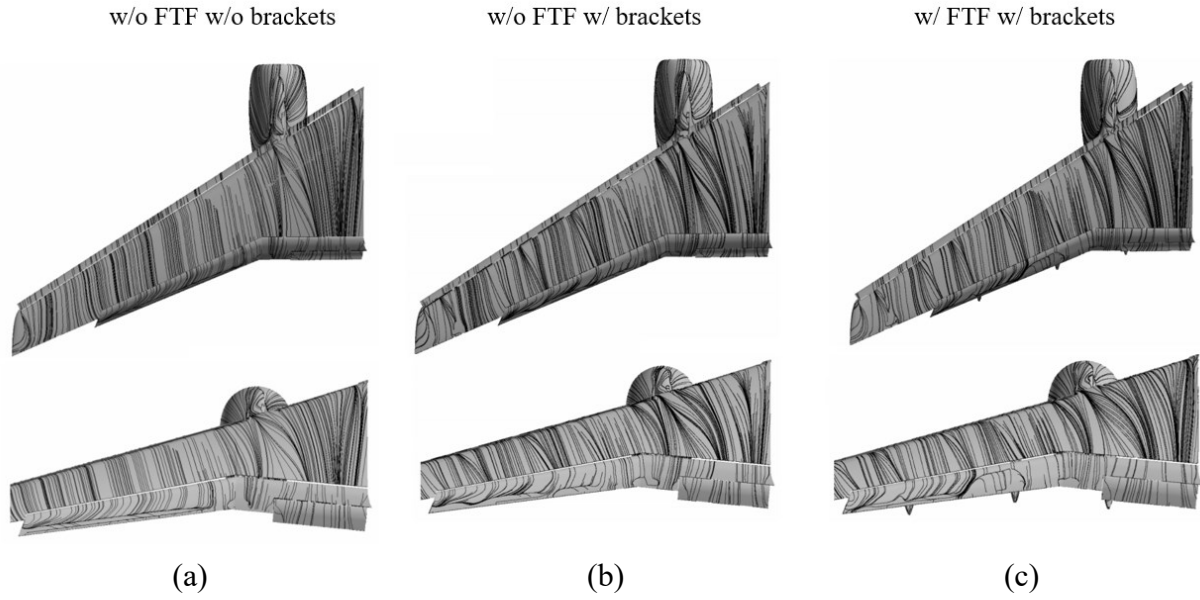


Figure 2-5 Computed surface flow pattern at $\alpha = 10^\circ$ [43].



Figure 2-6 Measured surface flow pattern and comparison of aerodynamic forces [43].

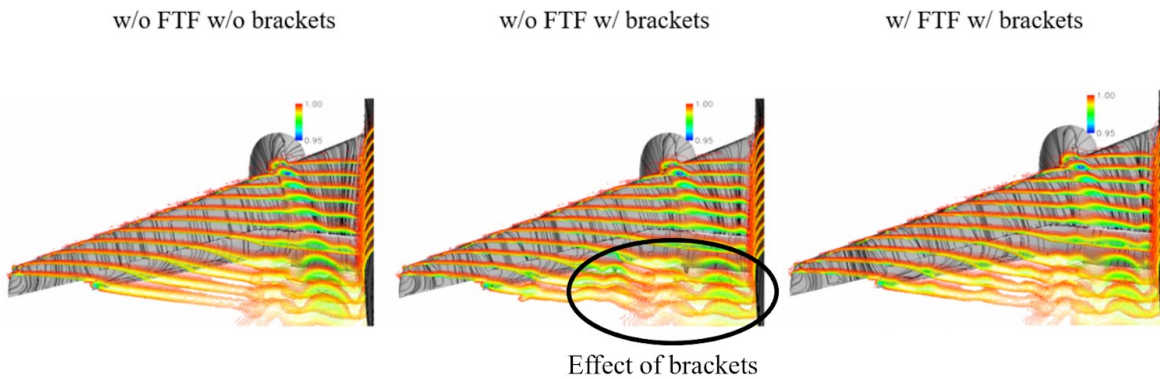


Figure 2-7 Total pressure distributions at $\alpha = 10^\circ$ [43].

Aeroacoustic is also an effect caused by the high-lift supporting system. Fares et al. [44], provided a simulation of a full span Gulfstream business jet model using PowerFLOW to capture the flow physics and acoustics in the near field. Three configurations were investigated: a full-scale model with high fidelity brackets, a modified model with simplified brackets, and a model with no brackets. Fig. 2-8 shows the configuration of the full-scale model, and the bracket difference between both models. In Fig. 2-9, the static pressure distribution on the flaps shows that the two bracket cases produce nearly identical patterns of wake separation in the vicinity of the flap brackets. Flow separation still occurs even if the brackets are removed, this is probably due to the retainment of bracket tracks. Although previous results did not reveal a big difference in the modified brackets, the following farfield SPL directivity maps in Fig. 2-10 shows interesting results of the influence of the holes and gaps on brackets. Small openings on the brackets generate artificial high frequency noise, by covering these holes and gaps, the sound pressure level on the outside diameter of the map is reduced. Removing the brackets completely identified that the brackets are a prominent secondary noise source to high frequency noise.

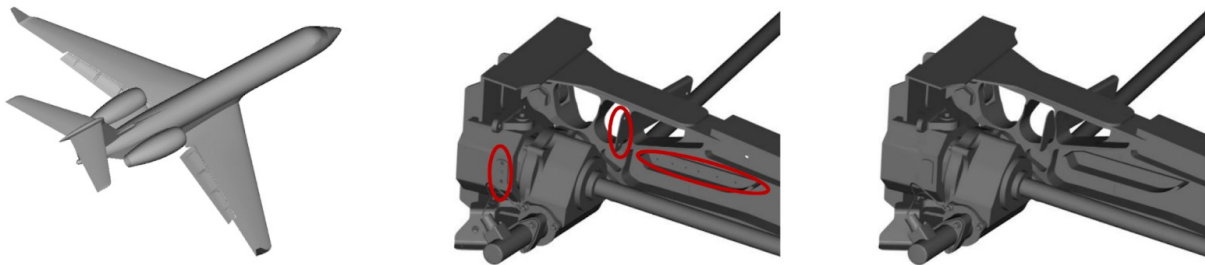


Figure 2-8 Configuration of the full-span model, the original and modified brackets [44].

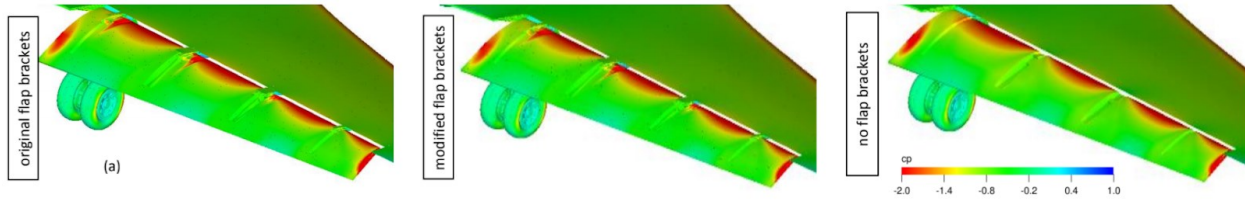


Figure 2-9 Comparison of static pressure distribution on the flap [44].

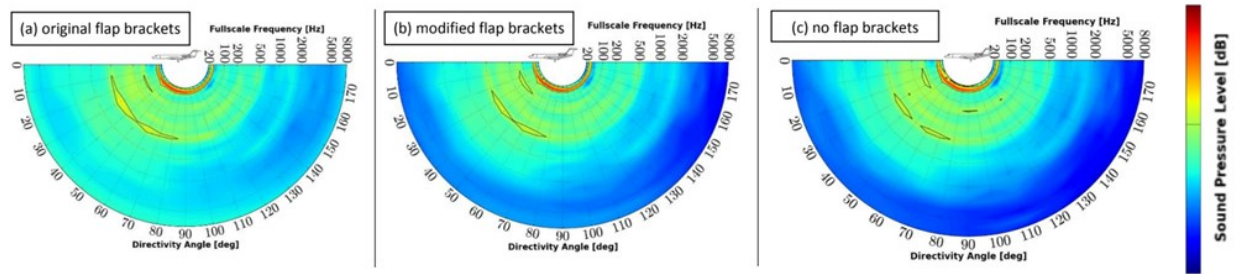


Figure 2-10 Results of the farfield SPL directivity map [44].

In order to investigate the impact of bracket noise in more detail, Wang, et al. [45], studied the mechanism of the slat track system on the slat noise propagation. 3D CFD simulation of a baseline system without slat tracks, and one with a slat track system is built. Configurations of the CFD model and the position of the measurement cross section is shown in Fig. 2-11. Dipole sources are placed on the slat trailing edge and on the slat cusp to study the interaction between the fluid and solid surfaces. In Fig. 2-12, when line dipole sources are placed along the slat trailing edge, the sound waves are mainly propagating upwards. The influence of the slat track system is shown in the radiation at the direction towards the ground, even for the position (S3) which is far away from the slat track system. For the case of the sound waves are found mainly propagating towards the ground (Fig. 2-13), diffraction is observed when the sound waves pass by the opening and propagate upwards, and less cancellations are observed due to more overlapping of the radiated sound waves and the reflected waves.

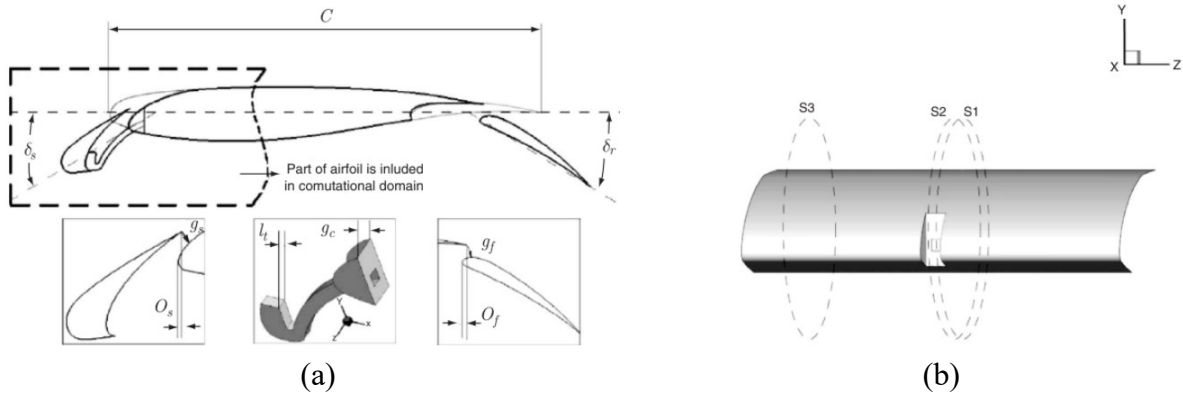


Figure 2-11 (a) Sketch of a three-element HLD in landing configuration with the slat track system (b) Positions of typical cross sections [45].

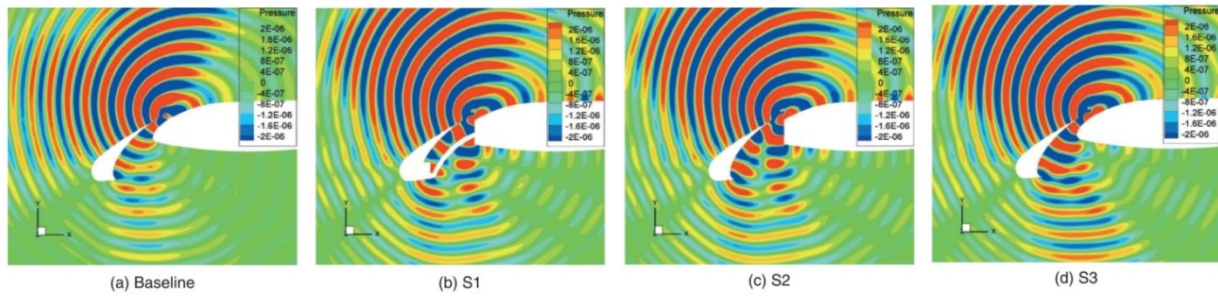


Figure 2-12 Contours of instantaneous pressure fluctuations with dipoles along the trailing edge [45].

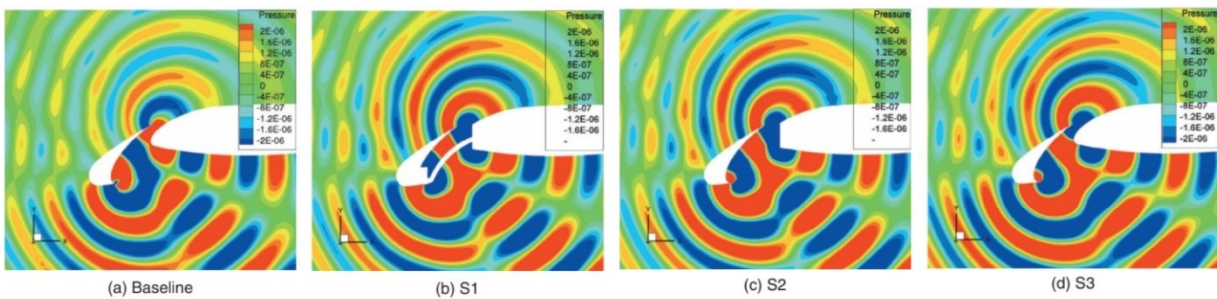


Figure 2-13 Contours of instantaneous pressure fluctuations with dipoles along cusps [45].

In general, research in supporting brackets studies the overall effects of leading edge and trailing edge supporting devices such as slat track, flap brackets, and FTFs on aeroacoustics and aerodynamic forces. However, the mechanism that causes the effect, like the studying flow

structure, the location of the brackets and the interaction effects around the supporting devices are still unknown. The lack of interest in this area is probably because of the minimal influence on lift by brackets. For instance, in Fig. 2-6, simulation results show that the lift coefficient from the model with brackets are nearly the same as the one without brackets. Moreover, most studies focus on numerical simulations and only verify the results by using experimental data from models with brackets only, this is due to the difficulty in obtaining data from an actual non-bracket model. Hence, if more experimental data like aerodynamic forces, surface pressure, flow structure of a non-bracket model can be obtained, more problems might be pulled out and investigated, then the understanding of bracket effect can definitely take a big step forward.

2.3 CONFIGURATIONS WITH AND WITHOUT NACELLE/PYLON

The installation of nacelle and pylon on a classical backward swept wing transport aircraft may lead to separation on the wing surface during landing and takeoff, since the wing should have sufficient lift during low speeds and high angle of attacks. Experimental and numerical studies showed that these separations are mainly caused by the wing region without mounted leading edge devices. As shown in Fig. 2-14(a), the lack of high lift devices in the slat cutout region generates vortices in the downstream that lead to separation. The effect of nacelle-wake separation induced by current VHBR (very high bypass ratio) engines have been fully studied in the past, and this separation can be suppressed by applying a passive flow control device named as nacelle strakes or nacelle chine on the nacelle. Although nacelle strakes have already been applied on current commercial aircrafts, the study in this phenomenon has not stopped. This is because of the high potential of installing UHBR (Ultra High Bypass ratio) engines on future aircrafts. UHBR engines are high efficient engines that use large bypass ratios to reduce fuel burn. These engines have larger diameter and need closer coupling to the wing to satisfy ground clearance requirements and

to avoid a redesign of the landing gear. The installation of UHBR engines needs a larger slat cutout, which leads to an enhancement in nacelle-wake separation. The difference between the size and slat cutout of a VHBR engine and a UHBR engine on the same wing is shown at Fig. 2-14(b). Previous method that uses nacelles strakes does not have the ability to sufficiently suppress the separation caused by UHBR nacelle-wing junctions. Hence, the study continues by applying active flow control techniques on nacelles during landing or takeoff [46].

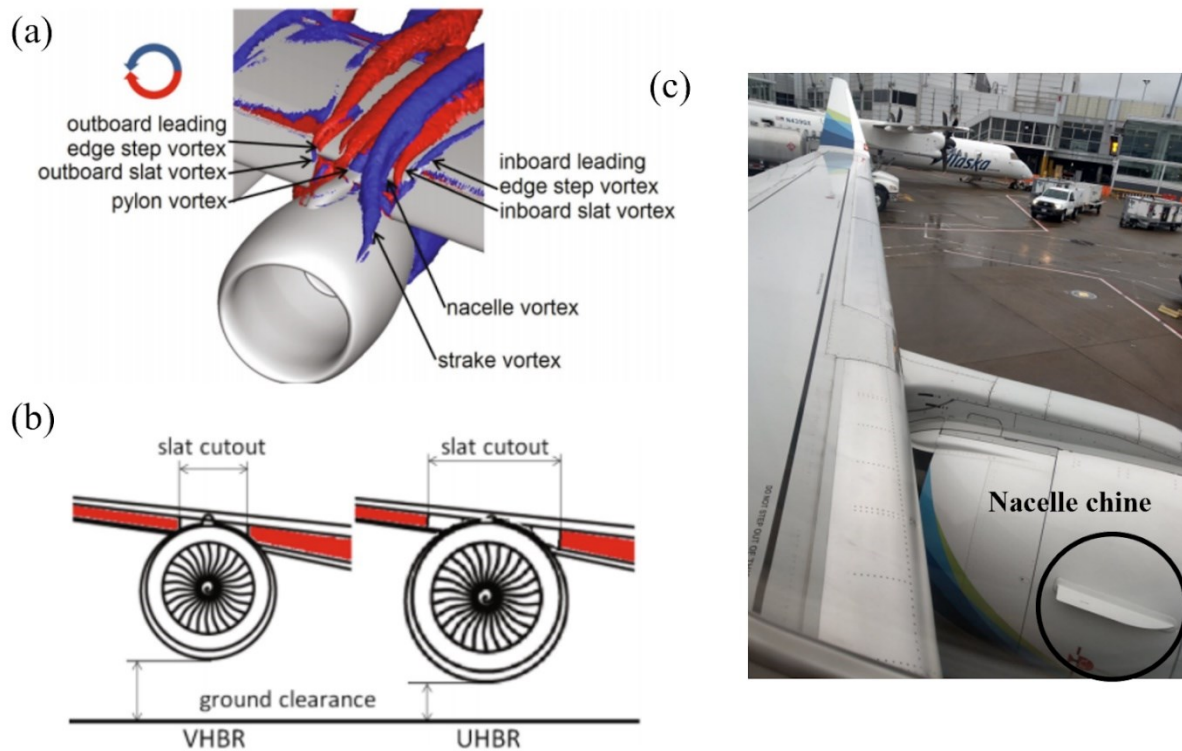


Figure 2-14 (a) Flow structure at the engine-wing junction area during pre-stall conditions from CFD results (b) Comparison of VHBR and UHBR engine attached at the same wing [46]. (c) Photograph of a nacelle chine on a Boeing aircraft

The European High-Lift Project EUROLIFT II first studied the nacelle-wake separation and applied a nacelle chine (Fig. 2-14(c)) on a high lift model and analyzed its effect. A complex high lift half-model with three stages was tested in the wind tunnel. The first stage which is the baseline configuration included an inboard span wise gap between the slat/flap and fuselage, a slat

horn and an onklet. Stage 2 added a nacelle, a pylon, and a slat cut out at the engine position. The last stage installed a nacelle chine on the nacelle outer surface. The configurations and the aerodynamic results of all three stages are shown in Fig. 2-15. Lift curves in stage 2 and 3 are lower than the baseline configuration, this is due to the additional drag of the nacelle. In addition, the difference between the installation of nacelle chines is caused by the separation induced by the nacelle. The details of the separation process are shown from the infrared flow visualization, shown in Fig. 2-16. The angle of attack A, B, C, and D represents the AOA before stall, near stall, stall, and after stall. Infrared images for both configurations at angle of attack A are similar. Then separation starts to happen at the slat near the nacelle region for angle of attack B at stage 2. At angle of attack C, separation continues for stage 2, and the flow remains attached for the other stage. From angle of attack C to D, the flow starts to separate, the case with nacelle chine eventually switches to separated flow, performing similar images as the case without nacelle chine [8].

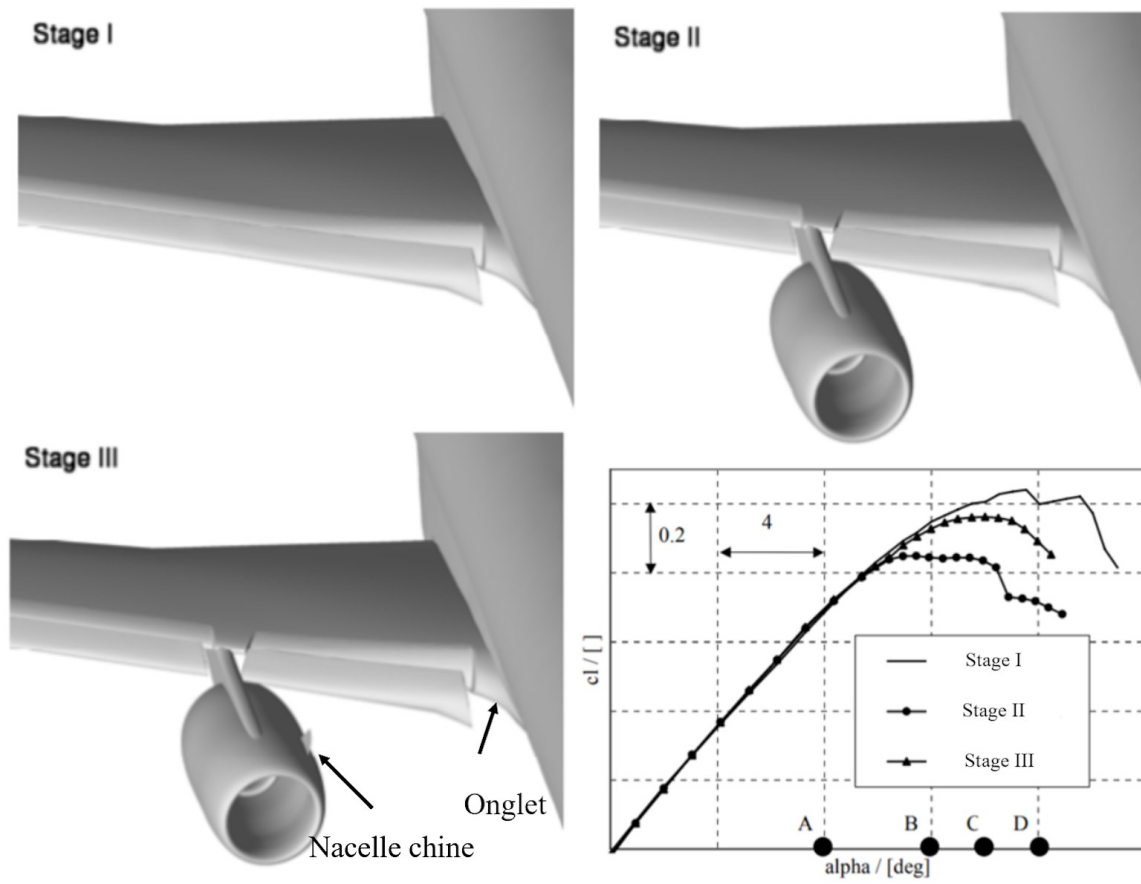


Figure 2-15 Configurations and aerodynamic results of different testing stages [8].

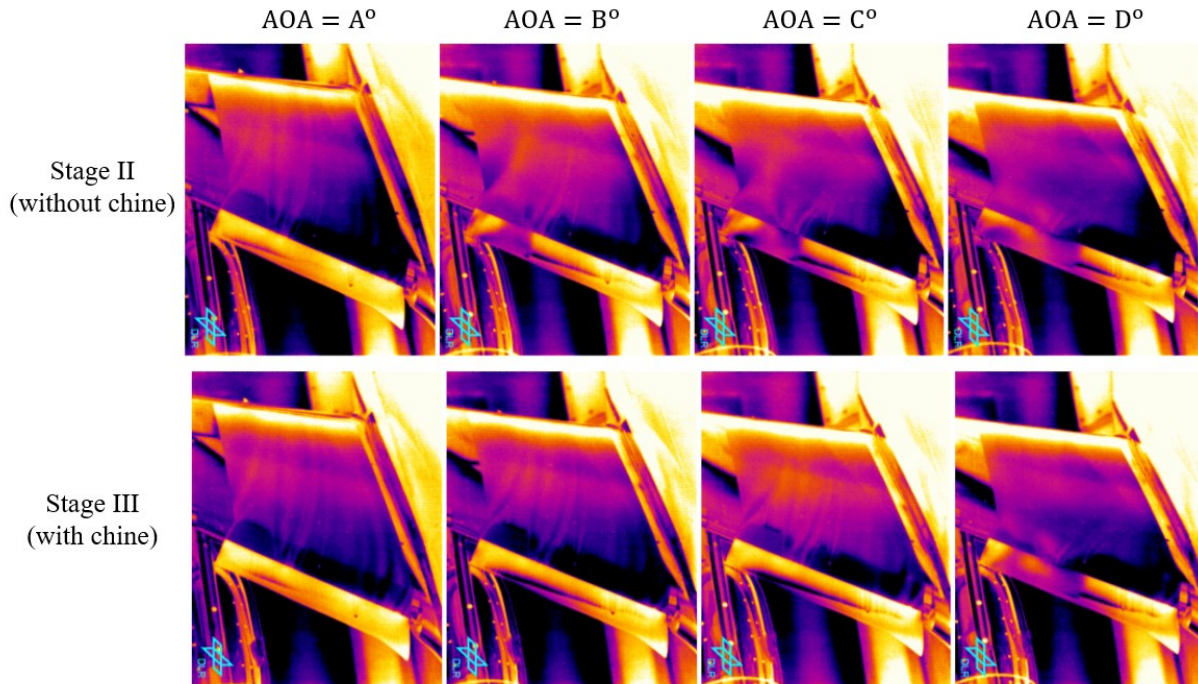


Figure 2-16 InfraRed flow visualization results for the effect of nacelle chines at different angle of attacks [8].

The mechanism of nacelle chine was further studied by JAXA. Nacelle chine was installed at different positions on the nacelle of a high-lift configuration aircraft model JSM. The position of the chine was initially determined by an exploration methodology (Efficient Global Optimization with Genetic Algorithm), the results are shown in Fig. 2-17. Three points (point_F, point_A, point_B) are then selected for further investigation of the flow field using PIV. Fig. 2-18 shows the PIV results from the inboard area and from upstream for the best and baseline case. In Fig. 2-18(a), observation from the baseline configuration (no chine case) shows a loss in velocity magnitude in the downstream region caused by the separation of a longitudinal vortex after the slat cutoff region. This mechanism leads to a reduction of lift performance at high AOA conditions. When the chine is installed, the vortex generated by the chine seems to reduce the vorticity of the pylon-vortex and lower the low speed region close to the surface. In Fig. 2-18(b), the pylon-vortex is visible above the pylon and it induces an upwash flow near the edge of the inner slat that moves

the low-speed flow region upwards and eventually merges with the vortex core when the chine is not installed. When the chine is installed, the pylon-vortex disappears, and the low-speed flow region remains in the same position because there is no pylon-vortex induced upwash flow [12].

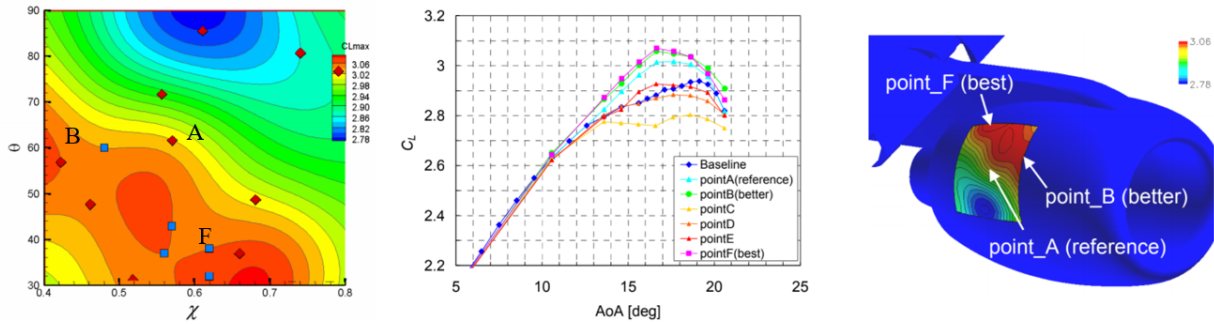


Figure 2-17 Predicted maximum lift coefficient from the EGO methodology and the selected points for experimental testing [12].

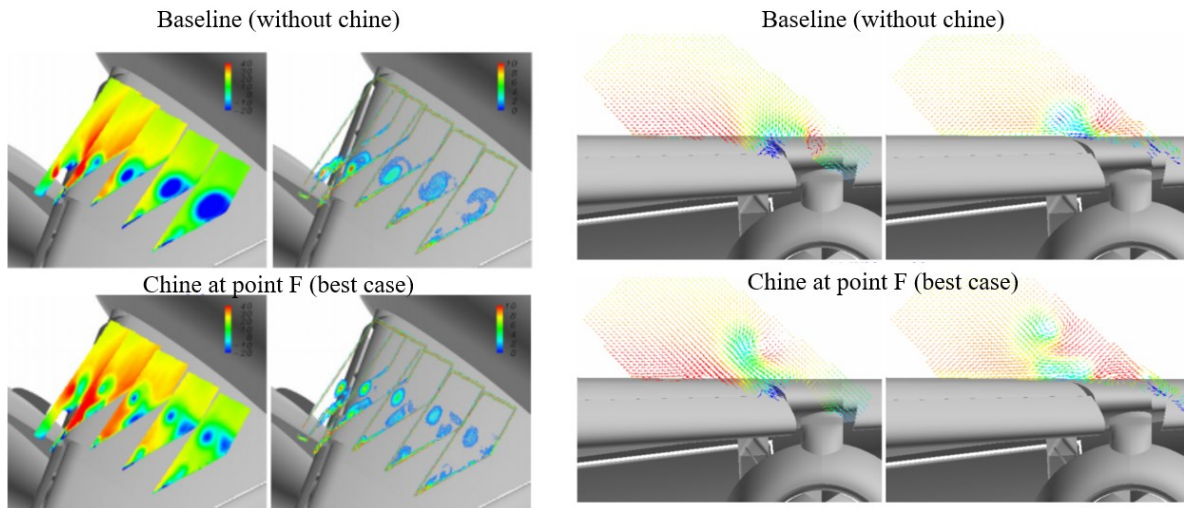


Figure 2-18 PIV results for the effect of nacelle chins at different views [12].

On the other hand, a numerical study using RANS computations was brought out first by a recent European project AFLoNext (Active Flow, Loads and Noise Control on Next Generation Wing). The purpose is to study the potential of applying active flow control techniques on an engine-wing junction model (shown in Fig. 19) during landing configuration, aiming to suppress the separation caused by the nacelle wake. Fig. 20 demonstrates the CFD results of different cases

with different blowing momentums and angle of attacks. It can be clearly seen that the size of the separation area downstream of the nacelle is reduced with an increasing C_{μ} at the same AOA. Moreover, steady blowing from the actuators also forced the airflow at the outboard region towards the inboard side.¹ Due to the limitations of the numerical model, CFD investigations focused on the flow topologies and influence of the flow control on the separated areas only. A real-scale experiment of a wing model that represents the segment of the pylon-wing region of a large transport aircraft with fluidic actuators are carried out, shown in Fig. 21. Fig. 22 then shows the aerodynamic and tuft flow visualization results from the experiment. The lift curves indicated that the application of AFC successfully extends the linear range of the lift coefficient. It is a pity that the wind tunnel model did not reach full stall at the maximum possible angle of attack. The reason for the above difference is related to the local flow separation behind the slat cutout. This can be observed from tuft flow visualization results: significant separation spreads across the area downstream of the slat side edge for the baseline; when AFC is activated, despite that a small area of separation still remains, but most of the separation downstream of the AFC location is eliminated [46] [47].

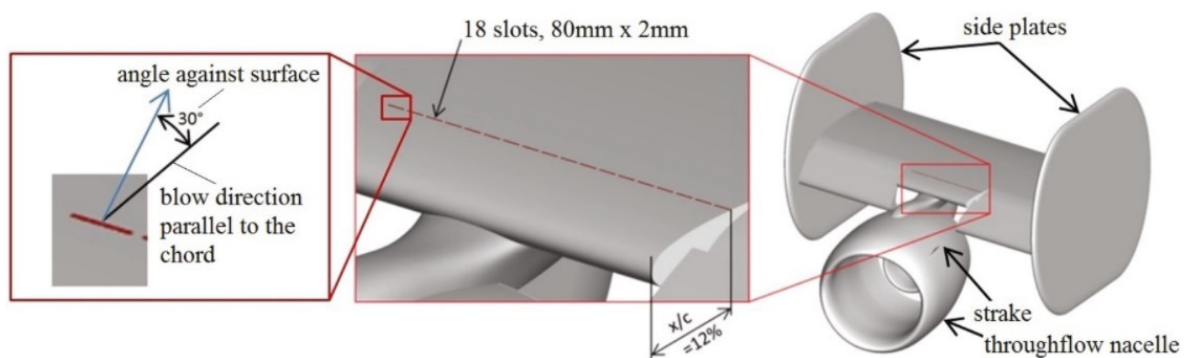


Figure 2-19 Analyzed geometry and position of the actuators for numerical simulation [46].

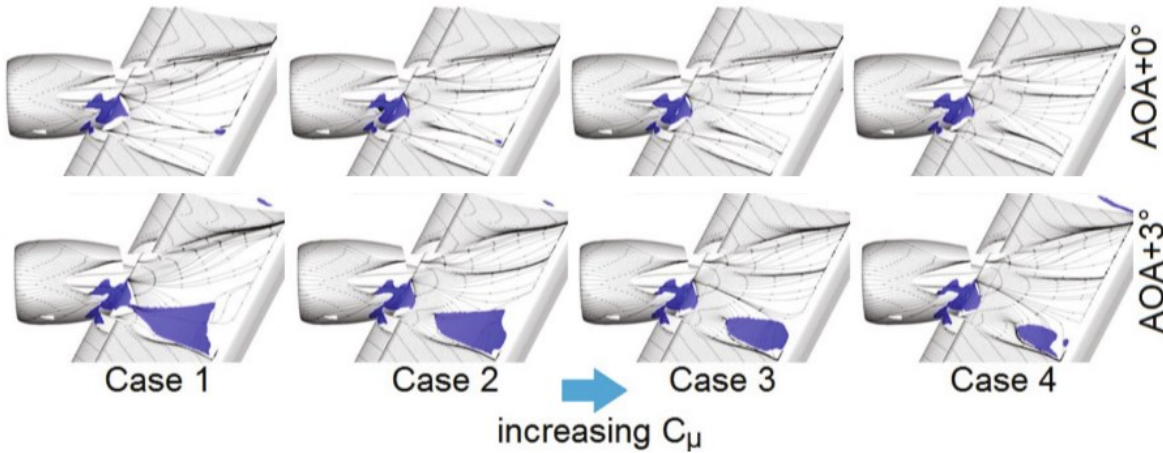


Figure 2-20 Results of the RANS computations of different cases with different blowing momentums [46].

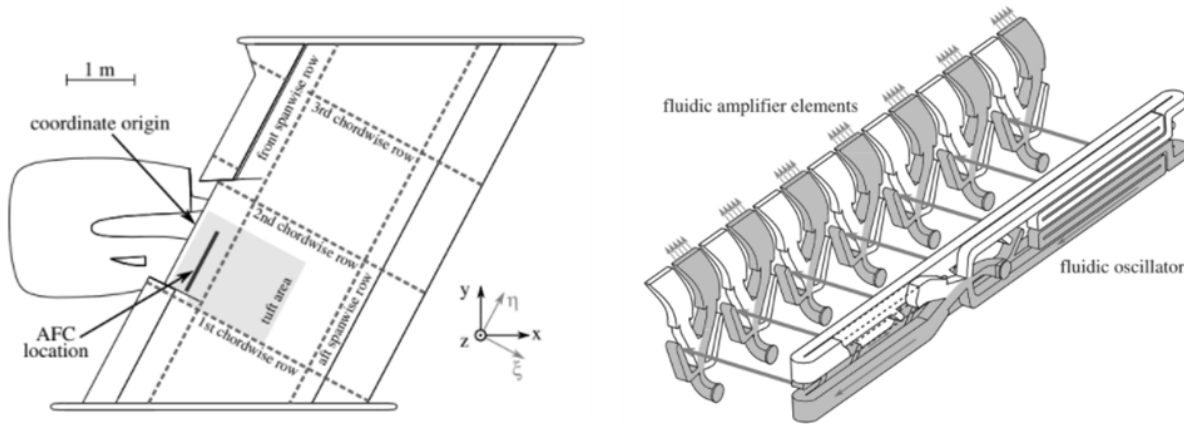


Figure 2-21 Schematic of the wind tunnel model with actuators and fluidic AFC actuator [47].

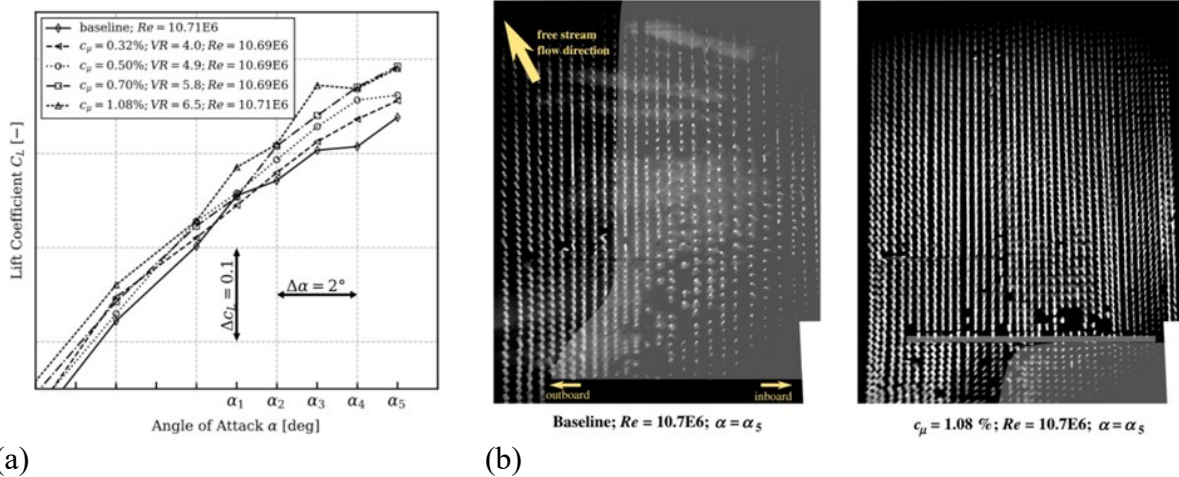


Figure 2-22 (a) Aerodynamic results of different actuators with different blowing momentum (b)

Tuft flow visualization for the baseline flow and actuated flow [47].

Separation delay and drag reduction can't only rely on flow control, the shape of the nacelle matters also. A better nacelle design in aerodynamics can increase the lift coefficient, thus reducing the needs for flow control devices. In the JAXA high lift program, two types of nacelles (long cowling and short cowling) are equipped in the JSM high-lift configuration aircraft model. The configurations of both nacelles and their C_{Lmax} are shown in Fig. 2-22. The installation of the short-cowling nacelle does not affect the wing performance, but the long-cowling nacelle has a great influence on the performance. The difference is explained by the results from oil-flow visualization, shown in Fig. 2-23. As shown in the picture, large separations which govern the stall performance was induced by the interaction between the wing and the nacelle/pylon occurs after the nacelle/pylon and the slat-cutoff region for the long-cowling nacelle. For the short-cowling nacelle, large separation still remains but appears at the wing root, which behavior is not so serious [16]. Moreover, Smith et al. [48] performed a shape optimization using a new optimization framework for a UHBR engine including the design of wing airfoil sections, wing twist distribution, and the nacelle shape and placement for cruise flight conditions. Fig. 2-24 presents the pressure distributions on the nacelle inboard side and the upper surface of the initial and final design. It can be seen that the strong shock in the trench has been eliminated.

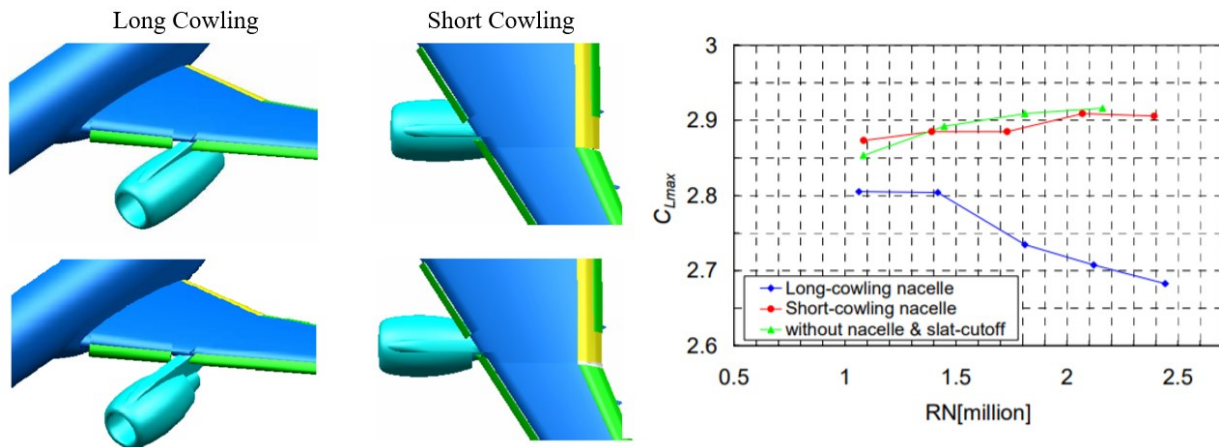


Figure 2-23 Configurations of two types of nacelle and their aerodynamic difference [16].

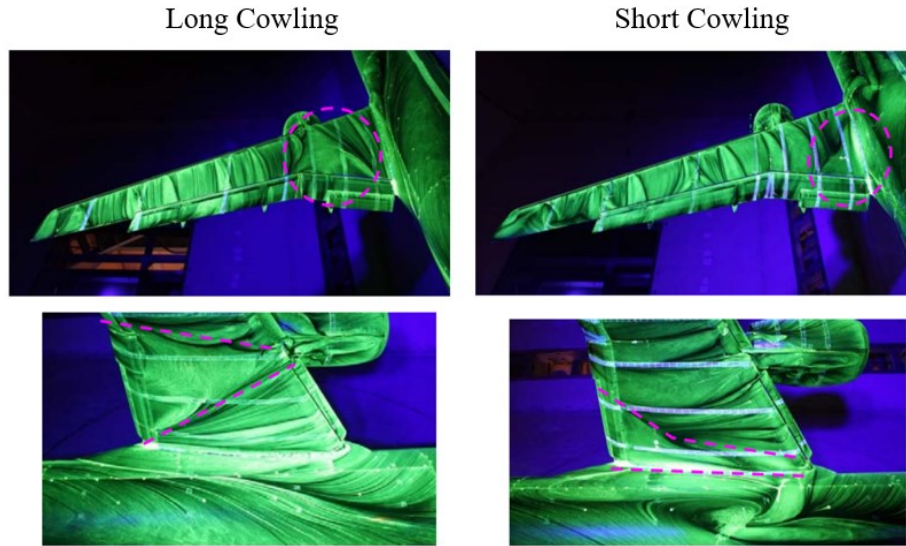


Figure 2-24 Surface flow pattern at post-stall conditions for both configurations [16].

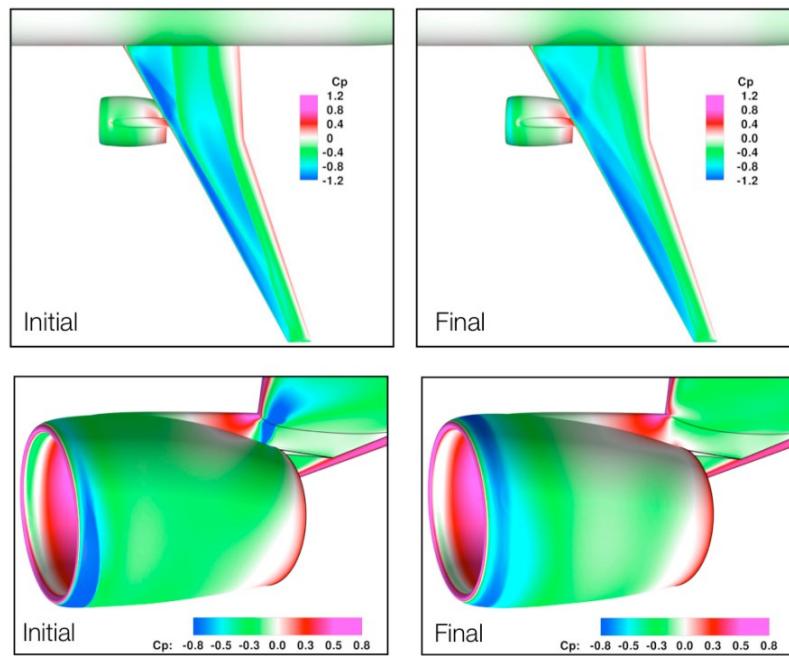


Figure 2-25 C_p contours on inboard side of initial and final nacelle [48]

Previous studies investigated the nacelle installation effect under an undisturbed flow condition during landing and takeoff. However, when the airplane is flying close to the ground, the effect of atmospheric turbulence on aircrafts must be considered: gust can hit and interact with the nacelle and it might cause separation. Research has shown that the gust could increase the fluctuation of

the inlet flow of a stalling nacelle (Fig. 2-26). However, no studies have been done on the effect of gust on the nacelle-wake separation: will the separation on the wing caused by the nacelle-wake increase under gust conditions?

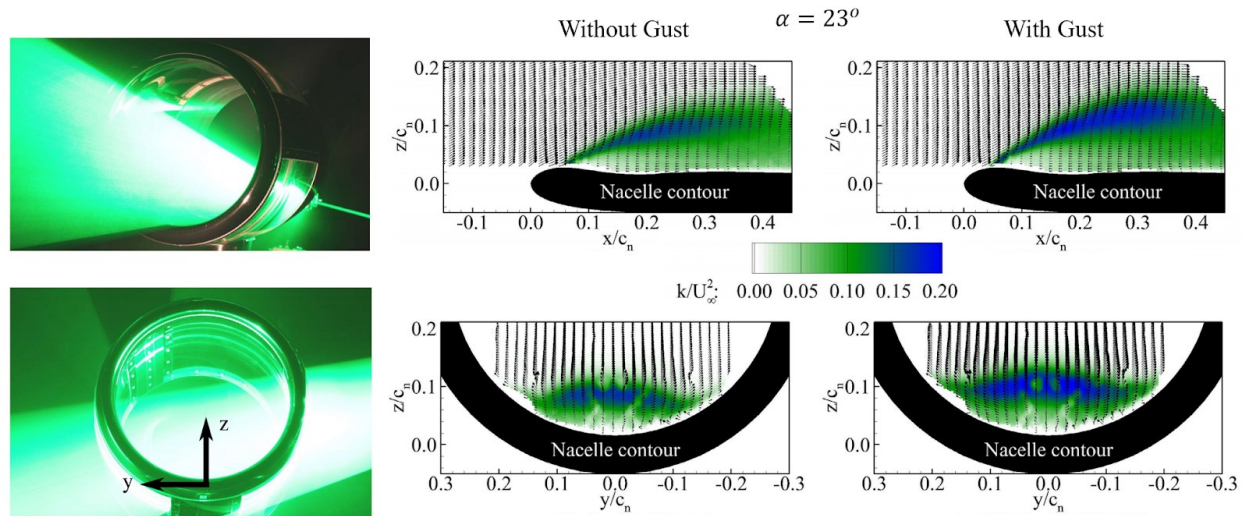


Figure 2-26 Color mapping of the normalized turbulent kinetic energy with the nacelle in undisturbed flow (without gust) and in disturbed flow (with gust) [49]

In conclusion, the mechanism of using nacelle chine for nacelle-wake separation delay is fully understood by the aid of force measurement techniques (six components strain gauge balances), pressure measurement techniques (pressure taps), and the technique of oil flow visualization including PIV, PSP, Infrared, and minitufts. For the effect of AFCs, the overall effect has been verified, but the mechanisms of how AFC induced flow suppress separation have not been explained, more tests need to be done with the help of the technologies mentioned previously. In addition, different types of AFC devices needed to be tested to find out the best way to delay separation. Lastly, the interference effects that arise from installation of the nacelle on the wing will not be present on either component in isolation. In order to reduce the separation at the engine-wing junction area, the nacelle shape and the shape of the wing airfoil section also needs to be considered during the design phase.

2.4 NOISE AND MAIN LANDING GEAR

Landing gear acoustics are considered as one of the dominant airframe noise sources for commercial aircraft. (Fig. 2-4) Prediction of landing gear noise and aerodynamics is still inaccurately and requires further investigation because of the highly complex geometry and the accompanied complicated flow phenomena [50]. For example, Fig. 2-27 showed a spectra comparison of the noise emission a full-scale landing gear measured in a wind tunnel and obtained from noise prediction models [51]. This is probably because most of the testing models is an unrealistic landing gear that does not include other parts of aircraft like the fuselage, wing, flaps, nacelles. This may cause an ignorance of cavity flow and the influence of other components. Moreover, there's also a lack of sufficient experimental data to verify the CFD code, especially for the HL-CRM model, since the NASA HL-CRM is a half model that doesn't include landing gears.

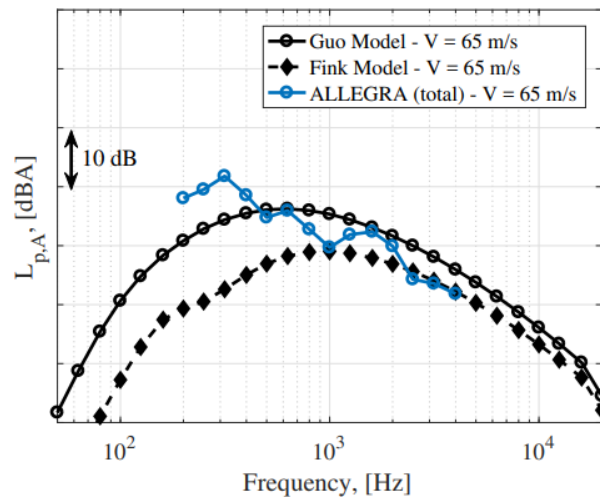


Figure 2-27 Spectra comparison of the noise landing gear emissions in the side direction between noise prediction models (Guo & Fink) and ALLEGRA wind tunnel experimental results [51]

Previous studies on landing gear targeted more on main landing gears having four or more wheels. This is because the airframe noise generated by the main landing gear is more dominant for large civil aircraft. The aeroacoustic characteristics of nose landing gears received little attention. However, for small regional jet class aircraft, both the main landing gear and nose landing are equally important sources [52]. Fig. 2-28 shows a source localization map for a Gulfstream G550 aircraft at gears up/down configuration at different frequencies. The plots indicated that the main gear and the nose gear sources have comparable strength [53]. Hence, more research is needed for nose landing gears. On the other hand, the research on main landing gear acoustics relies on simplified and high-fidelity main landing gears. For instance, Lazos [54] studied the mean flow features around a four-wheel landing gear using digital PIV technique. Presented in Fig. 2-29, the results showed that a vortex persisting in the wheel gaps is responsible for noise generation. These data are very helpful for CFD code and turbulence model verification. However, this research does not provide enough unsteady data. Guo [55] et al., presented experimental study and data analysis on a full scale Boeing 737 landing gear showed that high-fidelity landing gears are too complex to analyze, it is better to catalog gear components into groups. Later research then found out that the unrealistic model is responsible for inaccurate predictions, especially small details like bolt holes, hydraulic lines, electrical wires became important in noise generation.

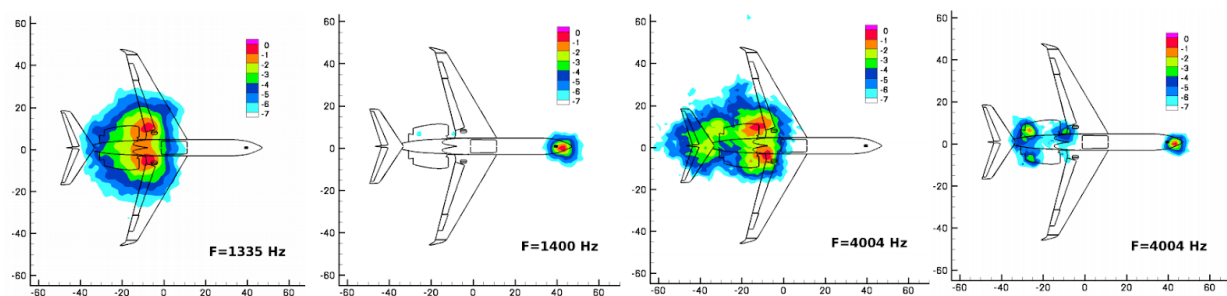


Figure 2-28 Source localization maps for a Gulfstream G550 aircraft at gears up/down configuration at different frequencies [53]

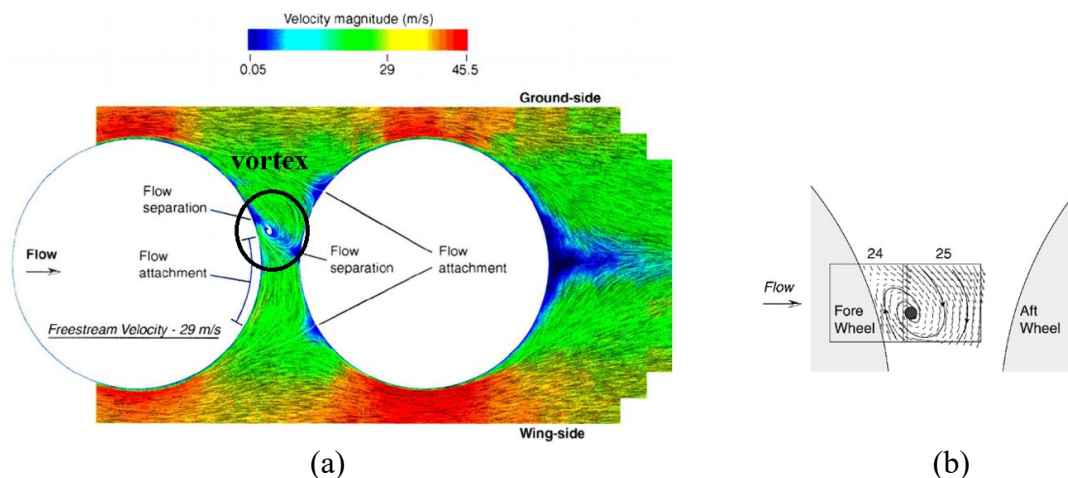


Figure 2-29 (a) 2D PIV measurement of the velocity magnitude and direction in inline wheels (b) Zoomed in view of the vortex between the wheels [54]

Jaeger and Soderman [56] studied the difference between high-fidelity landing gears and simplified landing gears experimentally, shown in Fig. 2-30. Results showed that different cases and the baseline model is almost 15dB at some frequencies. The author suggests that the design should be able to reduce 2 - 6 dB of noise if the details were designed carefully. Fig. 2-31 showed the results of the flow simulation of a full complexity geometry including the entire fuselage with wings and empennage and a simplified model both with and without the fuselage, wings, and empennage. The results show differences in RMS values but similar in mean values, the reason for this is unclear and the numerical results needed to be verified [57]. By comparing the results in different studies, the author believed that the complexity of the model needed to be defined for development. For instance, Jaeger's [56] work mentioned their high-fidelity model includes details that appeared on actual landing gear, but how realistic is it? is it 100% duplicated? Moreover, the simplification of the simplified model is not mentioned also, like Ricciardi [57] listed that their simplified model has fewer details, but keeps the main feature. What is the difference between their simplified ones and others? Hence, probably a common research "landing gear" model is needed for published results and comparison in academia.

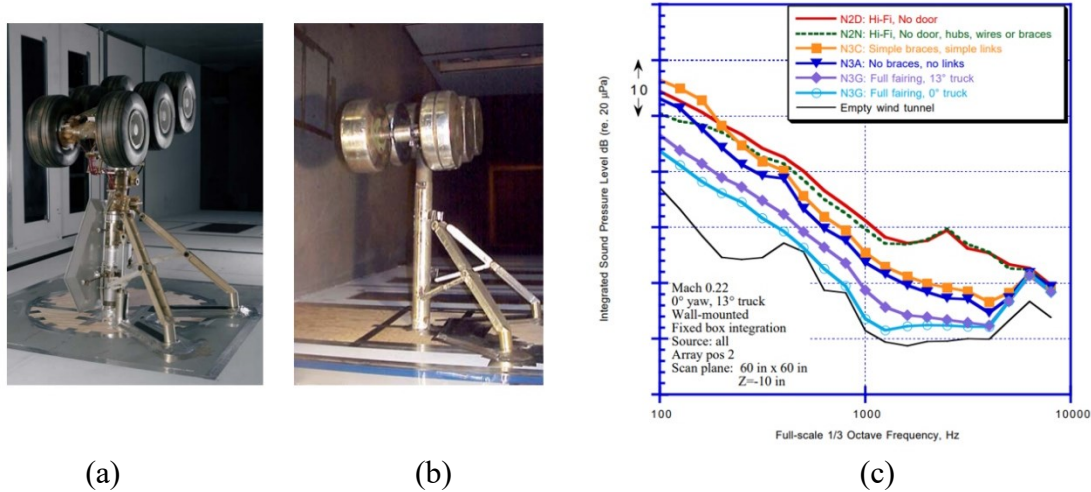


Figure 2-30 (a) Actual figure of the high-fidelity landing gear (N2D) (b) Actual figure of the low-fidelity landing gear with simplified braces and lock links (N3C) (c) Sound pressure level measured from different landing gear models [56]

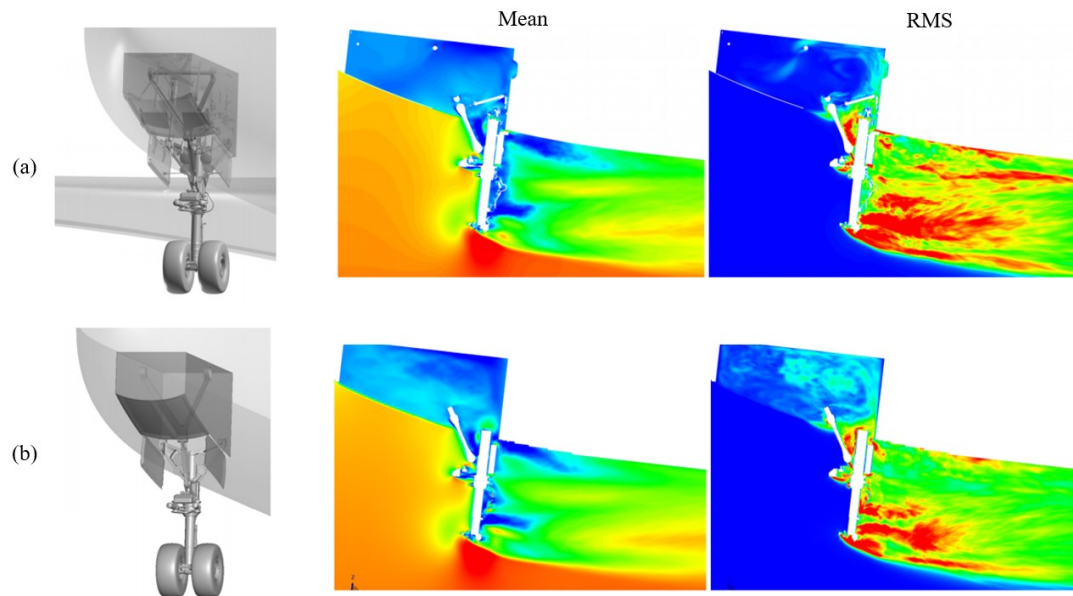


Figure 2-31 Velocity contours for both mean and RMS values for the complex configuration (a), simplified configuration (b) [57]

In addition to the missing details in the aircraft model, missing bay cavity in experimental models is probably also a reason for inaccurate simulation results. Cavity flow investigations are mostly done by CFD. For instance, Oza, et al. [58], studied the flow field in the cavity with nose

landing gear at different conditions, shown in Fig. 2-32, and concluded that cavity structural partition plays an important role in changing the mean flow features within the cavity. The results were verified by experimental data measured near the nose landing gear or pressure tubes inside the cavity. However, the results from CFD can't be fully verified due to the missing actual flow structure in the cavity. This is because it is hard to measure the flow inside the cavity. Hence, if the flow structure can be measured and can be used to verify numerical results, the understanding of landing gear noise can be improved.

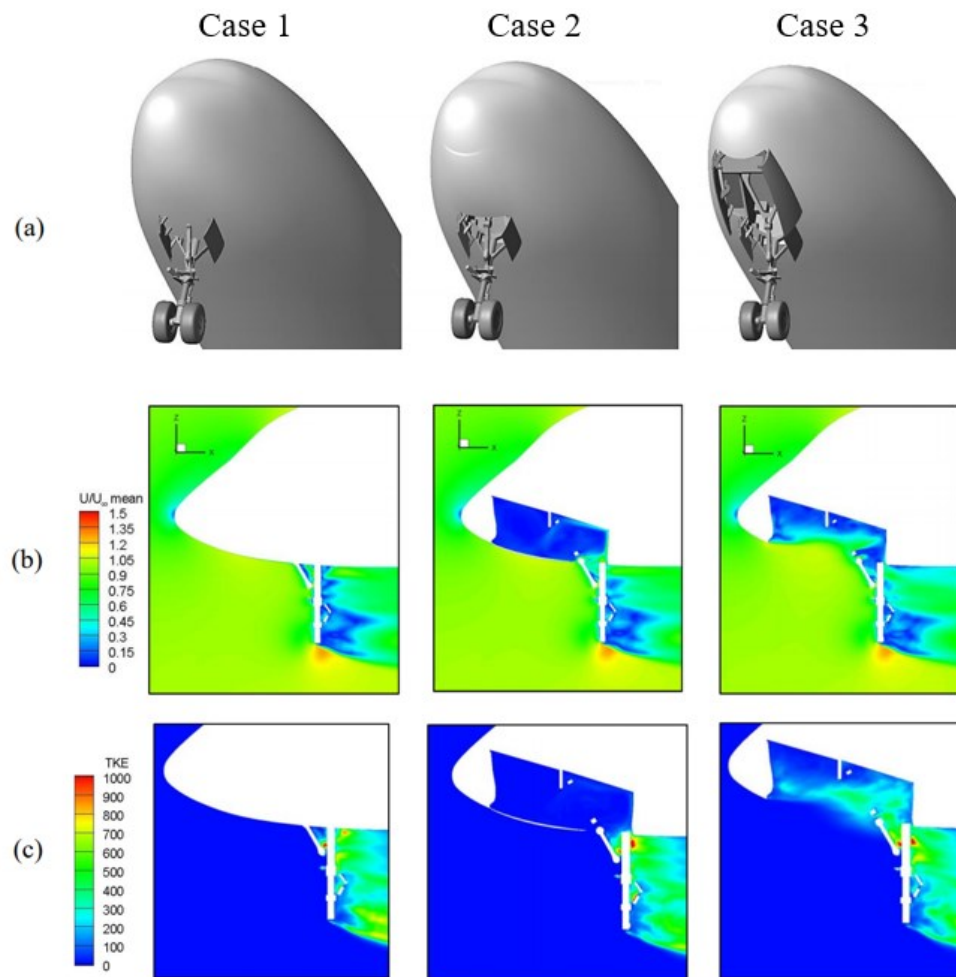


Figure 2-32 (a) Configurations of the testing models (a fully extended landing gear with a sealed cavity (Case 1)/partially exposed cavity (Case 2)/fully exposed cavity (Case 3)) (b) Results of the mean velocity magnitude through the same section that cuts through the strut and

the fuselage centerline (c) Results of the turbulent kinetic energy through the same section as (b)
[58]

Previous paragraphs mentioned the importance of the details of landing gears to noise, this is because that these struts, joints and dressings will lead to flow separation and vortex shedding in the boundary layers. An estimation of the contribution ratio of these components are shown in Fig. 2-33. High contributions of the braces, tire, and cylinder indicates why simplifying the details in these places will influence the experimental results. On the other hand, if the noise generated by these two sections are reduced, then the performance of airframe noise can be approved. Zhao et al. [59] mentioned three major methods to reduce landing gear noise in his review of noise reduction technologies for aircraft landing gears: add fairings, optimum design of wheel components, and applying flow control techniques.

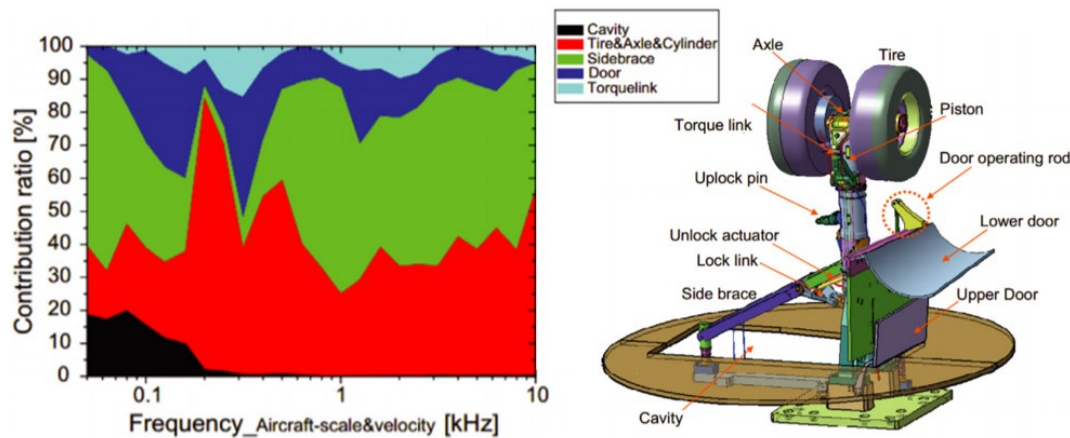


Figure 2-33 Estimation of the contribution ratio of model components to the noise emission and noise sources of a test model [59]

In Fig. 2-34, experimental results have shown that fairings and hub caps with porous materials could successfully reduce the noise from landing gears. The purpose of using porous materials rather than other solid materials is to prevent low frequency noise generated by solid materials, and to eliminate the deflection of high speed flow onto other uncovered gear components.

Another method to prevent vortex and flow separation in landing gears is to apply flow control techniques like air curtains, vortex generators, and plasma actuators. The original idea of the air curtain is to reduce the mixing and interaction noise in the wheels by shielding the incoming crossflow. The effect of the air curtain is shown in Fig. 2-35. Despite that the shield effect increases with a higher air curtain speed, the noise of the air curtain may also become a noise source. The purpose of VGs and plasma actuators is to eliminate the vortex shedding and wakes generated by the circular cylinder. Hence, plasma actuators and VGs are usually installed on the cylinder. Fig. 2-36 shows the results of noise reduction using plasma. The last method is to propose an alternative optimized design shape for landing gear components like an optimal designed side-stay, or different wheel spacing to avoid weight penalties [59].

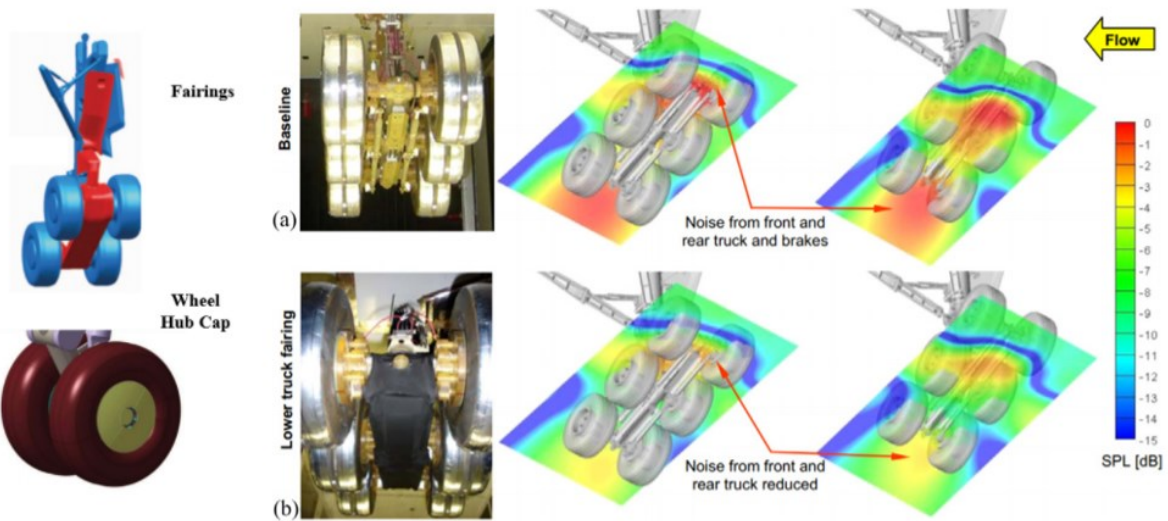


Figure 2-34 Conceptual design of fairings and the effects of porous fairings on landing gears [59].

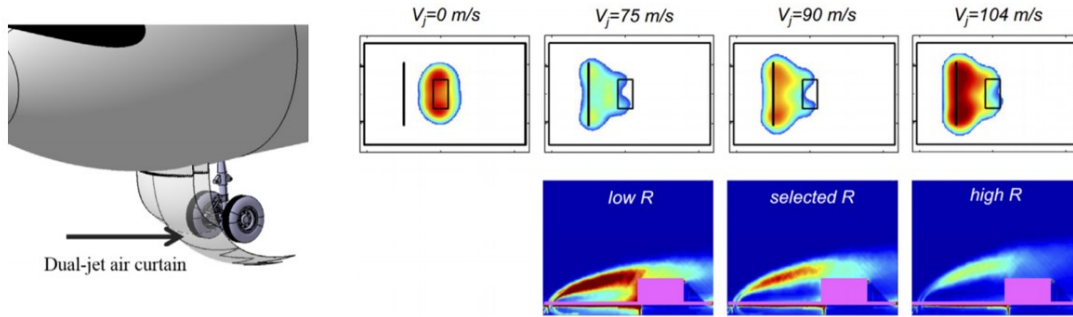


Figure 2-35 Effect of air curtain on landing gears [59].

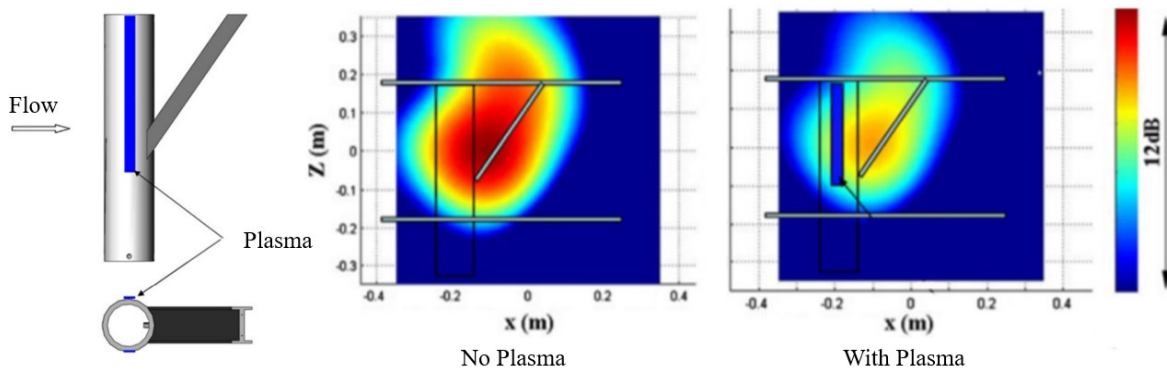


Figure 2-36 Effect of plasma actuators on the cylinder [59].

In general, the noise sources from landing gears are mainly confirmed. However, the industry still requires the ability to simulate airframe noise accurately. Moreover, the noise reduction techniques mentioned above are so far during the research stage, except for the hubcaps. Most methods are still in laboratory work since the tradeoff between the noise reduction and the additional weight and complexity is not clear. The above reasons provide the potential of including landing gears in the UW HL-CRM model in order to collect high-quality data using microphone arrays and to provide more noise reduction solutions for landing gears.

2.5 REMOVABLE HORIZONTAL AND VERTICAL TAILS

The application of horizontal and vertical tail behind the main wing is to provide stability and control during flight, helping to adjust the changes in position of the center of pressure or the

center of gravity and to overcome cross wind especially during landing and takeoff phases of the flight [60]. The aerodynamic force on both tails must be determined accurately before calculating the aircraft's longitudinal and directional equilibrium and stability. However, the calculations of the aerodynamics forces from an isolated horizontal or vertical wing is not correct. When the wing is attached to the aircraft during a sideslip, the fuselage and the downwash from the main wing will change the results, leading to a miscalculation. Hence, the research interest to study the downwash and sidewash effects is to provide accurate predictions [61][62][63]. Another interest is to reduce the size of the vertical tail. One of the most important functions of vertical tail is to provide enough yawing moment when the engine fails in the takeoff, known as the "One-Engine Inoperative (OEI)" or asymmetrical flight condition. A concept of the general OEI situation is shown in Fig. 1. The air flow around the aircraft in this situation is generally complex: the aircraft is flying at a sideslip angle, in high lift condition that results in low speed, high thrust, additional drag, and limited controllability. During the OEI condition, the yawing moment induced by the engine and crosswind has to be balanced by the vertical tail. Therefore, the forces and moments in the OEI condition have to be calculated in order to determine the size of the vertical tail. In the past, not only these data have been overestimated, but also the vertical tail is oversized during normal operations, resulting in an increase in drag, and decreasing in transport efficiency [64]. Hence, the industry hopes to design a smaller vertical tail that still provides enough side force for the OEI condition [65].

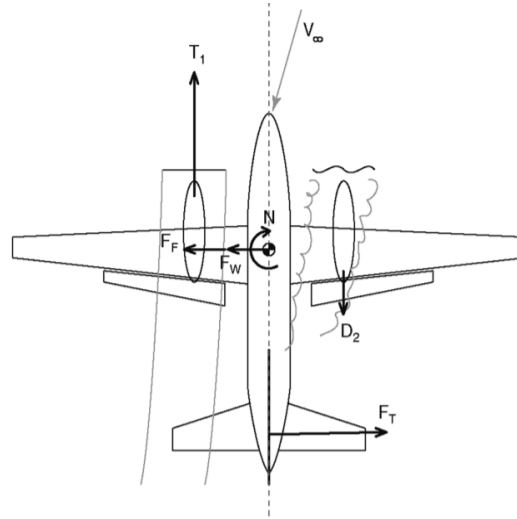


Figure 2-37 Concept of the OEI situation [64].

In aircraft design, the aerodynamic forces are calculated from the horizontal wing alone. The lift of the horizontal wing may be affected by the separation from the wing-fuselage region, known as the wing-fuselage interference effects, and the wingtip and downwash from the main wing, known as the wingtip and downwash effects. Hoang and Bui [61][62] provided good agreement experimental and numerical investigations on both effects. The results in Fig. 2-38 (a) show that the interaction between the fuselage and tail alters the lift coefficient on the horizontal tail when angle of attack is larger than 12 deg. Fig. 2-38 (b) shows the influence of the separation in the wing-fuselage region to the horizontal tail. The lift coefficient of the horizontal tail influenced by the wing with fuselage (WVH-WF) is 40% lower than the main wing only (WVH-W). Moreover, in Fig. 2-39, numerical results show that the streamlines were separated very strongly in the wing-fuselage area. These results point out that the horizontal tail was affected strongly by the vortices from the separations in the wing-fuselage interference region, thus resulting in balance and stability problems of the aircraft. In addition, the effect of downwash is shown in Fig. 2-40. By comparing a horizontal tail in viscous flow and a horizontal with main wing, the results show that the downwash of the main wing decreases the lift on the horizontal tail.

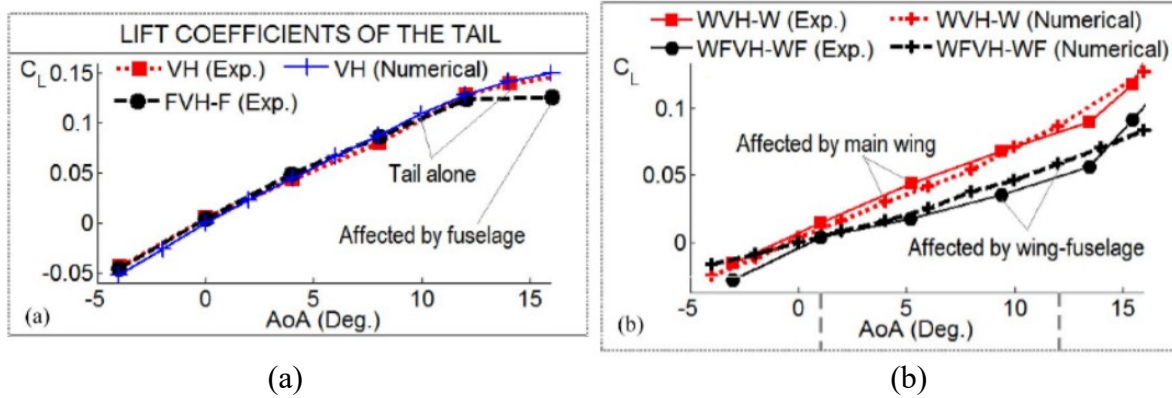


Figure 2-38 (a) Lift coefficients of the horizontal tail for tail alone and tail affected by fuselage (b) for tail affected by the wing only and wing-fuselage [61].

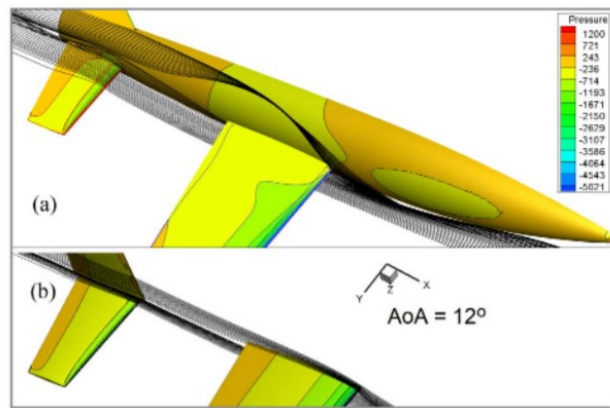


Figure 2-39 CFD results of flow through a complete model and wing tail combination [61].

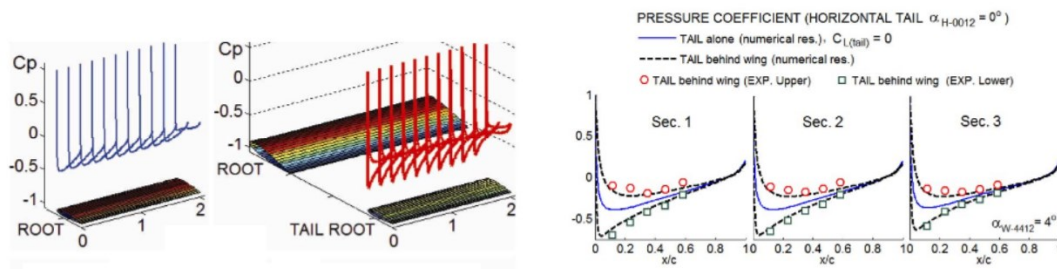


Figure 2-40 C_p on the horizontal tail [62].

On the other hand, the aerodynamic forces on the vertical tail are not calculated by an isolated vertical tail, but by mainly two classical methodologies (DATCOM ESDU) derived from NACA WWII reports. These two methods, according to Nicolosi et al. [63], give quite different

results for some configurations. Hence, a CFD analysis is done at the University of Naples “Federico II” to provide a deeper investigation on the body and wing interference effects on the vertical tail (Fig. 2-41). Fig. 2-42(a) shows the CFD results of the fuselage effect compared with different tailplanes (rectangular fin, constant taper) and the semi-empirical method. Y axis represents the sideforce coefficient ratio (where $C_{Y\beta_v}$ is the vertical tail sideforce coefficient due to the body sideslip, and C_{Y_v} is the sideforce coefficient of an isolated vertical tail), and the X axis represents the parameter $b_v/2r$ (where b_v is the height of the vertical tail and $2r$ is the fuselage average diameter). The results demonstrate the body effect of sideslip on the vertical tail, and the semi-empirical method, which have been derived decades ago, agrees with the CFD results. For the wing-effect, Fig. 6(b) shows the effect of wing position and wing aspect ratio on the vertical tail sideforce, where $C_{Y\beta_v}$ is the sideforce coefficient of an isolated vertical tail due to a sideslip angle, and $C_{YW\beta_v}$ is the sideforce coefficient influenced by the wing sideslip. The results show that there's nearly zero wing effect on the mid wing, but the difference between the high and low wing is around 5%. This is related to the influence of an induced angle wing-tip vortices (Fig. 2-41(b)). Last, Table 2 shows the comparison between CFD results and semi-empirical methods. Both methodologies show quite different results for certain configurations, this is probably due to an overestimate of the interference effects.

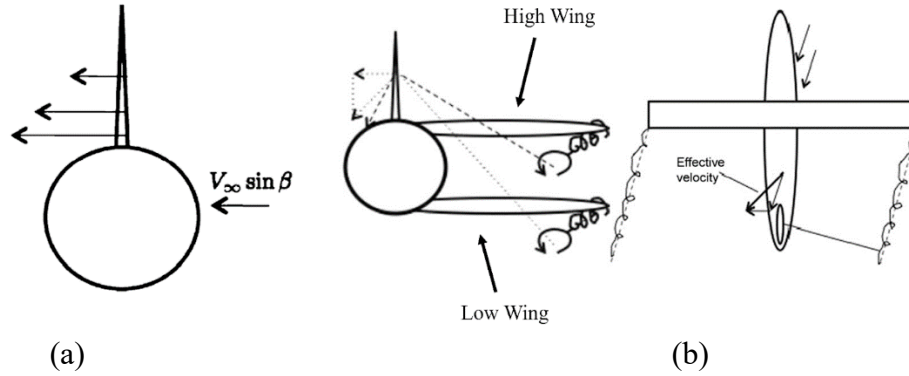


Figure 2-41 Concept of the (a) body effect (b) wing effect on vertical tail [63].

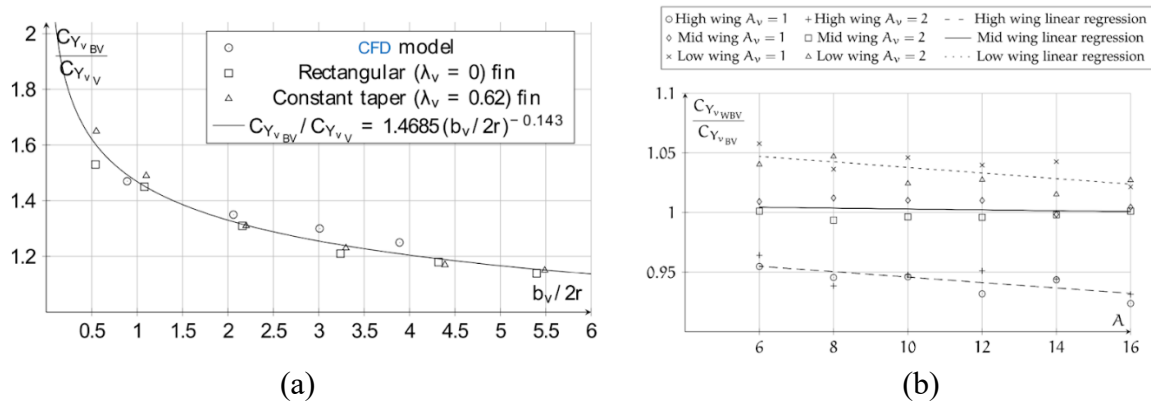


Figure 2-42 (a) Effect of the body effect of the CFD model comparing to different tailplanes (the taper ratio of the CFD model is 0.55) (b) Effect of the wing at various aspect ratios and positions (A_V represents different taper ratio of the vertical tail) [63].

Comparison among $C_{Y_{\beta V}}$ (rad^{-1}) results.

Hor. pos.	Wing pos.	CFD	DATCOM	% Δ	ESDU	% Δ
Body	High	0.6968	0.5619	-19.4	0.7423	6.5
Body	Mid	0.7231	0.6803	-5.9	0.8655	19.7
Body	Low	0.7310	0.7987	9.3	0.9835	34.6

Table 2 Comparison among the sideforce coefficients from the CFD model and the semi-empirical methods [63].

For the OEI studies, Schroyen, et al. [64], simulated a model at OEI conditions. The testing geometry and results of the sidewash at the tail are shown at Fig.2-43. Numerical data were

compared to an early wind tunnel test from the Nederlands Luchtvaart Lab. By comparing the simulation to the measurements, it is found out that the sidewash is overestimated. This is probably because of several reasons: inaccurate prediction of the wing trailing vortex sheet, removal of the propeller, and numerical dissipations due to a rather coarse grid. Moreover, the OEI condition in this research is a simplified version without a horizontal tail, no sideslip angles, and the aircraft is assumed to operate in the linear region of the lift curve. Hence, providing a more accurate OEI condition prediction ability is needed to determine the stability and controllability considerations for vertical tail design.

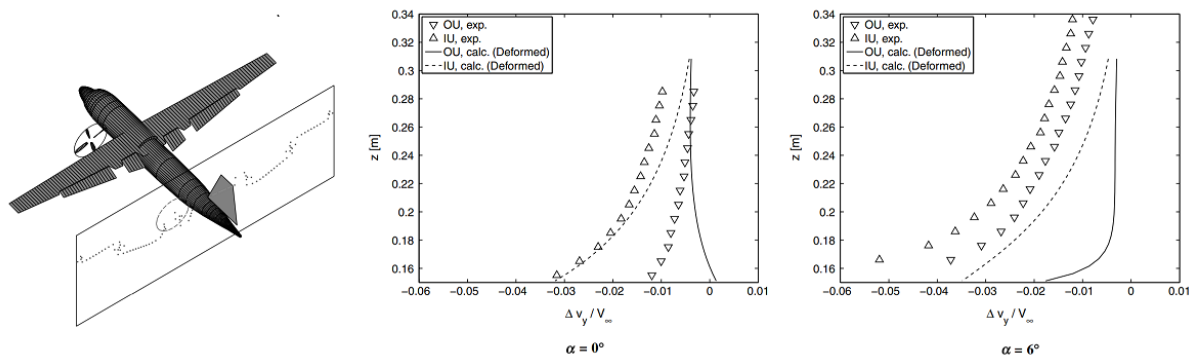


Figure 2-43 Testing geometry and numerical results compared with wind tunnel data of the sidewash at the vertical tail (IU: inboard, OU: outboard) [64].

AFC is a technology that can delay the flow separation on the rudder and increase the side force that it generates. With an additional side-force on the rudder, the size of the vertical tail could be reduced [65]. Among various AFC methods, synthetic jet actuators were first applied on vertical tails. This is a type of actuator that uses only the surrounding fluid, and does not include an external air supply, which creates additional complexity and weight. Ejections are formed by flow moving back and forth induced by movement of a diaphragm through a small opening. A schematic drawing of the synthetic jet actuator is shown in Fig. 2-44 [66]. Rathay et al. [67] studied the overall effect and the spanwise location (activating only the jets near the root, the tip or only at the

middle section) effect of synthetic jet actuators on a vertical tail, as well as the flow physics of the interaction of synthetic jets with the flow over the rudder using SPIV technique. Fig. 2-45 displays the location of the synthetic jet actuators and the measurement domain of the SPIV. Fig. 2-46 shows that the side force was increased up to 18% and the activation of the middle jets provides the greatest side force coefficient. In Fig. 2-47, results from SPIV measurements indicate the effect of synthetic jet actuators: flow separation was observed without flow control, but for the case when all jets are activated, the boundary layer is thicker and the flow never separates. Moreover, when only one jet is activated, the propagation of the jet causes the flow to reattach along its path to the plane.

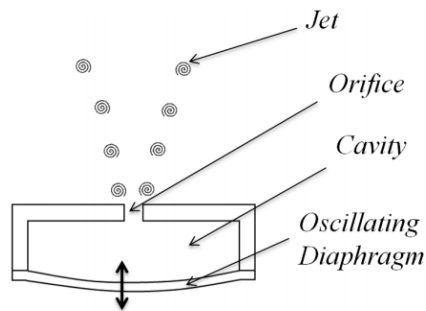


Figure 2-44 Conceptual design of synthetic jet actuators [66].

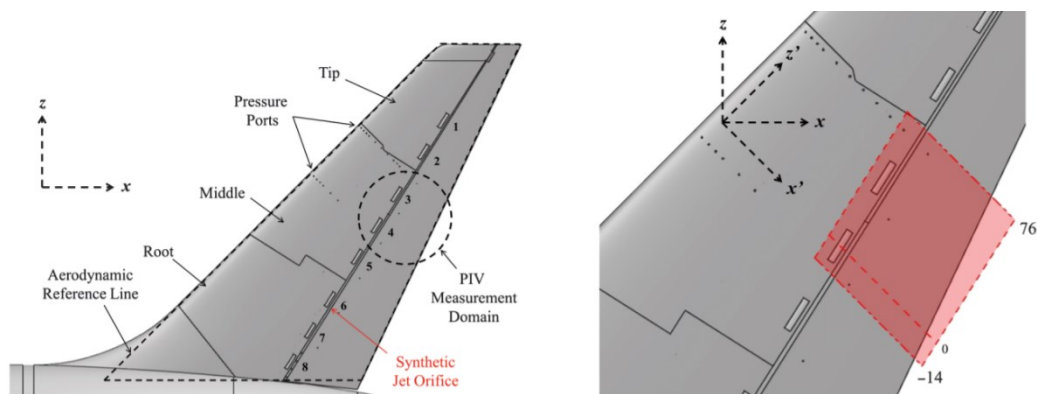


Figure 2-45 Location of the SJA and the measurement domain of the SPIV [67].

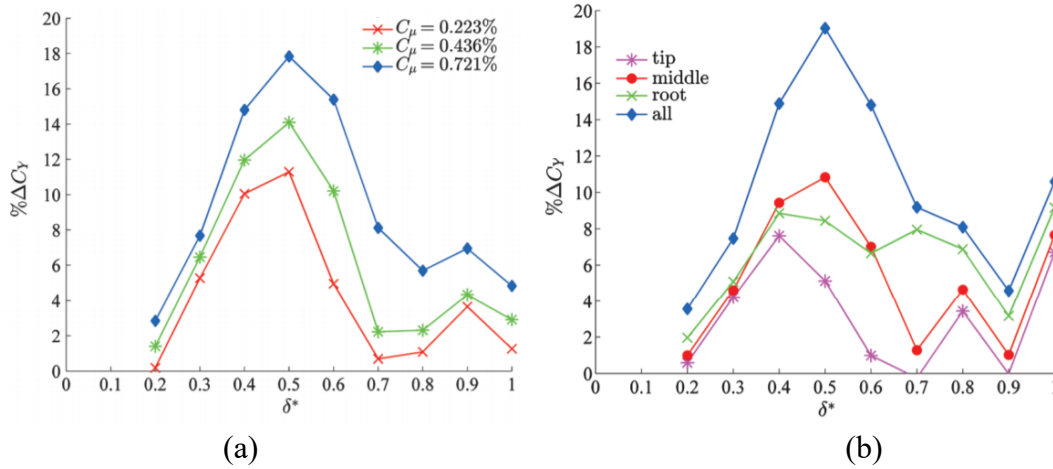


Figure 2-46 (a) Overall effect of spanwise jet on vertical tail with different jet momentum injection (b) Effect of the spanwise jet location [67].

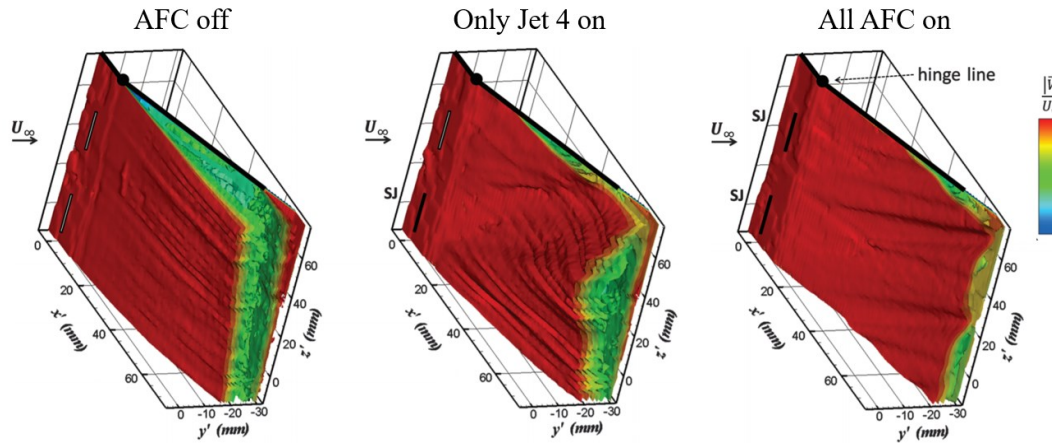


Figure 2-47 Results from the SPIV measurements [67].

Jansen et al. [68] proposed an explanation for the synthetic jet side-force generation mechanism from CFD results: a large, low-pressure footprint generated by an oblique vortex was observed on the surface of the rudder. This low pressure region will re-energize the spanwise flow and provide more impact on the side force rather than the synthetic jet itself. Moreover, this theory reveals the importance of the spanwise spacing of AFC devices: if the jets were placed too close together, this oblique vortex will not fully develop, and will limit the benefit from additional oblique vortices. Fig. 2-48 demonstrates the existence of the oblique vortex by comparing the

surface streak patterns of an AFC on/off case. Fig. 2-49 displayed the views of streamlines around root/inboard separation to show the impact of the root oblique vortex and its action to the oblique vortex formed by synthetic jet actuation.

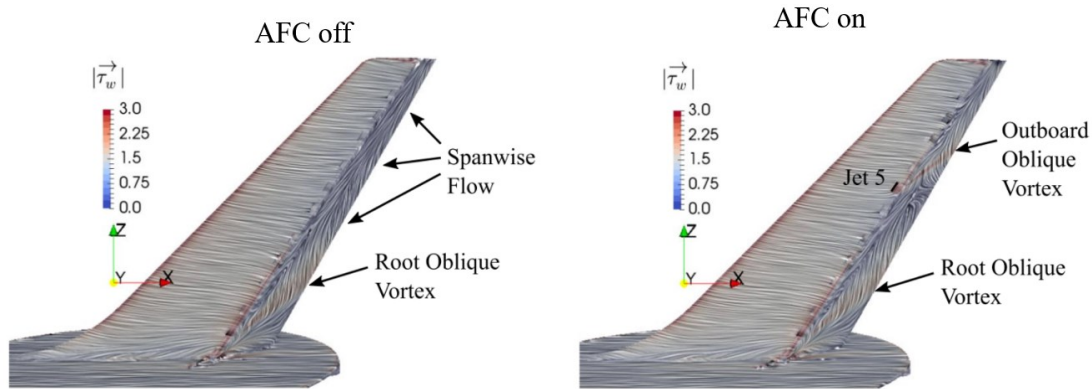


Figure 2-48 Comparing the surface streak patterns colored by wall shear-stress magnitude [68].

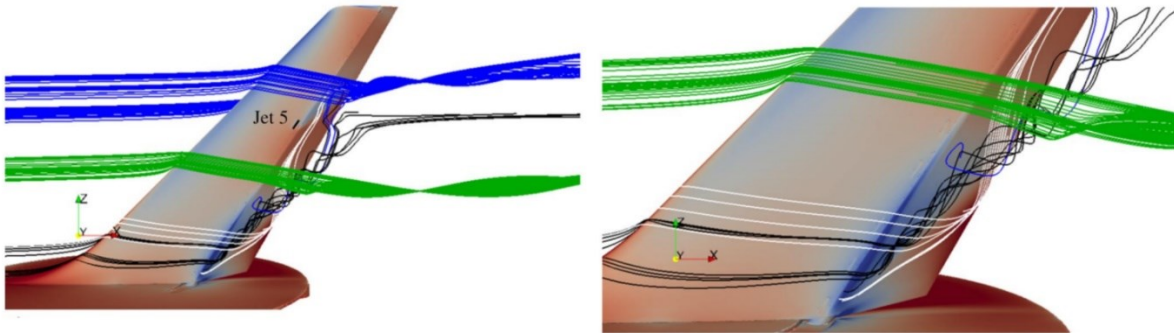


Figure 2-49 Views of streamlines around root/inboard separation [68].

Another type of AFC used in research is the sweeping jet actuators. Fig. 2-50(a) shows a schematic drawing of the concept of sweeping jet actuator. Despite the fact that this type of actuators requires steady supply of compressed air, they use less air for similar levels of effectiveness for steady blowing. The supplied air will enter the main cavity of the actuator, being curved and rushed to the outlet increasing the pressure at the inlet to the upper feedback channel. This creates flow in the feedback channel that pushes the entering jet back to the opposite surface and repeats the process. Experimental results of the effect of the numbers of sweeping jet actuators

done at NASA Ames Research Center are demonstrated in Fig. 2-50(b). Vertical tail equipped with 31-actuator AFC configurations can provide greater than 20% increments in side force. However, it is surprising that reducing the number of actuators down to 12 did not significantly affect the side force [69].

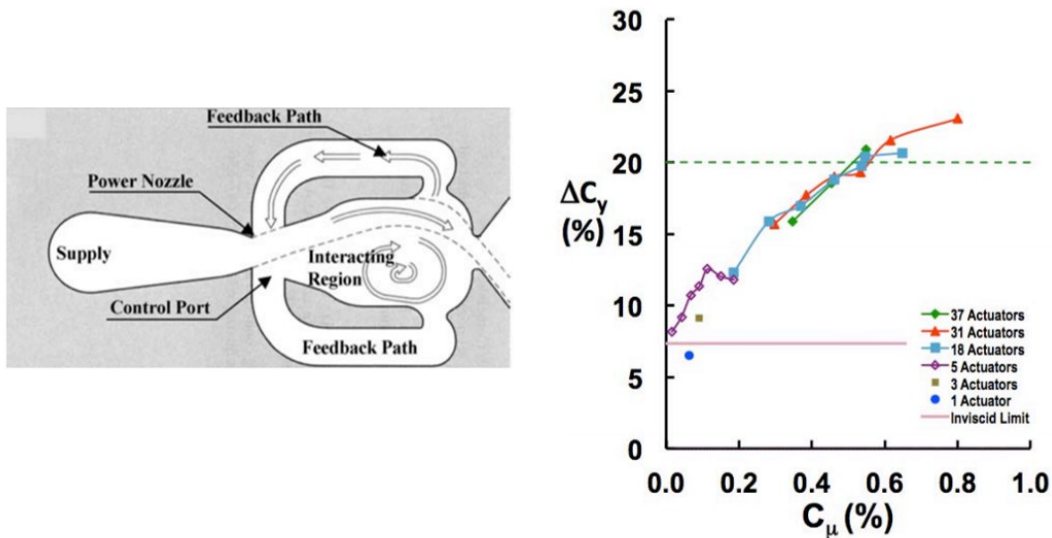


Figure 2-50 (a) Conceptual design of sweeping jet actuators (b) Effect of different numbers of sweeping jet actuators installed [69].

Previous research focused on studying the efficiency of a specific type of AFC actuators. Understanding the mechanism and parameters that influence the overall effect. A comparison of the efficiency of different flow control devices (including mechanical vortex generators, fluidic vortex generating jets VGJs, Tangential Wall Jets, and Pulsed Jet Actuators) on a vertical tailplane was tested by Scholz [70], shown in Fig. 2-51. The results showed that different methods turned out to be better than others at different ΔC_Y or $\Delta C_{Y_{max}}$ regions, and Scholz summarized the best method that is suitable for each region, shown in Table 3. Since previous research focusing on studying the effects of the same type of AFC techniques on the vertical tail. Probably a

combination of different flow control techniques is the way to increase the potential of the vertical tailplane to create a large side force.

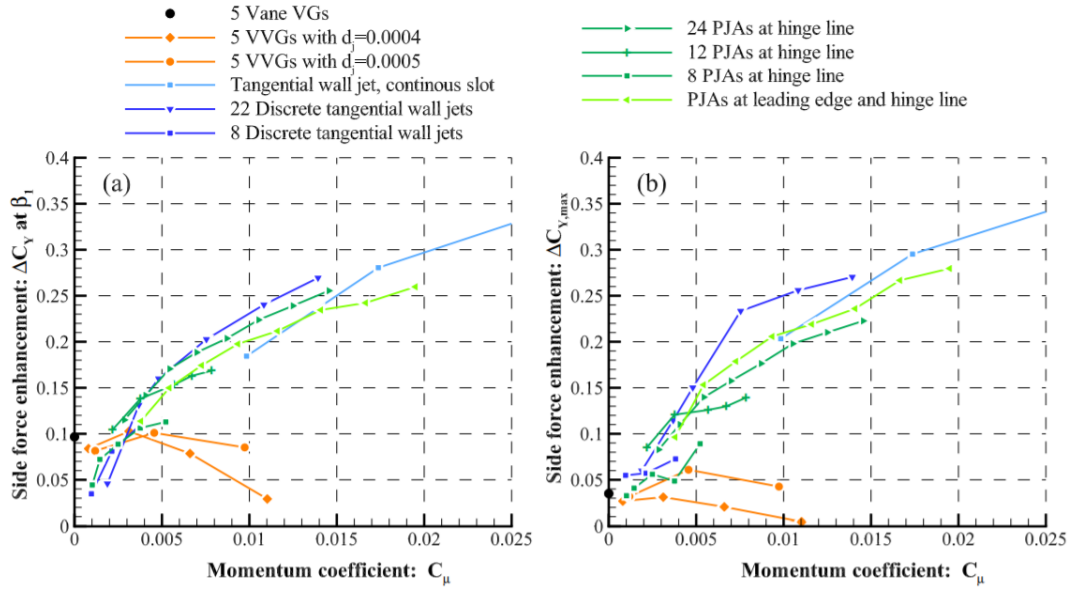


Figure 2-51 Effect of different AFC methods on side force [70].

ΔC_Y @ β_1	Best method	$\Delta C_{Y,max}$	Best method
0 ... 0.1	Vortex generating devices	0 ... 0.05	Vortex generating devices
0.1 ... 0.2	Pulsed jet actuators or Discrete tangential jets	0.05 ... 0.15	Pulsed jet actuators or Discrete tangential jets
0.2 ... 0.25	Discrete tangential jets	0.15 ... 0.25	Discrete tangential jets
> 0.25	Continuous tangential wall jet	> 0.25	Continuous tangential wall jet

Table 3 Summary of the best AFC method at different C_Y regions [70].

The purpose of adding AFC to vertical tails is to reduce its size, and the ultimate goal is to remove the vertical tail but remain its functions. Hence, studying the aerodynamic configuration from nature like the flight of different birds, which have no vertical tails, might lead us to a tailless aircraft design. Sachs [71] provided research on bird flight, trying to figure out why birds need no vertical tail. This research indicated that if the wing alone can generate enough yawing moment $C_{n\beta} > C_{n\beta min}$, then a vertical tail may not be needed. The sweep of the slotted tip wing or the gull sweep wing is the most important feature for the ability of the wing to generate stabilizing yawing

moments. Fig. 2-52 then compares the aerodynamic yawing moment of bird wings (gull wings with sweeps/wings with slotted tips) with sweep and without sweep. The results showed that there's a significant difference in the yawing moment $C_{n\beta}$ in the higher lift coefficient range for both types of wings. This behavior demonstrates the importance of wing sweep for yaw stability. Hence, if the main wing of the aircraft can provide enough yawing moment, then a vertical tail may not be needed.

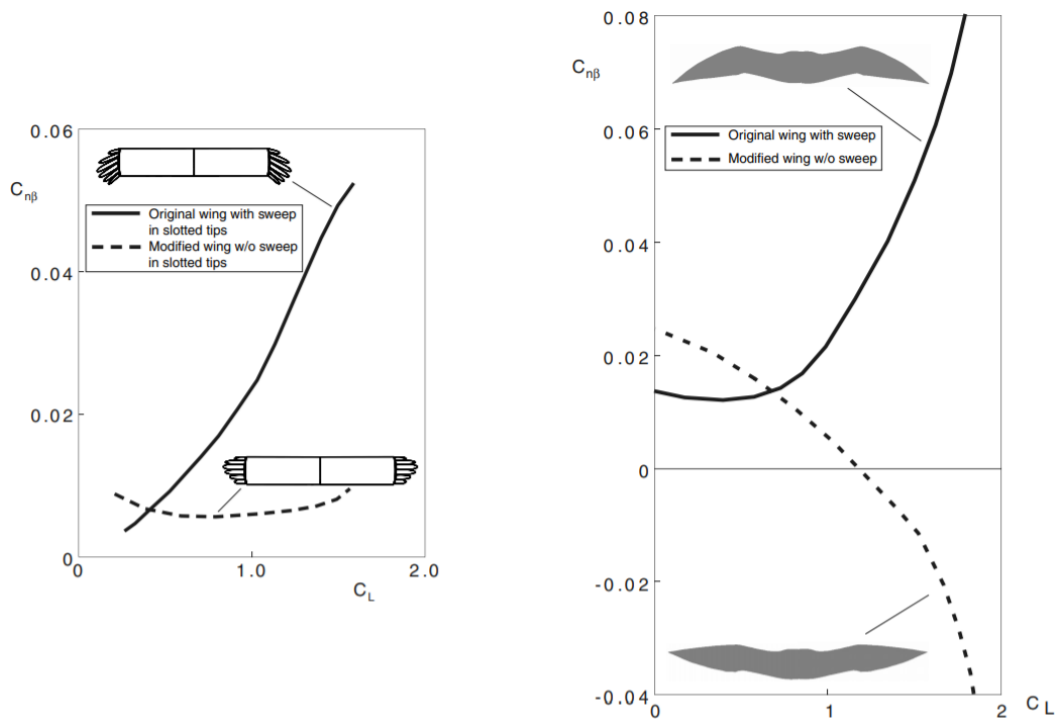


Figure 2-52 Comparison of yawing moment due to sideslip of different bird wings [71].

In general, the research in reducing the size of vertical tail attracts way more interest than other topics. This is because the vertical tail is oversized during normal flight conditions. This also explains that nearly no research has been done on reducing the size of the horizontal wing. Moreover, previous AFC research at NASA uses an isolated vertical tail model only, which neglects the effect of the fuselage and wing. This provides a potential of studying overall AFC effects on the aircraft and the interference effects on the horizontal tail and vertical tail on the UW

HL-CRM using flow visualization techniques like PIV to study the flow field affected by AFC devices, PSP and oil film to measure the aerodynamics forces (pressure & shear) on the horizontal and vertical tail.

2.6 FLOW VISUALIZATION AND OFF BODY FLOW FIELD STUDIES

Flow visualization is one of the most effective tools in analyzing flow fields, and it is a very important method for improving our understanding of complex fluid flows. These flow patterns may be displayed in many ways using various techniques such as photos of smoke, pressure distribution on the surface, or vector field measured by PIV [72]. For high lift studies, flow visualization is an important technique to find separation areas on aircrafts. For example, the high lift program at JAXA studied the flow field on the wing at stall conditions using various methods (Fig. 1) like using PSP to measure surface pressure, using tufts, oil flow, and china clay for surface flow field study and PIV for off body flow field study, and then provided a solution to eliminate the separation on the wing (more details shown in Chapter 2.03, and 2.07) [11]. These articles provide some examples of the study interests including slats, flaps, transition, skin friction, wing wake, wing flutter condition, and juncture flow for applying flow visualization technologies on the high lift common research model.

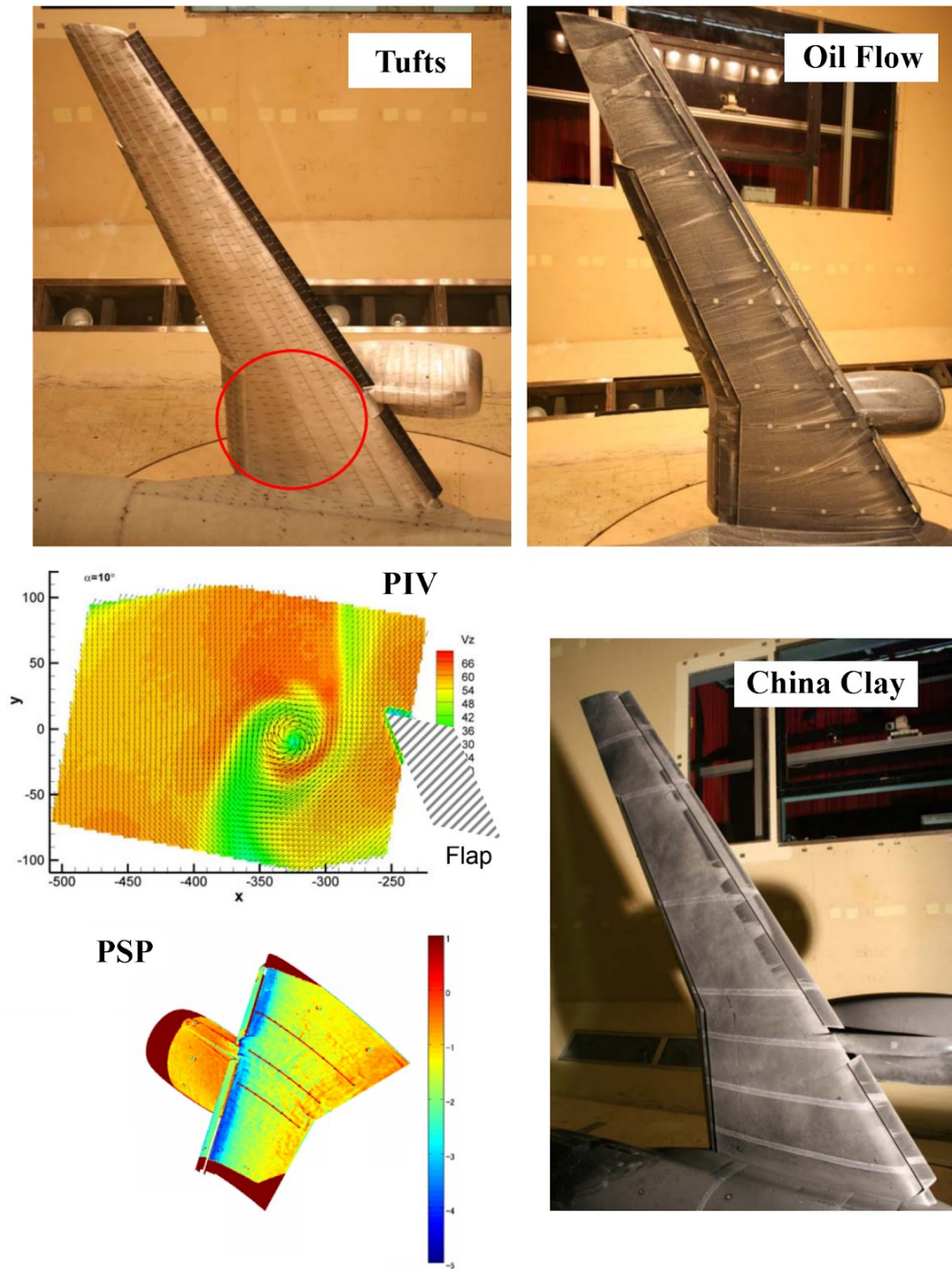


Figure 2-53 Different visualization methods used in JAXA study [72][11].

Apart from the methods used in JAXA, Temperature Sensitive Paint (TSP) is also a method to measure transition and displays the separation region. The mechanism of TSP is to use the temperature difference between the boundary layer regimes of laminar and turbulent flow. Klein

et al. [73] applied TSP on a nacelle and studied the difference between different heating methods: increasing or decreasing the flow temperature by adjusting the injected liquid nitrogen rate (temperature step method), placing a heated carbon nanotubes (CNT) on the nacelle surface, heating the nacelle surface by infrared laser. In Fig. 2-54, the results show that TSP might be a good way to measure boundary layer transition on complex models since all the heat methods successfully display the laminar to turbulent transition on the nacelle model, and the transition position is not affected by the heating method.

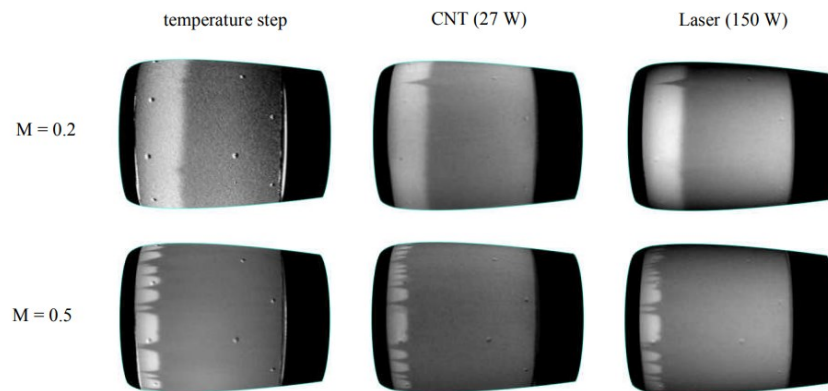


Figure 2-54 Visualization of boundary layer transition on the nacelle model [73].

Smoke wire is one of the oldest visualization techniques that offers the insight into the phenomena in complex fluid flows [72]. Kirk and Yarusevych [74] investigated the formation and development of the roll-up process in separation bubbles using smoke wire visualization, providing a new outlook on separation bubble dynamics compared to the previous surface flow visualization techniques. In Kirk's work, smoke-wire was first installed upstream of the airfoil to give a general overview of the flow development around the airfoil, identifying the extent of the separation bubble and the separation location (Fig. 2-55(a)). Then, the smoke-wire is installed within the separated flow region across the span of the airfoil to visualize the flow structure of the separation bubble. Fig. 2-55(b) and (c) displays the shear layer roll-up process, breakdown of vortices and

transition to turbulence. More results presented in this paper are discussed in Chapter 2.07 Boundary Layer Studies.

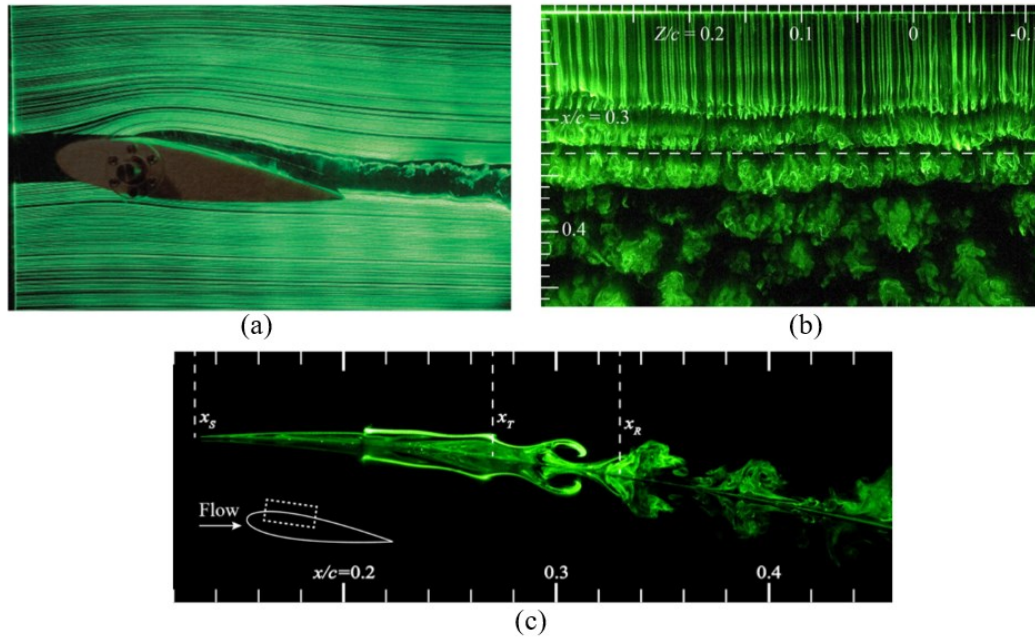


Figure 2-55 Smoke wire visualization of the flow (a) around the airfoil (b) within the separation bubble (top view) (c) within the separation bubble (side view) at $\alpha = 8^\circ$ [74].

As mentioned in Chapter 2.01, low drag is one of the main goals for designing the next generation aircrafts. In order to achieve drag reduction beyond the current technology level, an accurate computational design tool is necessary. Hence, this requires a number of high-quality experimental skin-friction data on aircraft wings to validate CFD results [75]. Among various skin-friction measurement methods, oil film interferometry (OFI) is suitable for displaying skin friction throughout the whole wing surface. However, the uncertainty of the OFI measurement is relatively high, mainly caused by temperature effects. This leads to a sparse comparison of computed results and measured raw data in early developments [76]. After studying the effect of wall temperature on OFI measurements, JAXA displays the ability to provide skin friction distribution from OFI measurement corrected by temperature distribution measurements from IR camera: Fig. 2-56(a)

shows the initial image from OFI measurements, and Fig. 2-56(b) displays the corrected skin friction distribution [75].

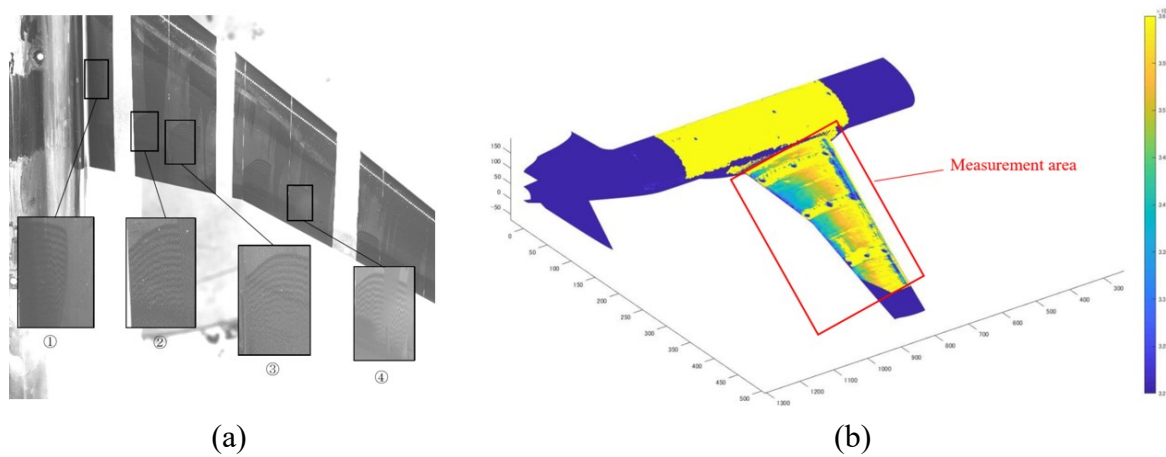


Figure 2-56 (a) Original oil film interferometry results (b) Corrected oil film interferometry results [75].

Another interest is juncture flow. junction flow happens when a boundary layer encounters an obstacle attached to the same surface, they are usually formed around a wing-fuselage or wing-pylon intersection area for civil aircrafts. These flows are complex, hard to predict and may exhibit flow separation regions that could reduce aircraft performance. For instance, Fig. 2-57 displays large variations in the predictions of the velocity and turbulent stress for wing-body junction flow [77]. Hence, NASA build a Juncture Flow (JF) model (Fig. 2-58(b)) aiming to provide a publicly high quality flow field and to understand the aspects of the wing-body juncture flow physics such as how several vortical structures (horseshoe vortex, corner vortex) coexist, and how different factors (wing bluntness, wing sweep) influence the flow. A validation experiment was done by Kegerise, and Neuhart [78] using various measurement techniques like Laser Doppler Velocimetry (LDV), infrared imaging, oil-flow visualization, pressure ports and transducers [77][78]. PIV measurement is also available on the NASA JF model, Fig. 2-58(c) shows a photograph of the PIV system installed at the trailing edge of the JF model. Despite that the JF model has the ability to

do PIV experiment, it was not done in the previous junction flow CFD validation experiment, data from PIV measurement have not been published until the end of this paper [79].

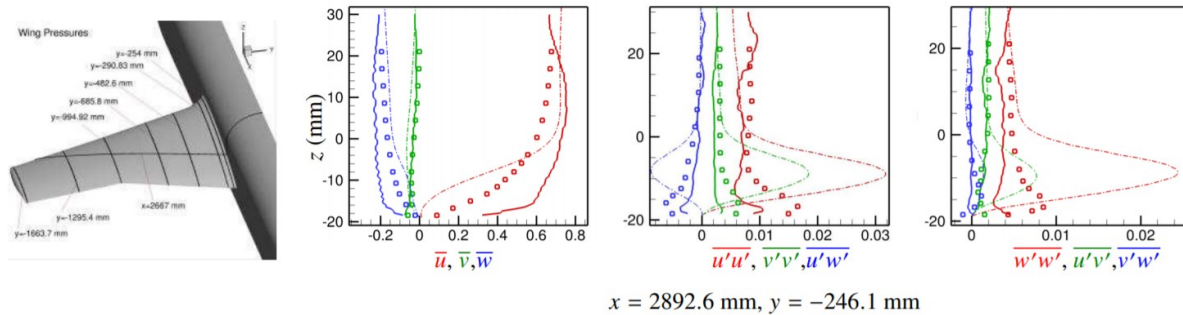


Figure 2-57 Predictions of the velocity and turbulent stress for wing junction flow: solid line (WMLES simulation), dashed-dot line (FUN3D RANS), symbols (experiment) [77].

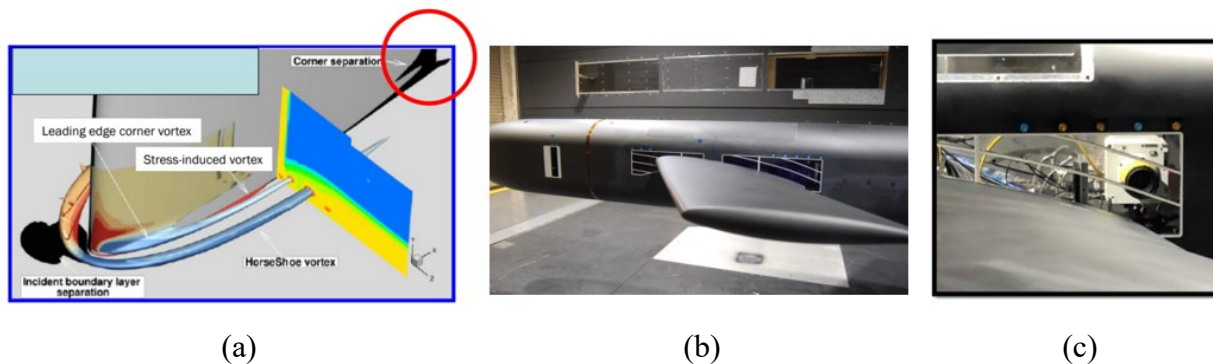


Figure 2-58 (a) Generic organization of a junction flow (b) Photograph of the NASA Junction Flow model (c) Photograph of the PIV system installed at the trailing edge of the JF model [79].

For the CFD validation experiment, infrared images were first used to determine the location of boundary-layer transitions. As shown in Fig. 2-59, for the baseline case, the darker tones near and along the length of the wing leading edges indicates that laminar flow prevails in this region. When the angle of attack increases, transition happens and is located very close to the wing leading edge in the region outboard of the wing planform break. Next, oil flow visualization was used to display the surface topology of the corner flow separation, shown in Fig. 2-60. The

remaining measurement: flow field measurements from LDV probes, and unsteady pressure measurements with pressure transducers will be discussed in the next section [78][80].

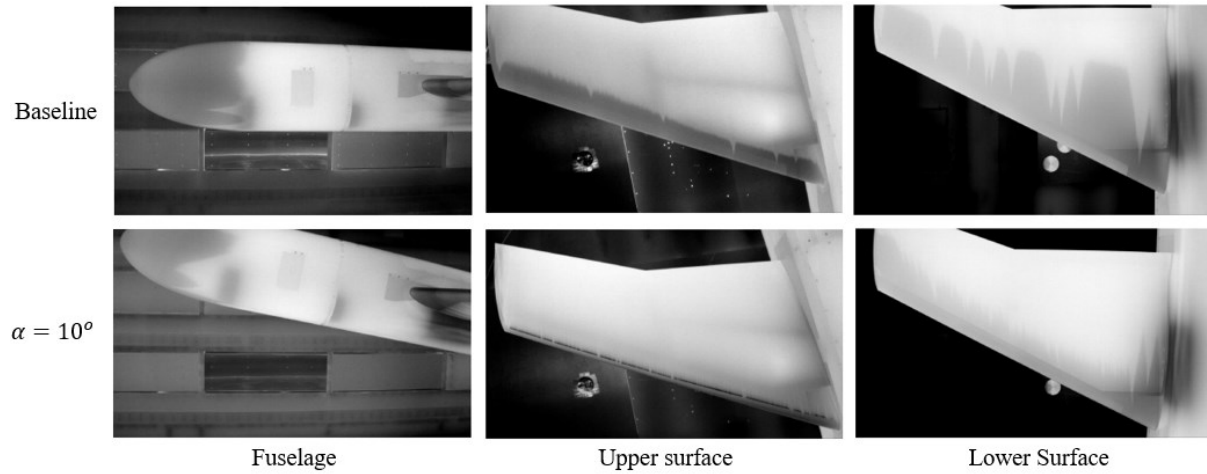


Figure 2-59 Infrared images of the fuselage and of the upper and lower wing surface [78].

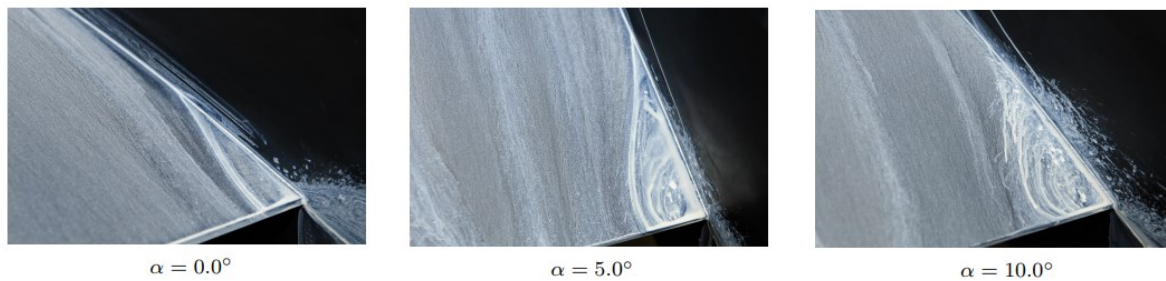


Figure 2-60 Oil flow visualizations in the trailing edge corner [78].

Other interest areas in applying flow visualization techniques are the flap and slats due to aeroacoustic reasons. Noise from wing leading edges and trailing edges are from the unsteady flow within the slat cove region, oscillations in flap cove region and vortex interaction with flap surfaces. Applying flow visualization techniques allows us to study the flow structure in this area. For instance, Kamliya Jawahar et al. [81], studied the effects of slat cove fillers using PIV in Fig. 2-61: vortices appear in the slat cove region and the shear layer impinges at the very tip of the slat trailing edge for conventional leading edges. The use of slat cove fillers eliminates the large vortices in the slat cove region thus reduces the noise. Yet, same flow visualization techniques

may be applied to the wing trailing edge area to study the mechanism of flap noise. More details are provided in Chap. 2.14.

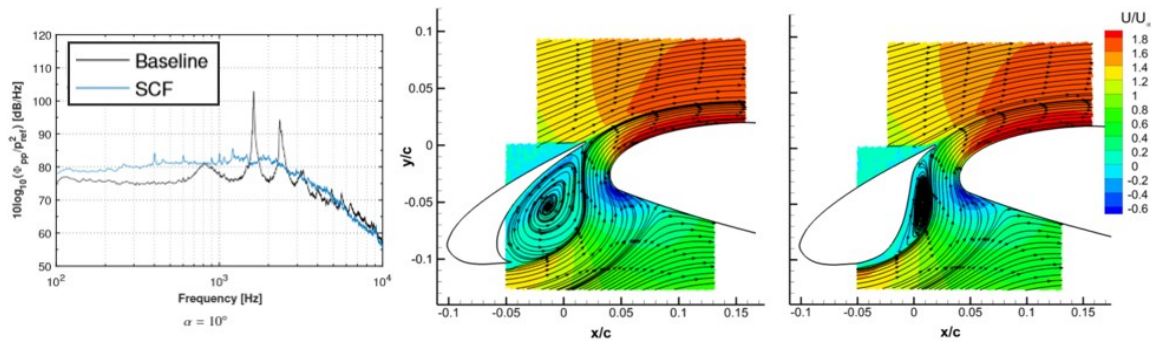


Figure 2-61 Aeroacoustic results for the effect of slat cove fillers (SCF) and PIV flow visualization around slat region for the cases w/ and w/o slat cove fillers [81].

Wakes are the vortices produced by the aircraft. These long lift-time wakes can endanger any following aircrafts. Hence, tracking vortices near the ground and at high altitude is necessary for predicting the decay of vortices and to improve safety standards. However, wake measurements in the far-field have been a problem due to the limitation of data collection devices and wind tunnel sizes. Despite that, near-field measurements could still be done in wind tunnels using PIV techniques to track the trajectory of the wake. Chapter 2.16 provides some examples of using PIV to track wakes and more information about Wake Studies. Another flow visualization method to measure off-surface flow is using Femtosecond Laser Electronic Excitation Tagging (FLEET). FLEET is a new technique that focuses a femtosecond laser pulse to dissociate/ionize molecular nitrogen in the flow of the interest region. While the gases flow downstream, the camera can capture the emitted photons and provide a sequential image. Reese, et al. [82] applied this technique to the wake region behind the wing of the NASA CRM, shown in Fig. 2-62. Fig. 2-62 shows the data obtained from the FLEET signal and the measurement location, it is obvious that the structure of the wake region is shown on the velocity map.

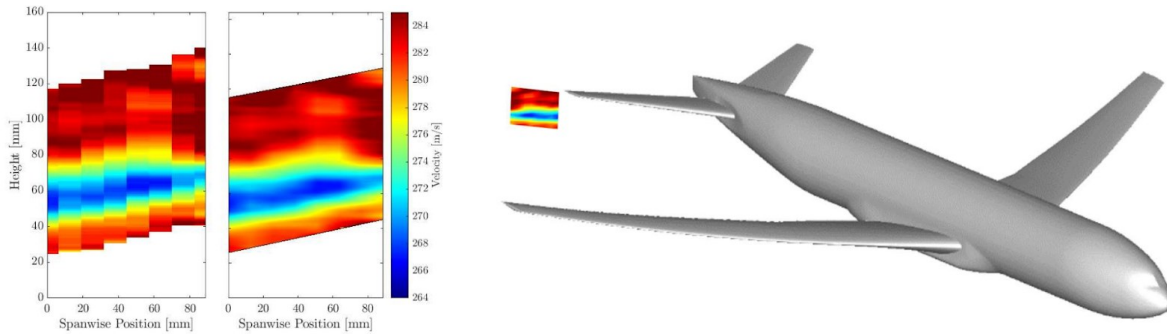


Figure 2-62 (a) Results from the FLEET signal scan (left: original velocity map, right: interpolated velocity map) (b) Measurement region [82].

The last phenomenon is wing flutter. Wing flutter is a dynamic instability occurring in flight, which leads to unstable oscillation on aircraft wings. Gkiolas, and Mathioulakis [83] analyzed the flow field of an airfoil at a simulated flutter condition (an oscillating wing) using 2D PIV techniques, the results are shown in Fig. 2-63 and Fig. 2-64. During upstroke, when the angle increases, the separation point moves upstream to the middle of the chord, the separated flow region expands, and the velocity at the leading edge increases. In addition, the external flow follows the contour of the airfoil, remaining high lift and low drag in this case. For the downstroke case, a shear layer which is detached from the wing is formed at the leading-edge area, forcing the external flow not to follow the contour of the airfoil that increases the drag. Moreover, a re-attached flow is observed when the angle is reduced.

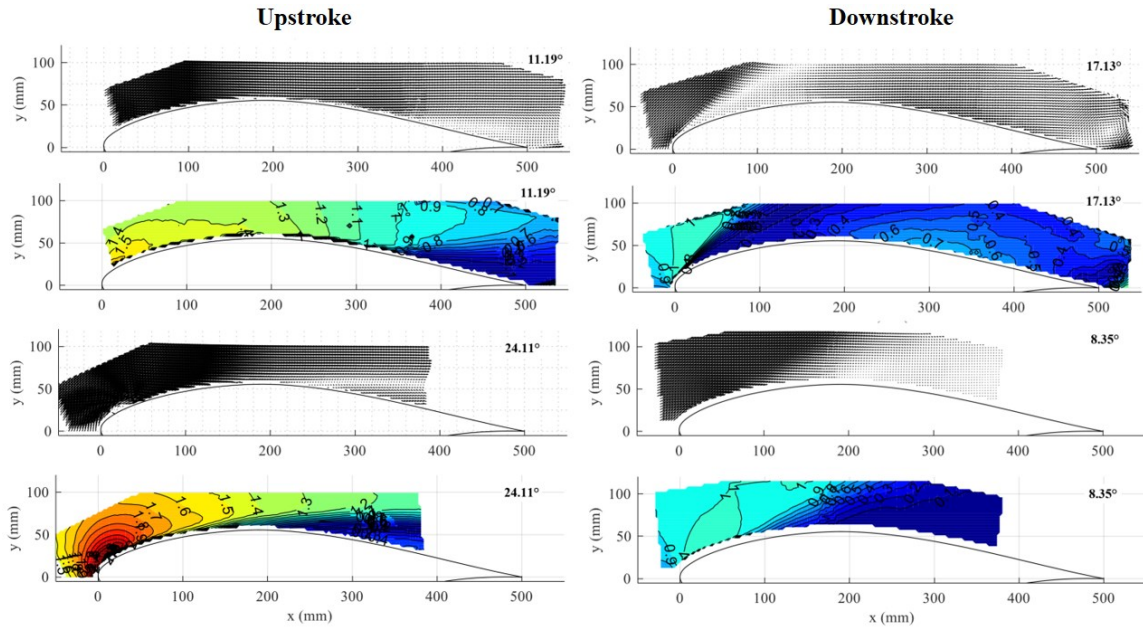


Figure 2-63 PIV results (velocity vectors and magnitude contours) of airfoil in simulated flutter condition [83].

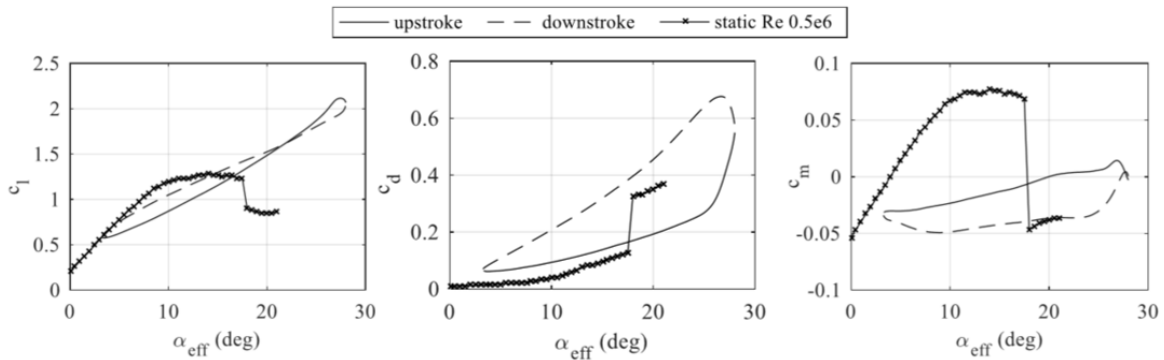


Figure 2-64 Aerodynamic results of airfoil in simulated flutter condition [83].

Zhu et al. [84] to study the trailing edge flow physics of an airfoil in an oscillating freestream uses another technique: Background Oriented Schlieren (BOS). BOS method is an optical density visualization technique that can investigate the flow without disturbing particles using the deflection of light rays caused by changes in the local index of refraction. Fig. 2-65 (b) shows the vertical displacement field of an airfoil at $\alpha = 4^\circ$ under steady freestream flow condition. The vertical displacement upper and lower the airfoil surface indicates the viscous

boundary layers. The red and magenta color bands indicate the airfoil wakes that continues downstream. In the steady case, adverse pressure gradient on the upper surface causes higher momentum to exist on the bottom surface, and the imbalance of low-momentum fluid deflects the wake centerline. For the unsteady case in Fig. 2-66, the deflection of the wake centerline first flattens out at phases 0 to 90, then it grows in a vertical direction, and curves up and deflects abruptly to follow a more horizontal direction downstream.

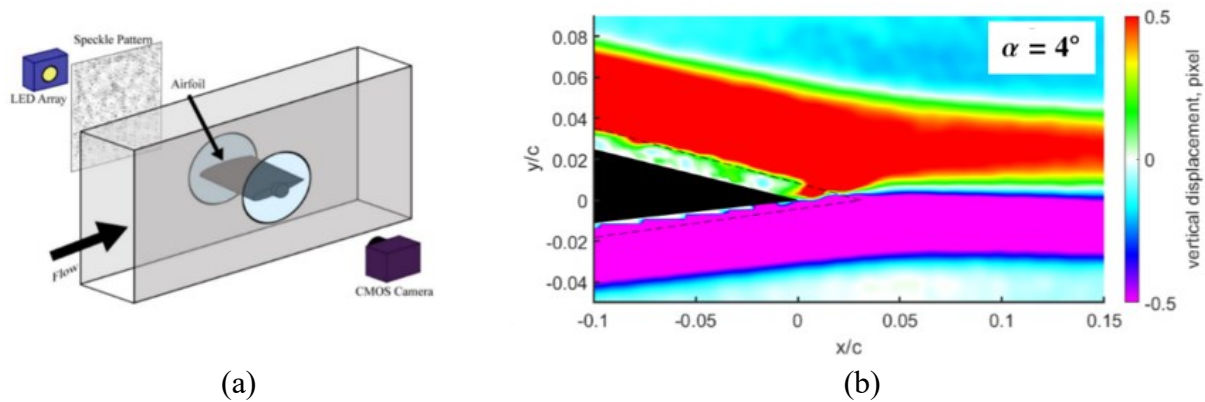


Figure 2-65 (a) Schematic of the BOS measurement setup (b) vertical displacement field of an airfoil at $\alpha = 4^\circ$ under steady freestream flow condition [84].

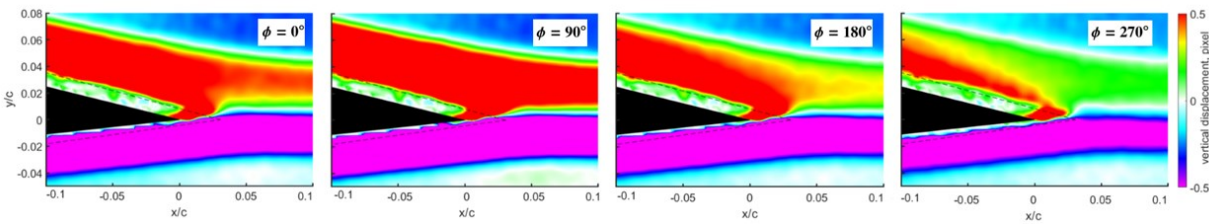


Figure 2-66 Displacement field around airfoil trailing edge under unsteady flow condition at $\alpha = 4^\circ$ [84].

In general, this chapter mentioned the commonly used technologies to study aerodynamics on aircraft: tufts, oil flow, china clay, TSP, PSP, OFI, infrared imaging for on-body surface display and PIV, smoke wire for off-body flow measurement. However, the uncertainties, errors, limitations, and operating considerations have not yet been discussed. The pros and cons of each

method needed to be studied in order to determine the feasibility to apply each method on the UW HL-CRM. More details of different flow visualization techniques are shown in [72].

2.7 BOUNDARY LAYER STUDIES

Airfoil performance within the transition process has aroused people's interest in modern subsonic aerodynamics. Boundary layer on a low Reynolds number airfoil often remains laminar well. However, with an increasing angle of attack during landing and takeoff, laminar separation happens with transition to turbulence occurring in the separated shear layer, shown in Fig. 2-67. These transition and separation regions will influence the performance of the aircraft, reducing the lift generated by the airfoil [85]. Hence, the interest in boundary layer studies relies on using various techniques to find out the separation region on the airfoil, the effect of flow control techniques, investigation in the mechanisms of transition, and juncture flow and to provide an experimental database for CFD.

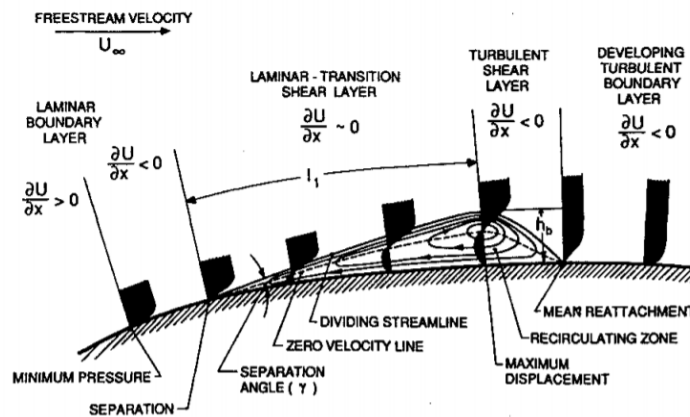


Figure 2-67 Main features of the boundary layer on a low Reynolds number airfoil [85].

The high lift program at JAXA analyzed the flow field of a half-span testing model with high-lift devices at stall condition using surface pressure and flow visualization measurements.

Pressure tapes and Pressure Sensitive Paint were used to measure the surface pressure. Pressure taps were located along five cross sections on the main wing, slat, and flap, shown in Fig. 2-68. At the leading edge, all distributions except cross section (d) showed a very high suction peak. For cross section (d), the nacelle-pylon causes separation and vortex at the downstream flow that makes the high suction pressure drop after the peak. By comparing cross section (a) and others, the effect of flaps is shown around the trailing edge: the lift force is produced by efficient suction pressure due to the flaps. Moreover, the pressure gradient is nearly zero at $X/C = 4.5 \sim 5$ for cross section (a), this is designed by aircraft manufacturers: hoping that stall could occur near the wing root first. The above results could only show pressure distributions on certain locations, due to installation and cost limits of pressure tapes. Hence, PSP technique is applied to present a detailed pressure map on the wing, indicating the actual separation area on the wing, shown in Fig. 2-69 [11]. (Note that PSP can only measure pressure distributions on surfaces, it can't measure transition; TSP can be used to measure transition, for more detail, see Chap. 2.06)

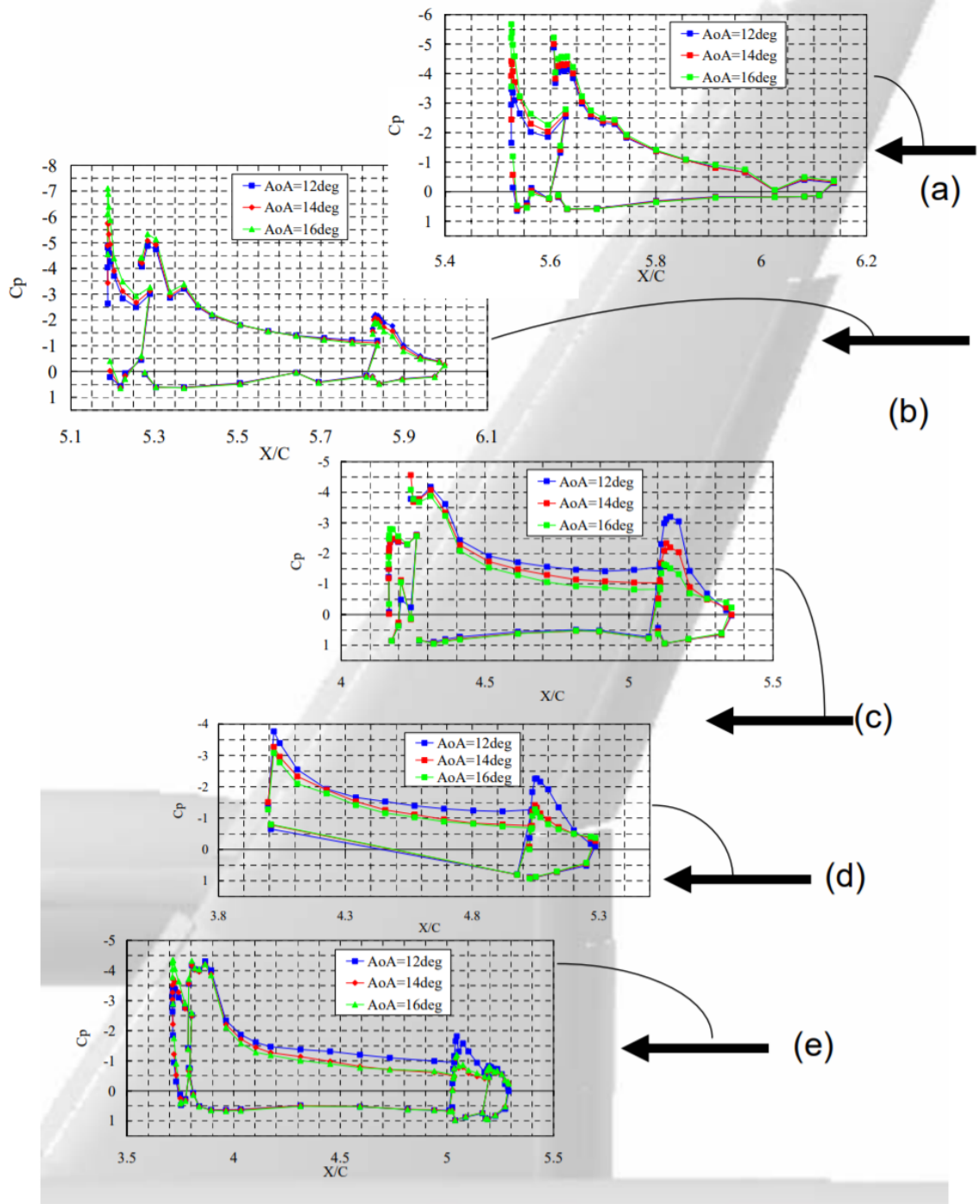


Figure 2-68 Surface pressure distribution with pressure tapes at 60m/s and $\alpha = 12^\circ, 14^\circ, 16^\circ$ [11].

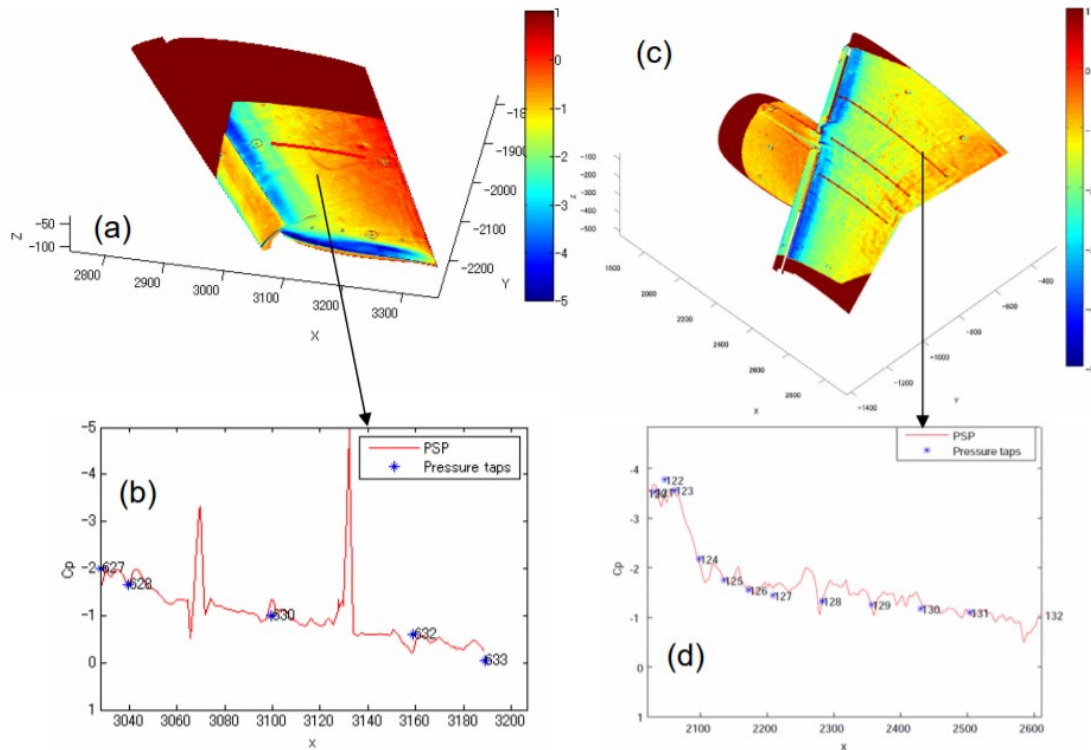


Figure 2-69 Surface pressure distribution on the wing using PSP at 60m/s and $\alpha = 10^\circ$ [11].

On the other hand, flow visualization investigation using tufts, oil flow, china clay and PIV provided more detail of the flow field. In Fig. 2-70(a), tuft visualization showed that flow separation partially occurred at the downstream region of the nacelle. Fig. 2-70(b) exposed the transition on the main wing and slat using china clay method. Moreover, oil flow visualization in Fig. 2-70(c) shows that there were lots of vortices and separations around the pylon and the leading edge of the main wing, indicating that this area must be the key for the flow field to reveal the stall at high angle of attack. Furthermore, in Fig. 2-71, wing tip vortex from the outboard flap, separation region at the downstream of nacelle/pylon and on the upper surface of the flap were observed from PIV. These experimental results depicted that the stall is caused by flow separation on the wing, which is affected by the nacelle and pylon. The above measurement techniques allow JAXA to analyze the boundary layer conditions, and to understand the mechanism of the stall [11].

A solution is then provided to eliminate the separation on the wing, more details of the solution is presented in Chapter 2.03.

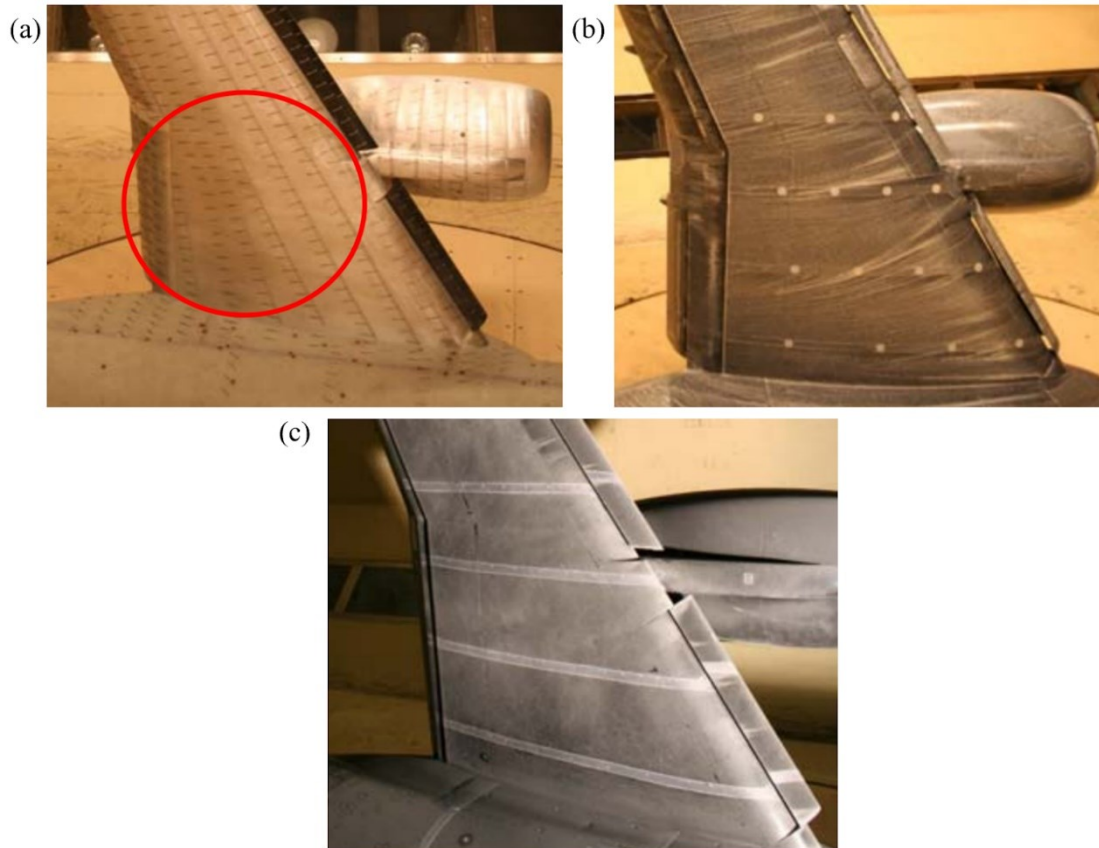


Figure 2-70 Surface flow visualization using (a) tuft (b) china clay (c) oil flow at 60m/s and $\alpha = 10$ [11].

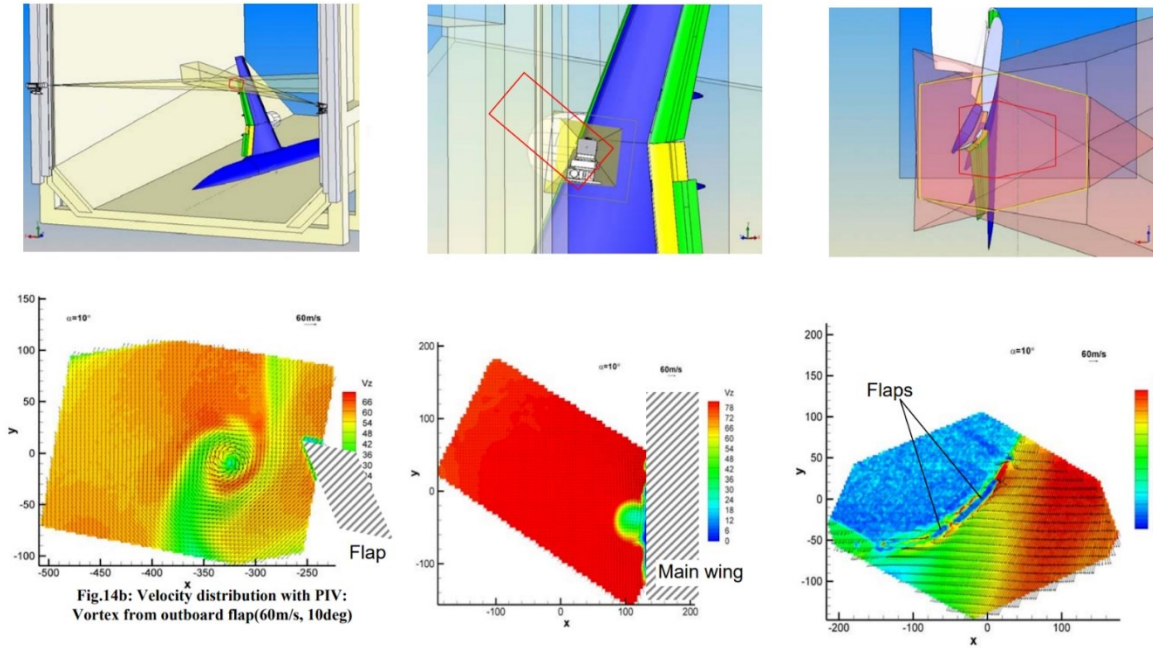


Figure 2-71 Velocity distributions with PIV [11].

In addition, the mechanism of transition is also investigated accompanied by the study of separation. The project EUROLIFT studies the transition phenomenon by analyzing the pressure distributions, hot film signals and infrared images of a swept model equipped with a slat and a flap in three different conditions. Fig. 2-72 shows results of the hot film signals located around the slat leading edge, and the leading edge contamination Reynolds numbers R computed from experimental C_p distributions. In the $Re = 4.5 \times 10^6$ case, the signal remains laminar, but in the $Re = 7.8 \times 10^6$ case, the flow is fully turbulent. By comparing the results from the hot film signals and R , the domain of leading edge contamination can be confirmed: when $\bar{R} > 250$ and $Re = 6 \times 10^6$, the contamination process makes the slat boundary layer turbulent. The infrared visualization measurements allow us to note the limit between the laminar part and the turbulent part, which is the presence of a separation bubble. For instance, in Fig. 2-73, at $\alpha = 10$, on the infrared image, there is a free transition on the slat upper side, but when the AOA increases, a

laminar bubble appears [4]. The above examples show the transition boundary layer studies done in the EUROLIFT project.

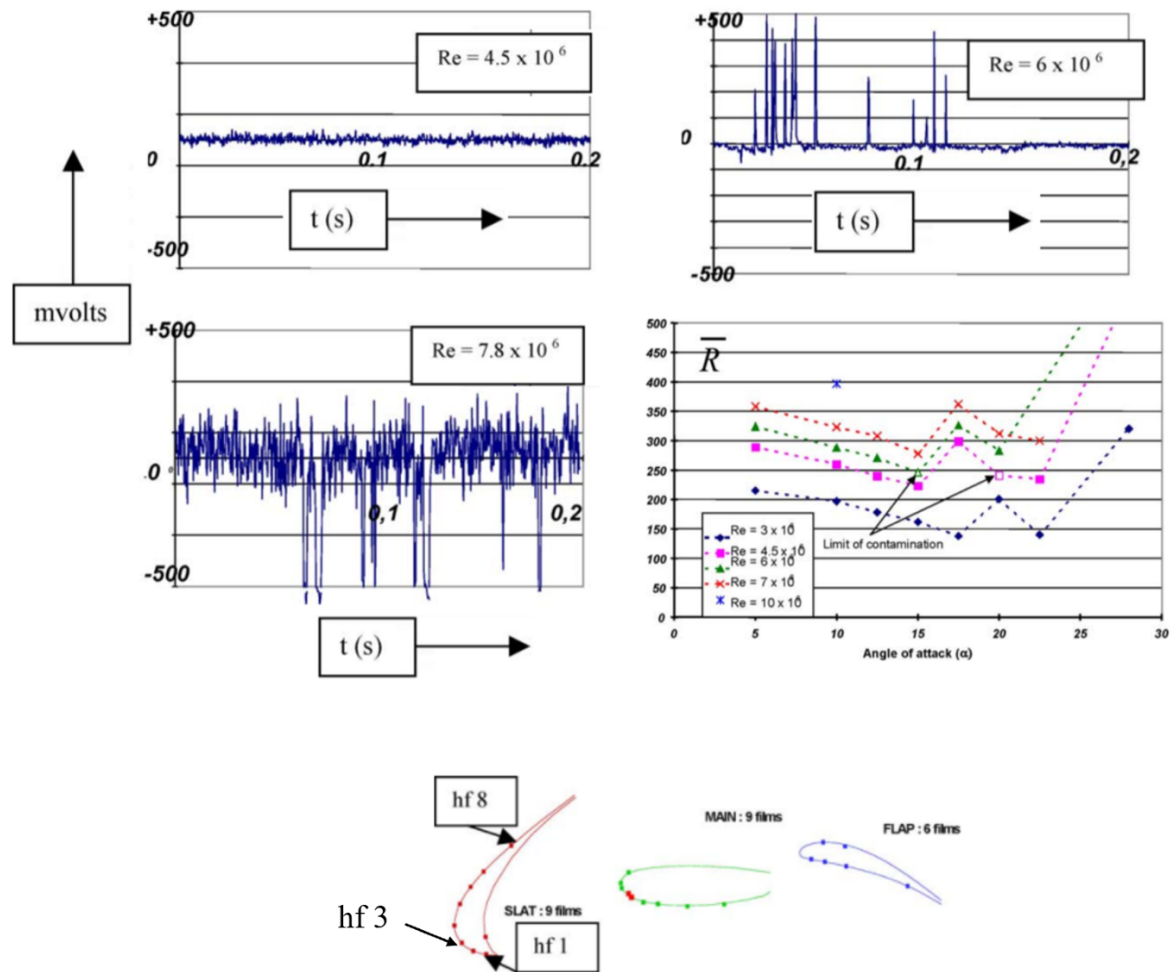


Figure 2-72 Results of the hot film signals (hf3) and $\bar{R}$ [4].

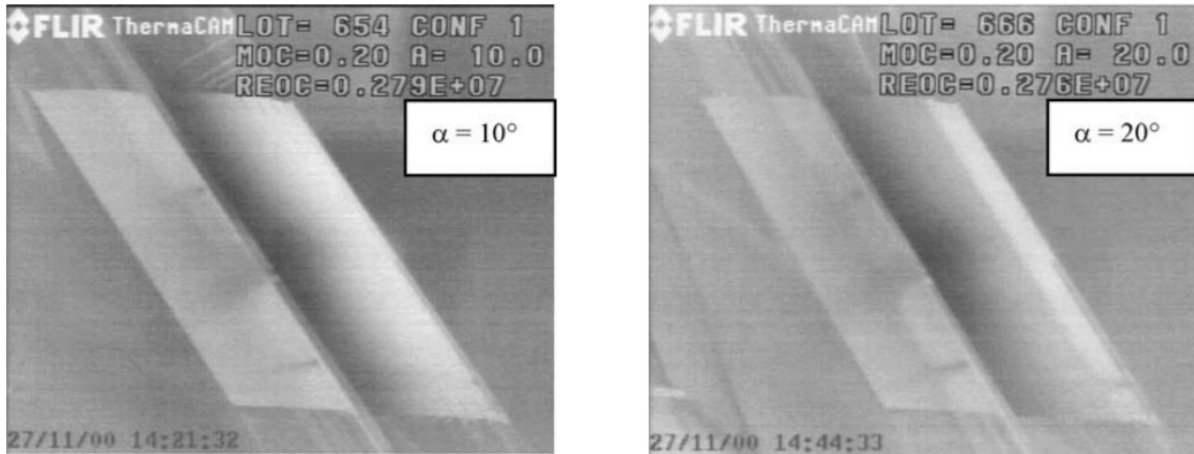


Figure 2-73 Infrared visualization results on an airfoil [4].

Flow visualization techniques can be used to study the detail flow structure in the transition boundary layer. Kirk and Yarusevych [86] investigated the formation and development of the roll-up process in separation bubbles on an airfoil by placing smoke wire upstream of the airfoil and within the span of the airfoil. Four angle of attack $\alpha = 0^\circ, 5^\circ, 8^\circ, 10^\circ$, were conducted in the experiment, and the chord Reynolds number is $Re = 10^5$. Placing smoke wire upstream of the airfoil (Fig. 2-74(a)) provides a general overview of the flow development around the airfoil: the streamline remains closely to the surface until the trailing edge where distinct unsteadiness develops. Smoke wire within the span of the airfoil (Fig. 2-74 (b)) provides a more detailed view of the flow field in the boundary layer: the transition of vortex cores to turbulence and the breakdown of vortices. Fig. 2-74 (e) displays a sequence of the vortex dynamics in the separated shear layer with three consecutive shear layer vortices V1, V2, and V3. The dash line of the core of these structures illustrate that the shear layers roll up into vortices and then develop into amplified disturbances. Moreover, the mean separation, transition, reattachment, and mean shear layer roll-up location obtained from the surface pressure distributions (Fig. 2-74 (b)) and from the side-view visualization (Fig. 2-74 (e)) at each angle of attack is presented in Fig. 2-74 (c). This

plot indicates that the roll-up process happens just before transition, and the roll-up vortices enhance momentum exchange that leads to flow reattachment. More details related to transition studies is shown in Chapter 2.17.

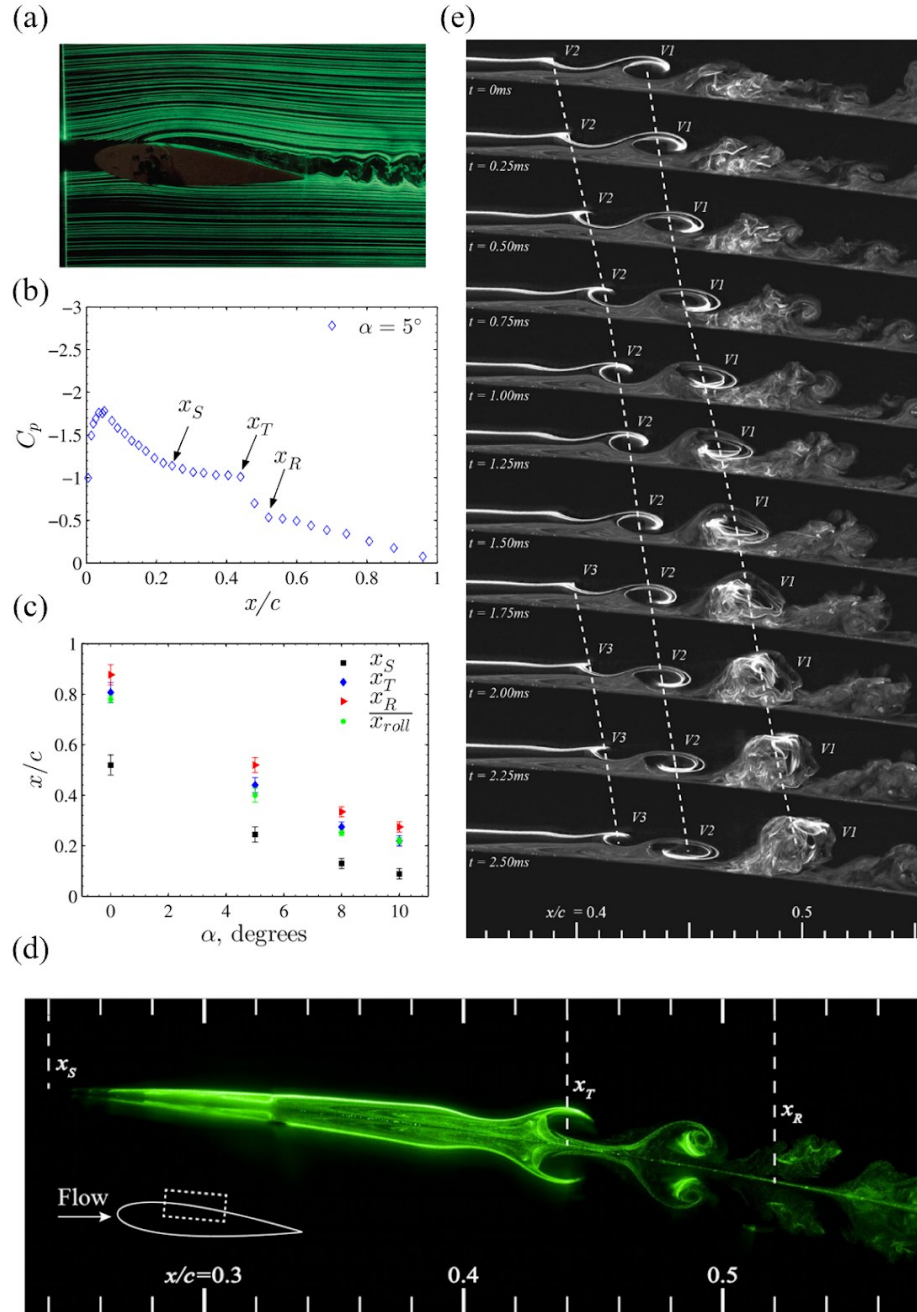


Figure 2-74 (a) Smoke visualization of smoke-wire at the upstream (b) Surface pressure distributions for $\alpha = 5^\circ$ (c) Separation, transition, reattachment, and roll-up locations (d) Side view visualization of vortex shedding (e) Sequence visualization of vortex shedding [86].

As mentioned in the previous chapter, juncture flows are complex flows that formed around a wing-fuselage or wing-pylon intersection area when a boundary layer encounters an obstacle.

NASA developed a juncture flow (JF) model to study the aspects of the wing-body juncture flow physics and to acquire high-quality flow data of the flow field. A validation experiment using various measurement techniques like Laser Doppler Velocimetry (LDV), infrared imaging, oil-flow visualization, pressure ports and transducers was done by Kegerise, et al. [80] Since, the results from infrared imaging and oil-flow visualization are mentioned in the previous chapter (Chap. 20.6), this chapter focuses on the results from the LDV and unsteady pressure measurements. LDV probes were located inside the model at the leading edge and trailing edge position, providing measurements of the velocity field in all three components in the wing leading edge and trailing edge (Fig. 2-75 (a)). Unsteady pressure was measured by pressure transducers clustered in the corner-flow region (Fig. 2-75 (b)) near the wing trailing edge, aiming to display the dynamic nature of the surface pressure field. Fig. 2-77 shows the mean velocity profiles measured near the wing leading edge (Fig. 2-76 (a)), the results indicate that the fuselage boundary layer is still attached and the w profiles indicates that the model is under a positive pitch angle. Fig. 2-78 shows the Reynolds shear-stress profiles in the corner-flow region (Fig. 2-76(b)). For the profile near the fuselage surface where there's separation, the component $u' w'$ displays a peak near the inflection point in the u' profile. For the flow outside the separation region, the inflection point then shifts toward the wall. The unsteady pressure results in Fig. 2-79 shows that broadband RMS pressure meets the oil-flow measurements: the pressure from the sensors in the separation region increases as the separation flow region increases.



Figure 2-75 (a) Photograph of the LDV system in the junction model (b) Locations of the static pressure ports (red) and pressure transducers (blue) on the model [80].

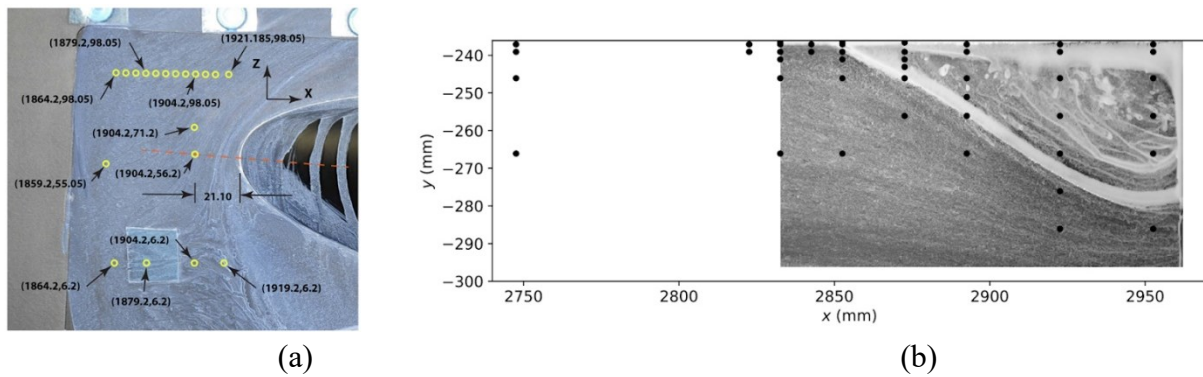


Figure 2-76 Locations of LDV measurements at the leading edge (a) and trailing edge (b) [80].

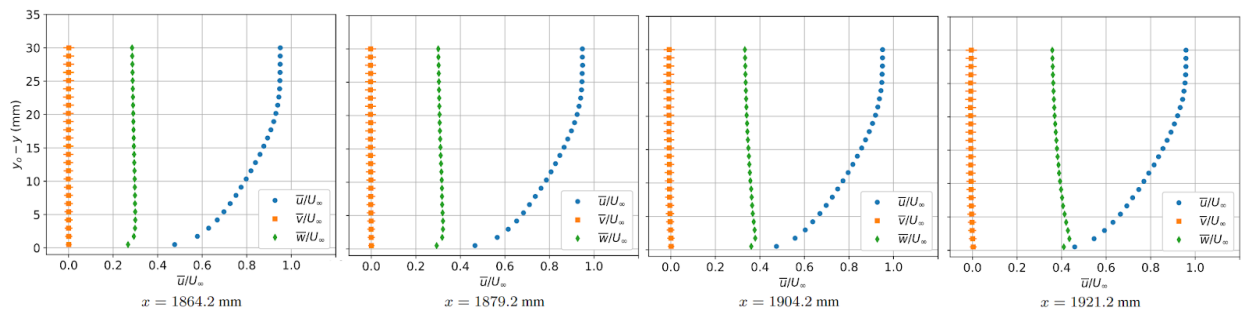


Figure 2-77 Mean-velocity profiles along a line located at $z = 98.05$ mm near the wing leading edge [80].

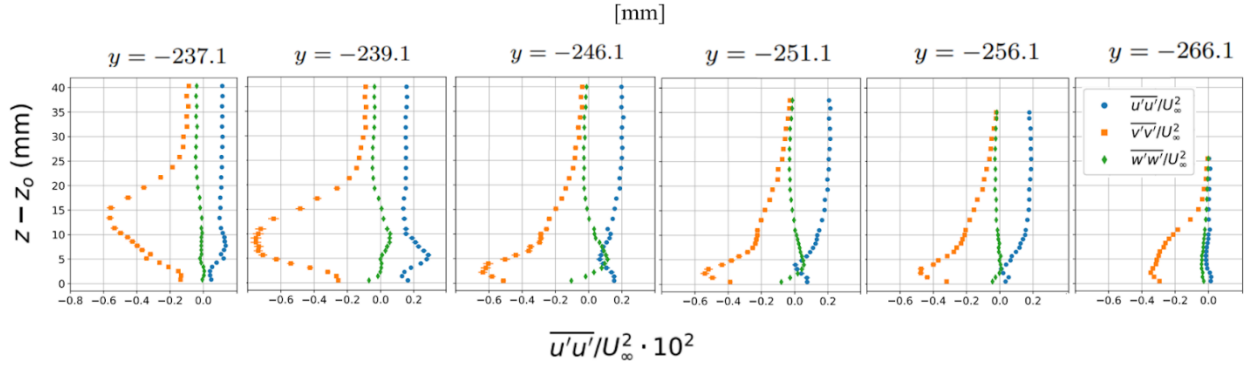


Figure 2-78 Reynolds shear-stress profiles in the corner-flow region of the wing with leading-edge extension at $x = 2892.6\text{mm}$ and $\alpha = 5$ [80].

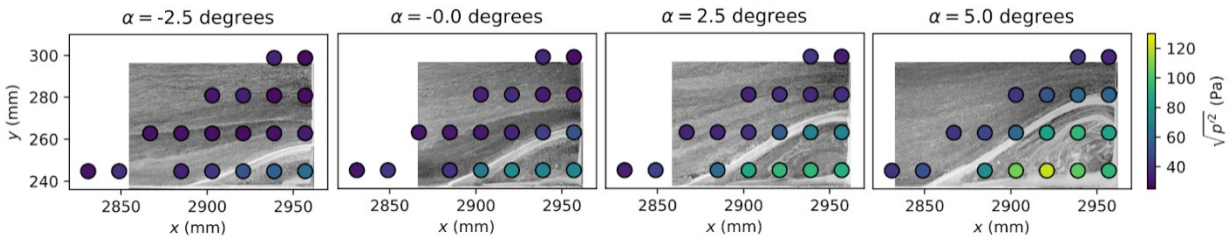


Figure 2-79 Results of the unsteady pressure measurement at the wing trailing edge region [80].

Another interest is the boundary layer affected by active flow control. Gul et al. [87], investigated the effect of synthetic jet actuators on the dynamics of the flow within the boundary layer of a 2D S809 airfoil using Constant Temperature Anemometry (CTA) measurements. Fig. 2-80 and Fig. 2-81 show the mean and fluctuation velocity at the traversed chordwise locations. For the jet off case, it is shown that at the 48.1% chord location, the level of fluctuation velocity is higher and separation happens at this point. After the separation point, fluctuating velocity level increases and a dead air region appears near the wall. Downstream the 65% chord location, the fluctuation velocities are almost constant near the wall, indicating that the flow is reattached to the wall, forming a laminar separation bubble. When the synthetic actuator is on, it is shown that the recirculation zone and the laminar separation bubble are eliminated by the periodic excitation from the synthetic jet actuators.

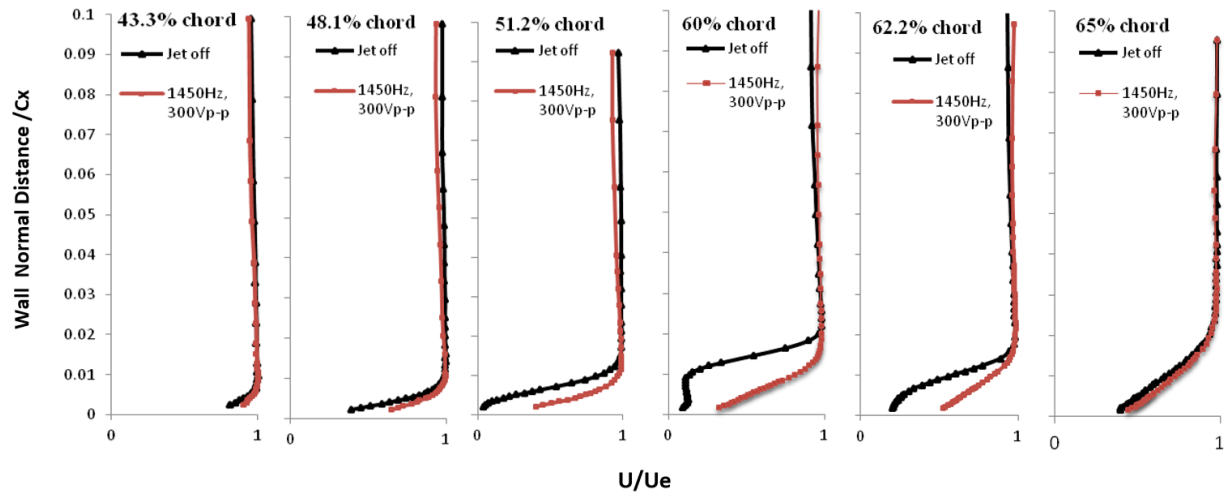


Figure 2-80 Boundary layer profiles of mean streamwise velocity obtained from CTA measurements at different chord stations [87].

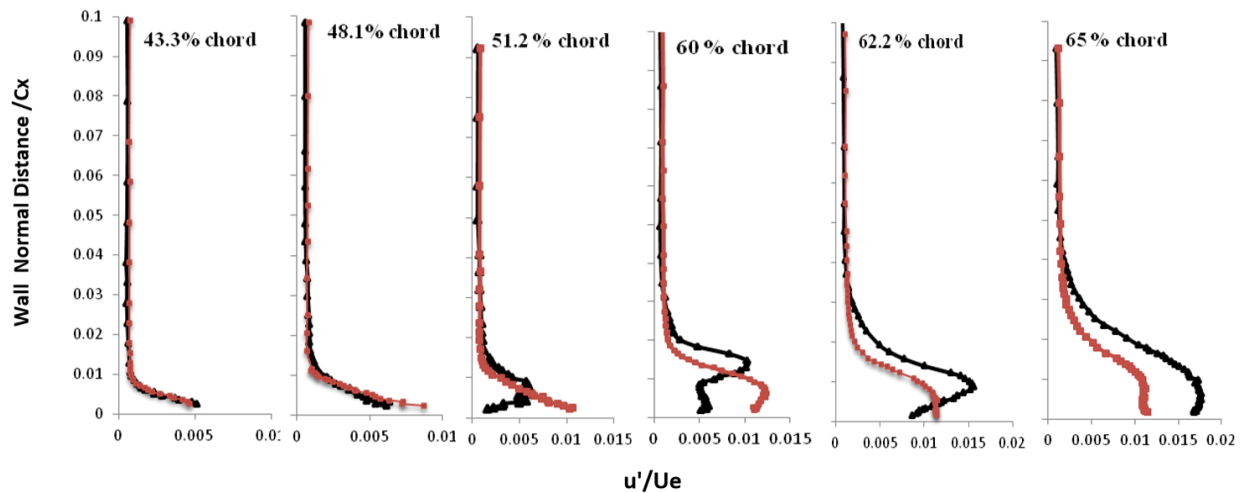


Figure 2-81 Boundary layer profiles of fluctuation velocity obtained from CTA measurements at different chord stations [87].

In conclusion, the interest in boundary layer studies focused on using various techniques to find out the separation region on the airfoil, using flow control techniques to delay the transition and separation, investigate the mechanisms of transition, and to provide an experimental database for CFD to predict transition and separation accurately.

2.8 INTERCHANGEABLE PRESSURE-INSTRUMENTED WINGTIPS

Winglets are devices installed at the tip of aircraft wings that could improve the efficiency of fixed wing aircraft by reducing the induced drag during landing and takeoffs. Winglets could reduce the drag caused by wingtip vortices and increase the lift generated at the wingtip. (Fig. 2-82) Induced drags are drags caused by wingtip vortices: the flow at the wingtip lower surface will curl upwards around the wingtip to the upper surface due to imbalanced pressure, contributing to the low pressure region on the upper wing surface. The role of the wingtip devices is to decrease the strength of the vortices and lead them to appear further downstream by providing an upward-oriented surface. A schematic of the mechanism of wingtips are shown in Fig. 2-83 [89][90]. Different types of wingtips designs that are used nowadays such as endplate winglet, whitecomb winglet, blended winglet, raked, winglet, hoerner winglet, spiroid winglet, bird winglet and multiple winglet are shown in Fig. 2-84. On the other hand, the installation of wingtips will cause additional weight to the aircraft, and also reduce the rolling performance since the increase in span will increase the moment of inertia [92].

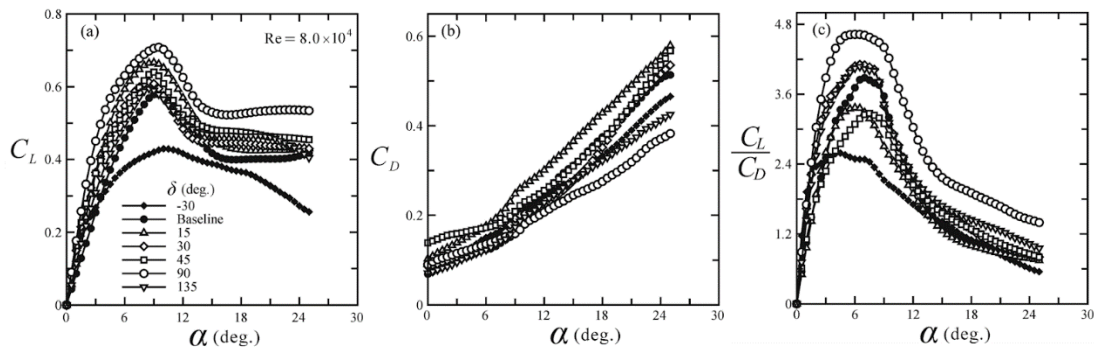


Figure 2-82 Effect of Winglets with different dihedral [88].

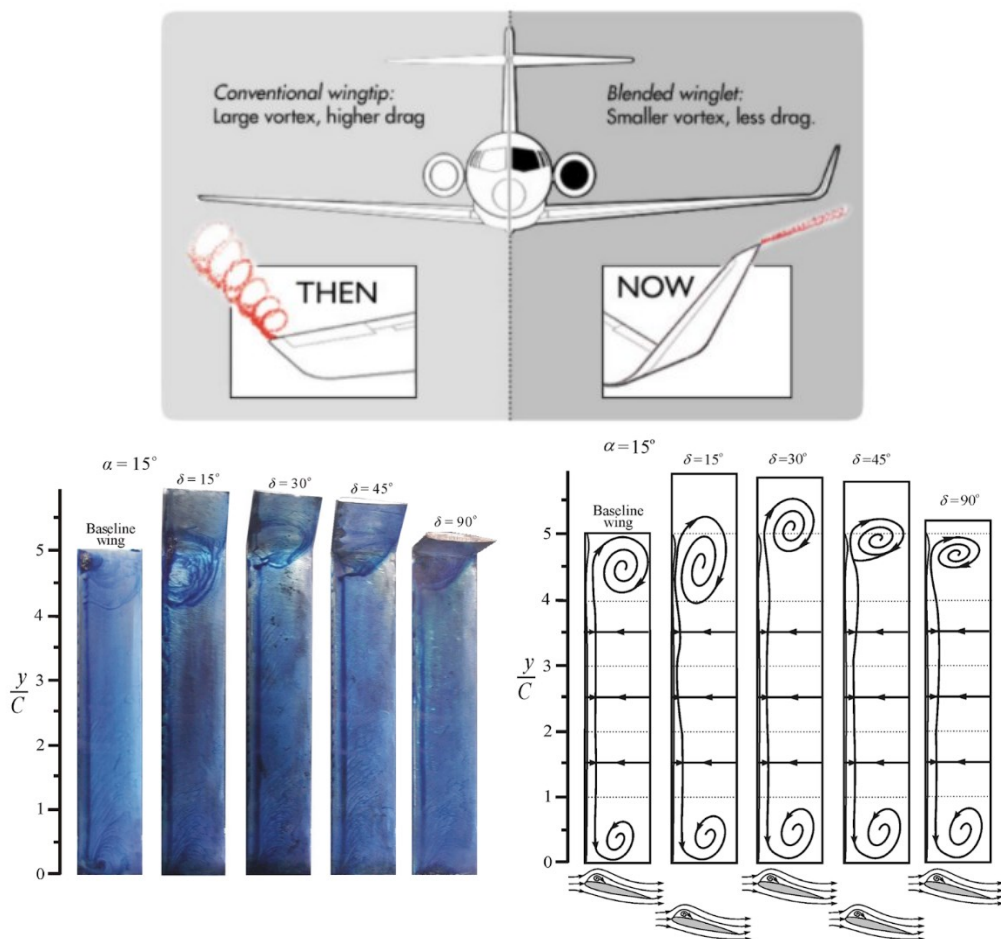
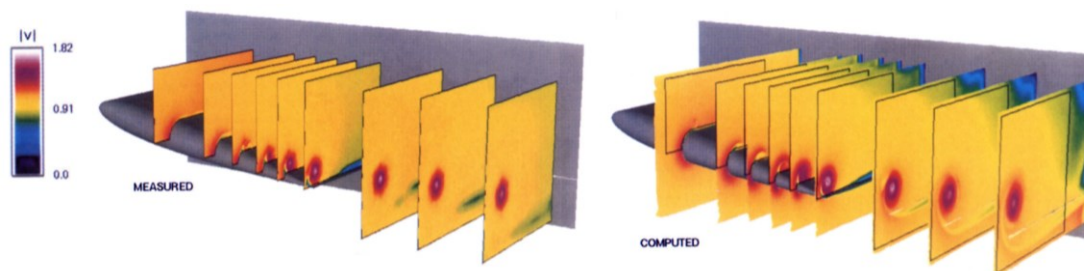


Figure 2-83 Effect of wingtips on wingtip induced wakes [88][89].



Figure 2-84 Configurations of various wingtip devices [89][90][91].

As mentioned above, wingtip wakes are the main reason that causes the induced drag. Despite that the mechanisms of induced drags are confirmed, the industry still lacks the ability to simulate wingtip flow accurately, especially within the vortex core. For instance, Dacles-Mariani et al. [93] ran a simulation of a wingtip vortex flow with a fifth-order compact biased upwind scheme for the advection term, second order scheme for the viscous term, and a modified Baldwin-Barth turbulence model nearly in 1995. Their results (Fig. 2-85) successfully captured the flow structure at the trailing edge, and the pressure distribution on the surface. For the data in the center of the vortex core, the computed core velocity (Fig. 2-86 (a)) agrees with the experimental results well, but the core pressure (Fig. 2-86 (b)) was under predicted by more than 25%.



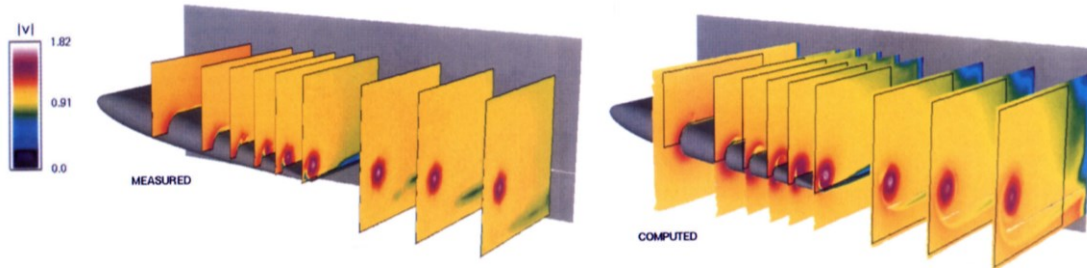


Figure 2-85 Comparison of the velocity magnitude contour of the wingtip vortices and the surface pressure distribution of the wing surface [93].

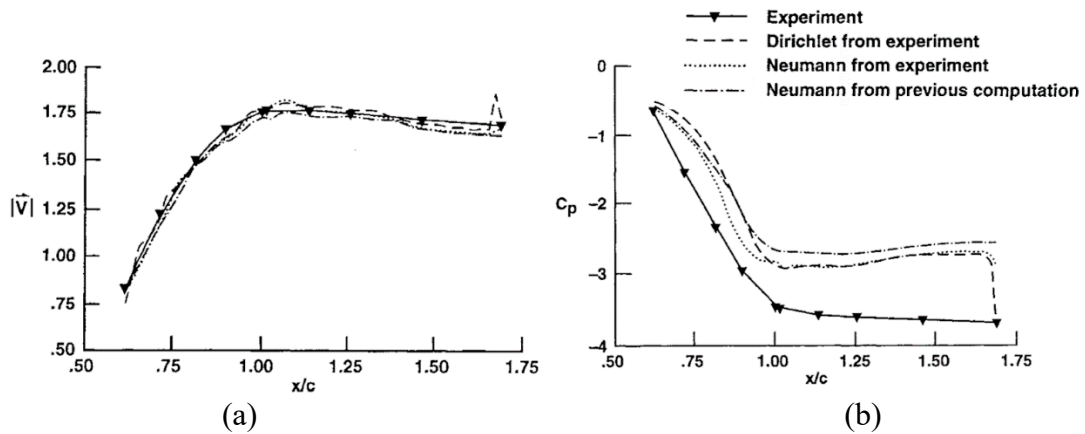


Figure 2-86 (a) Comparison of peak velocity magnitude at vortex core (b) Comparison of static pressure at vortex core [93].

Twenty years later, Lombard [94] performed a large-eddy simulation focus on the wingtip vortices in the near wake using an implicit LES model SVV. In Fig. 2-87, the computed axial velocity provides nice trends compared to the experimental data. The prediction of the C_p in the vortex core improved a bit by having an error less than 10% over the wing surface, but the error increased at the downstream of the trailing edge, reaching 30%. Fig. 2-88 shows the location of the simulated vortex core compared with an experimental measurement (Fig. 2-88(a)) and a previous LES result (Fig. 2-88 (c)). The results show that the location of the computed vortex core is 20% lower than others, but the flow structure in the region $0.03 < y/c < 0.06$ is more closer to the experimental results compared to the previous LES simulation. Apart from this, the flow in the wing-wingtip concave corner (Fig. 2-89) is confused: computational results showed that a high

velocity and pressure region in the concave corner might lead to flow separation on upper winglets, but there's no such effect for a dropping winglet [95].

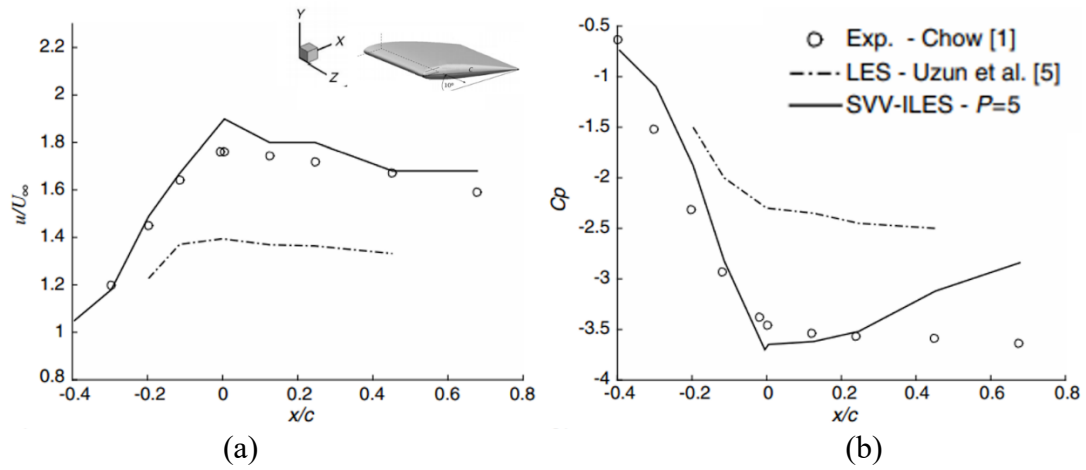


Figure 2-87 Comparison of the (a) axial velocity (b) C_p in the vortex core [94].

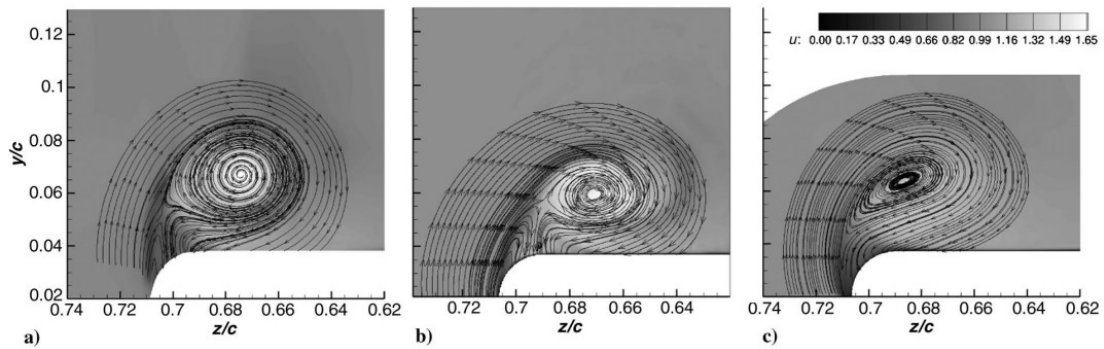


Figure 2-88 Comparison of vortex core location (a) from experiment (b) from SVV simulation (c) from previous LES model [94].

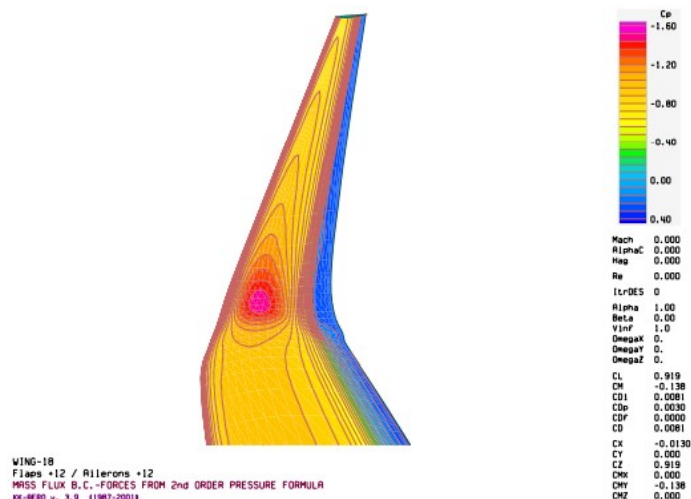


Figure 2-89 Pressure distributions in the wing-winglet concave region [95].

Furthermore, wingtip vortices not only influence the drag calculation of wingtips, but also influence wake turbulence. Wake turbulence is a disturbance that forms behind an aircraft and is very hazardous to aircrafts in the takeoff or landing phase. The composition of wake turbulence is various initial circulations from wing components, including wingtip vortices, but it has not been demonstrated how wingtip vortices interact with other small vortices and distribute in the vortex sheet showed up in the far-field characteristics [96]. Besides that, another interesting phenomenon is that during approach and landing, the primary vortex comes from the outboard edge of the outboard flap and not from the wingtip. However, in other phase of flights, the main vortices detach from winglet tips not from the flap [97].

Folding wingtips might be a way to increase the rolling behavior of aircrafts. Although folding wingtips have been used on military aircrafts in the past, and the latest Boeing 777-x for saving space on airports or aircraft carriers. Since the utilization of folding wingtips are only for saving space on the ground, questions have been raised whether folding wingtips can be used in-flight for providing aerodynamic and aeroelastic benefits. Healy et al. [92] studied a baseline wing with flared folding wingtips in the wind tunnel. The results in Fig. 9 not only demonstrate the

reduction of rolling behavior due to larger spans but also shows that folded wingtips could recover around 50-75% of the mean roll rate achieved.

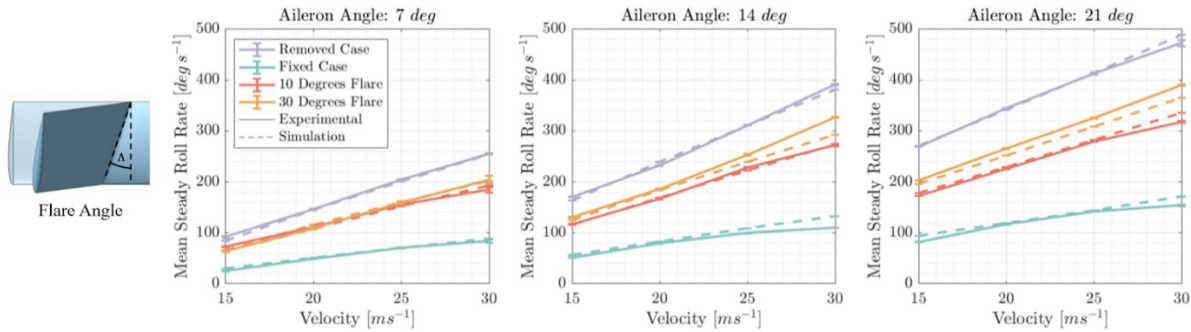


Figure 2-90 Mean steady roll rate at different angle of attacks [92].

In addition, Mills, and Ajaj [98], placed a small aircraft model in the wind tunnel to give a more detailed study of the rolling performance (Fig. 2-91). Both symmetric deflection and asymmetric deflections of the wingtips were tested. In Fig. 2-92, the aircraft model is placed in a static condition with no sideslip, the results show that the rolling moment coefficient of the wingtips are the greatest when the starboard dihedral is at negative 30 degrees for the cases $\alpha > 0$. Moreover, the maximum rolling coefficient increases as the angle of attack increases, meaning that the ability for winglets to increase the lift and reduce the moment is relied on the angle of attack. Fig. 2-93 shows the results of the aircraft model during a sideslip scenario. The results show similar trends with the no sideslip case. These experiments indicate that asymmetric deflection might be the most effective method.

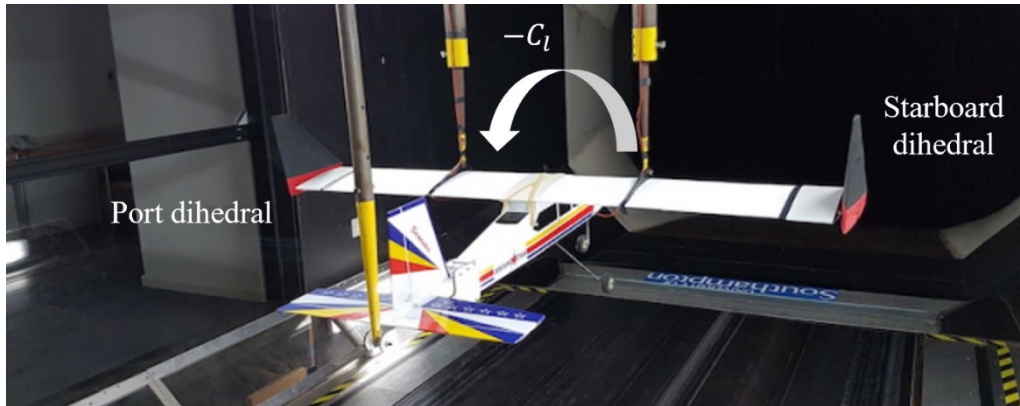


Figure 2-91 Photograph of the test aircraft in the wind tunnel [98].

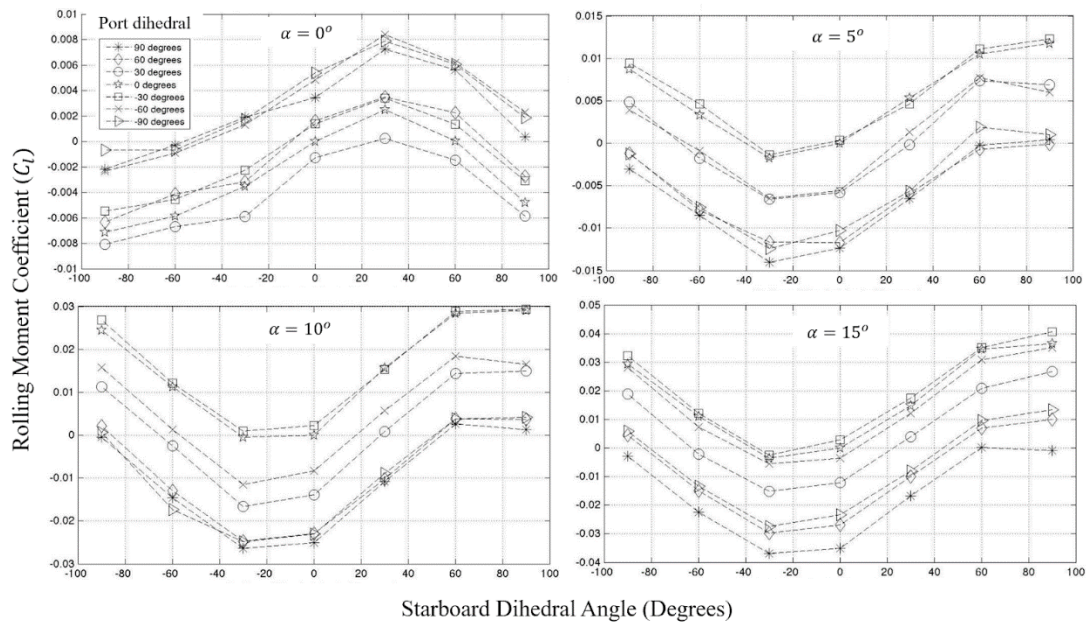


Figure 2-92 Effect of folding wingtips on aircraft stability in straight flight [98].

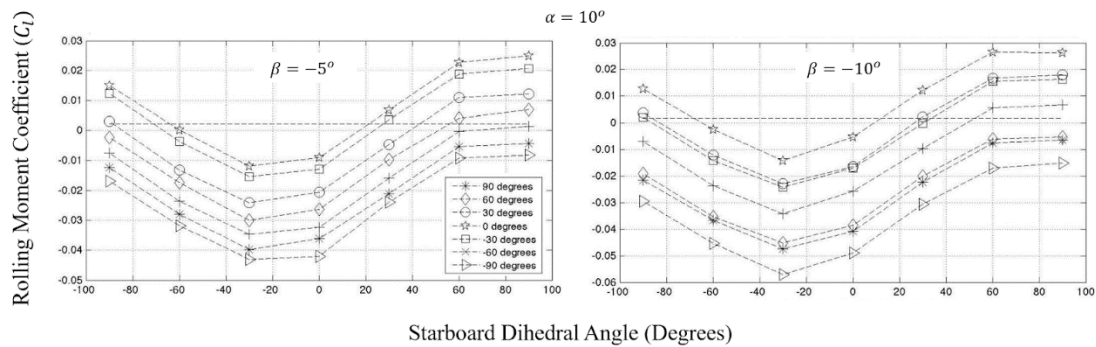


Figure 2-93 Effect of folding wingtips on aircraft stability during sideslip condition [98].

In addition to aerodynamics, aeroelastic effects such as wing deformation and wing flutter should also be considered while evaluating the overall benefits for wingtips. Hantrais-Gervois and Rapin [99] analyzed the structural behavior of the wings with wingtips and concluded that the aerodynamic efficiency and the structural strengthening of the wing deformation with the winglet might cancel or overcome the rigid aerodynamic gains. Fig. 2-94 shows the drag and the wing root bending moment (WRBM) increments of a wing with large wingtip to the reference clean flight shape at cruise and the gains and penalties of rigid and flexible wings with tip fence and large wingtips. Cui and Han [100] investigated the flutter characteristics of a transport wing with wingtips. Flutter is an uncontained vibration condition led by fluid-structure interaction. Results have shown that additional wingtips caused a degradation in flutter speed and improved flutter frequency, but the degree of influence was not similar. The effect mechanism of winglets aerodynamics and wing flutter was not clear enough, hence further studies were necessary to understand the winglet effect on flutter characteristics.

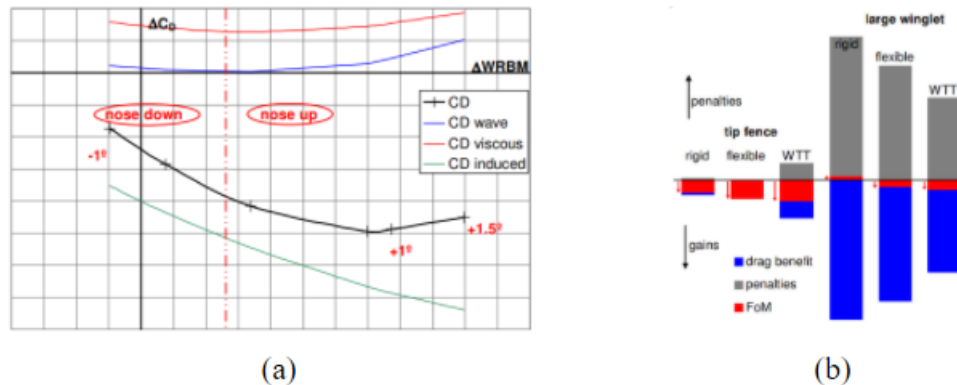


Figure 2-94 (a) Drag and WRBM increments of wing with large wingtip to the reference clean flight wing shape (b) comparison of the rigid and flexible assessment of tip fence and large wingtip [99].

In conclusion, in order to design an optimized winglet with low drag, the ability to predict wingtip flows must be improved by providing more high quality experimental data from flow

visualization measurements. Moreover, folding wingtips have gained a lot of interest for the next generation aircrafts. More research such as the performance of folding winglets in gust alleviation, the flow structure between different folding wingtip dihedrals, vortices of folding wingtips, comparison of different types of folding wingtips and morphing wingtips needed to be studied in the future.

2.9 CONSIDERATIONS FOR MULTI-ELEMENT FLAPS

The development of high lift systems for commercial aircrafts started from the mid-1950s when Boeing installed triple-slotted trailing-edge flaps on B727, B737, and B747, then with double-slotted trailing edge flaps on the B757, and double slotted-inboard and single slotted outboard trailing edge flaps on the B767. It is obvious that the evolution of high lift system development is from multi-element trailing edge flaps to single trailing edge flaps. Multi element flaps have a higher maximum lift (Fig. 2-95(a)), and higher L/D, but have a heavier weight and a more complex system (Fig. 2-95(b)). Hence, although simplified single trailing edge flaps are more lightweight, the maximum lift provided by single element flaps are not enough for landing. This is the reason why double-slotted flaps still exist on current aircrafts. The current research interest for trailing edges is trying to increase the lift performance of single-element flaps, but before the invention of an optimal single-flap on civil aircrafts, it is still necessary to study the flow physics and improve double-slotted flaps [101].

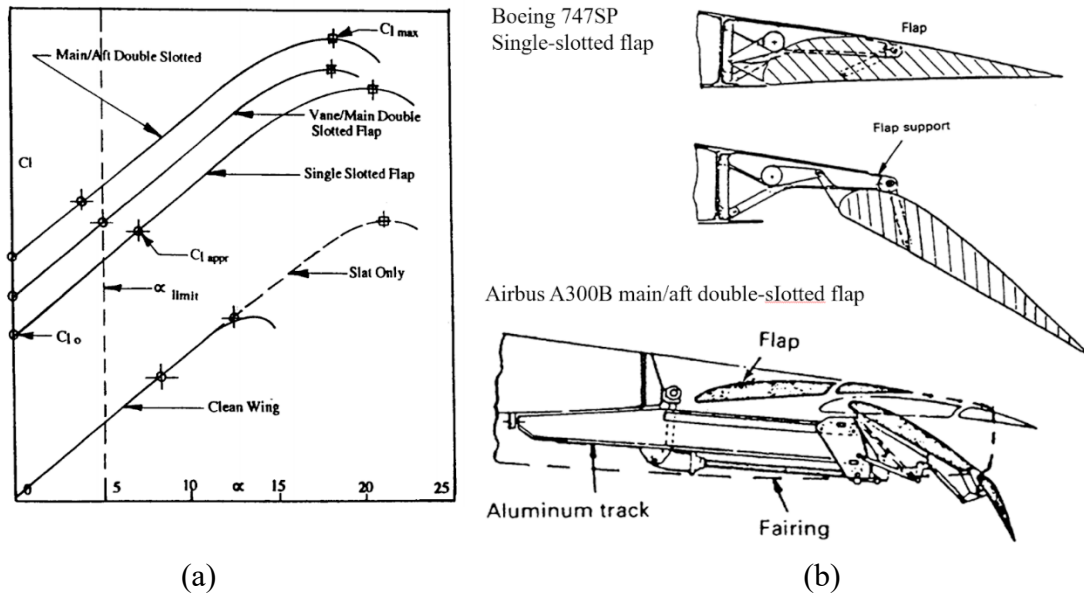


Figure 2-95 (a) Lift performance between different flaps (b) Mechanism of multi-element flap [101].

Rudolph [101] provided a study on aerodynamics performance, and the economic side of high lift systems on commercial subsonic aircrafts. Fig. 2-95(a) shows the lift curves of a clean wing with slat, a single-slotted flap, and two types of double slotted flaps in maximum landing position. The results show that the main/aft double slotted flap increases around 20% of maximum lift compared to the single slotted flap. The installation of flaps however, relies not only on the aerodynamic performance, their impact of weight and cost also need to be considered. Table 4 shows the specific weight of different types of trailing edge flaps. The specific weight of a double slotted flap is 2 times larger than the single slotted flap with track end supports. Moreover, Table 5 shows the manufacturing cost for different trailing edge flaps. The parts included in the single-slotted flaps is 46% less than the part counts from double-slotted flaps. The numbers of parts indicate the manufacture and maintenance cost for trailing edge flaps. Lastly, the cost shows that single-edge flaps have a 51% reduction in cost, saving around 1 million USD per aircraft.

	Flap type					
	Single-slotted	Fixed vane/main	Articulating vane/main	Double-slotted	Triple-slotted	Single-slotted
Support	Hooked track	Hooked track	Hooked track	Hooked track	Hooked track	Link/Track end support
Flap panels	2.7	3.0	3.5	4.8	5.5	2.7
Supports	3.0	3.2	3.8	4.7	5.6	1.5
Actuation	2.2	2.2	2.3	2.4	2.5	2.0
Fairing/flap area	(0.45)	(0.45)	(0.50)	(0.55)	(0.60)	(0.05)
Fairing	1.0	1.0	1.15	1.30	1.40	0.10
Total flap	8.90	9.40	10.75	13.20	15.00	6.30

Table 4 Specific weights of different types of trailing edge flaps (unit: lb/ft^2) [101].

	Trailing-edge-flap type					
	Single-slotted	Single-slotted	Fixed vane/main	Articulating vane/main	Double-slotted	Triple-slotted
Support	Link/track	Hooked track	Hooked track	Hooked track	Hooked track	Hooked track
Weight, ~lb	2080	2940	3100	3550	4340	4940
Part count	1140	1540	1715	1800	2430	2880
Cost, ~million \$	0.542	0.945	1.075	1.273	1.921	2.462
Savings, ~million \$	1.379	0.976	0.846	0.648	Base	(0.541)
Percent flap cost	28	49	56	66	100	128
Percent airplane manufacturing cost	1.4	2.5	2.8	3.3	5.0	6.4

Table 5 Manufacturing cost for trailing edge flaps (for airplanes weight < 250,000~lb) [101].

Aerodynamic data of double-slotted flaps in the public domain are outdated and mostly focused on wing performance such as L , D , L/D with a lack of the inside view of the flow structure at the flap region. This is probably because such data are confidential for aircraft manufacturers. Hence, the author believes that providing fundamental research to acquire up-to-date aerodynamic data with different effectiveness factors and to understand flow separation and interactions on different flap elements is necessary. Furthermore, other effects such as flutter analysis, and 3D

effect are also target issues related to both single-and double-slotted flaps. Wing flutters are dynamic instabilities when the wing is associated with the interaction of aerodynamic, elasticity, and inertial force. There is a lack of research in the comparison of wing flutter effects on single- and double-slotted flaps. [102] 3D effect is the difference of computational results between 3D models and 2D models. One example for this is that 2D models have more separation on the flap, and also a higher pressure coefficient [103].

Aeroacoustic is also another reason why aircraft manufacturers want to replace double-slotted flaps by single-slotted flaps. Single-slotted flaps usually exhibit lower far-field noise levels than double-slotted flaps [104]. Eliasson et al. [105] simulated the aeroacoustic performance of a laminar wing with single and double-slotted flaps. The configurations of the model, the overall sound pressure levels (OASPL), and the pressure fluctuations results on the lower surface are shown at Fig. 2-96. The results from pressure fluctuations indicate that large fluctuations occur at the wing cove for both cases, which accords to the results from OASPL. However, the front inboard flap in Config. 1 occurs more fluctuation than Config. 2, but the noise contribution for both cases in OASPL are similar. The aeroacoustics performance in flaps are still unknown and still need further research on understanding it [104].

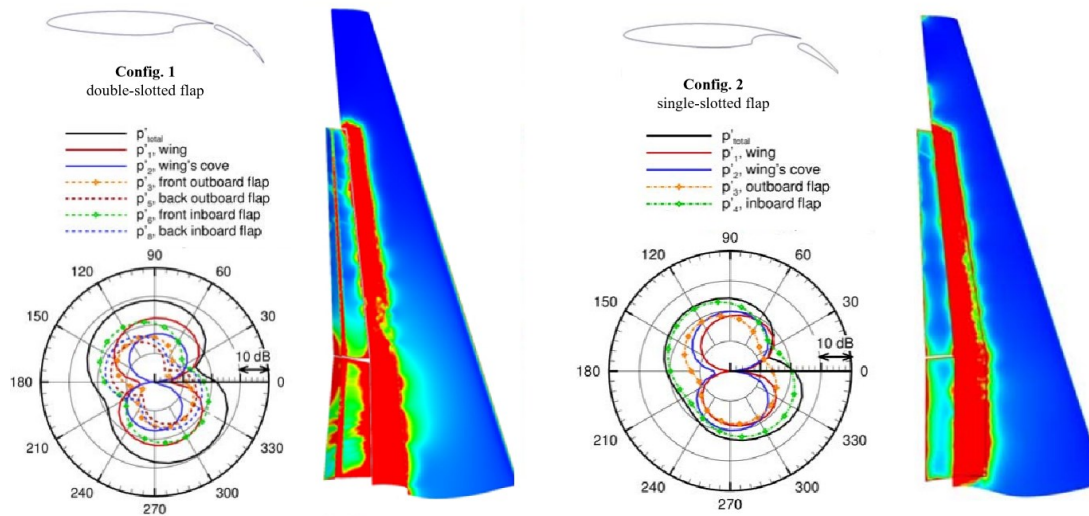


Figure 2-96 OASPL and pressure fluctuations results on the lower surface [105]

In addition to separations on-the-surface, the off-the-surface separation is also an interesting issue to multi-element airfoils. For multi-element airfoils during takeoff and landing, strong adverse pressure gradient can cause off-the-surface separation, which is also known as wake bursting. The sudden expansion of wake bursting will lead to a decrease in lift and an increase in drag, a conceptual sketch of wake bursting is shown at Fig. 2-97. Despite that Smith [106] mentioned this viscous phenomenon in 1975, the mechanism of wake burst hasn't been solved through the past decades. Previous results focused on flap plate tests since full-field measurement of wake bursting over a multi-element airfoil have not been captured at low Reynold number. Moreover, other issues like the influence of adverse pressure gradients to an edge of a wake in a curved flow field is unclear, and turbulence in a continuous burst-wake region over a multi-element airfoil haven't been measured [107].

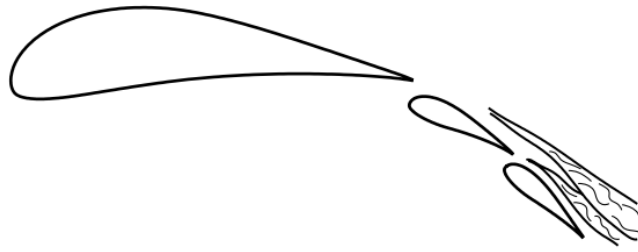


Figure 2-97 Concept of wake bursting over a high-lift multi-element airfoil [107].

Besides understanding the fundamental theories in aerodynamics, designing devices to improve lift and reduce drag is also important. The following shows some interesting devices that could increase the efficiency of multi-element flaps. Matpadi Raghavendra [108] reduced the effects of the wakes on a flapped wing by adding an external flap and placed it above the main flap. Fig. 2-98 shows the lift coefficient plot and the CFD results of the invention. Ortiz Melendez et al. [109] added a micro flap on the trailing edge of the aft element to increase the effectiveness of the flap, as shown in Fig. 2-99. For Fig. 2-100, Petrov et al. [110] uses spoiler drops as deflected wing tails which results in an enhancement of the lift. Lastly, a lower shroud sealing plate called plain flap (Fig. 7) was added to the DLR TuLam aircraft configuration aiming to improve high lift performance. However, the effect of plain flaps is still unknown, and needed future research [103].

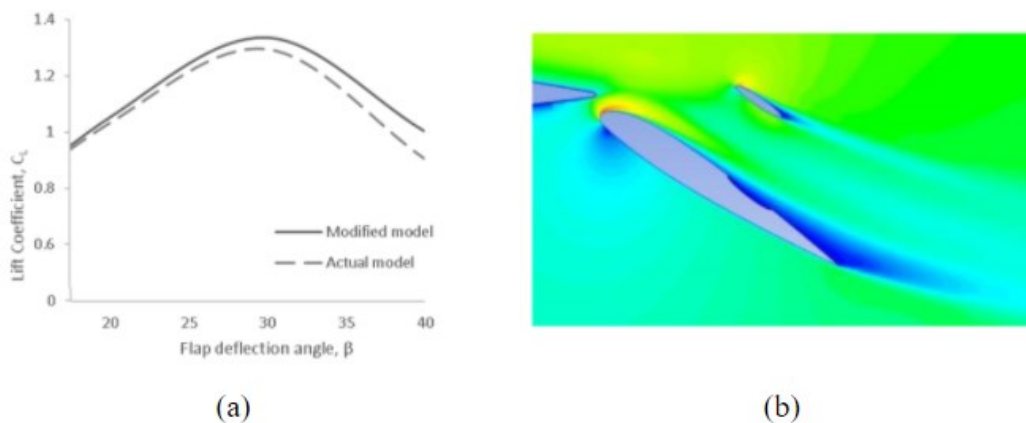


Figure 2-98 (a) Lift coefficient comparison of the designed model and the original model (b) Velocity contour of the designed model from CFD results [108].

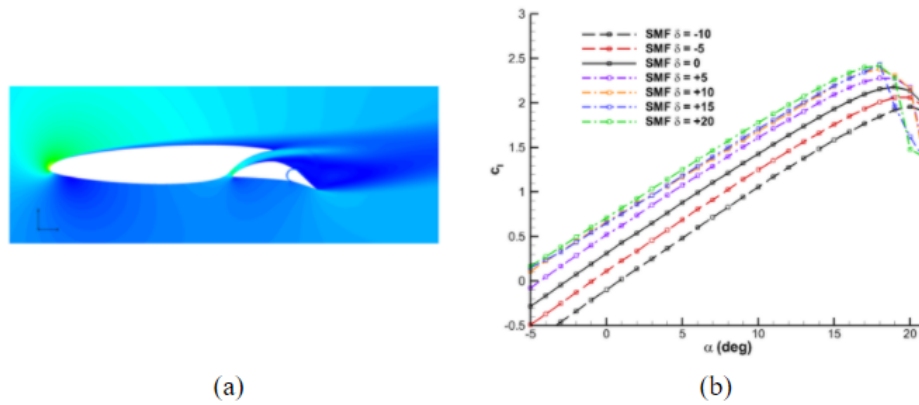


Figure 2-99 (a) Velocity contours of the micro flap from CFD results (b) Lift coefficient dependence on angle of attack of various sealed micro flap deflections [109]

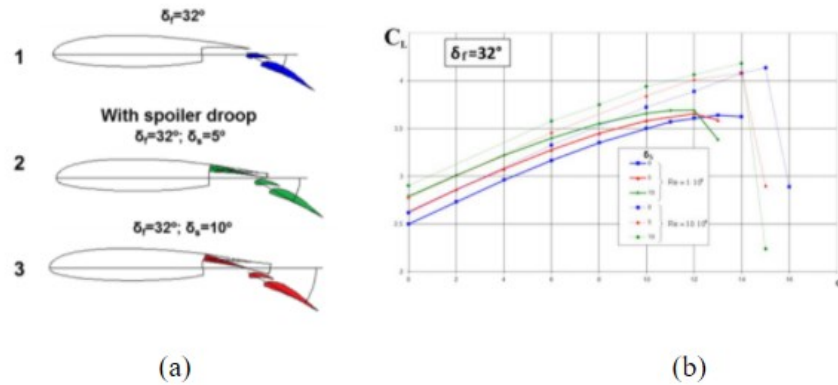


Figure 2-100 (a) Schematic sketch of airfoil with spoiler droop (b) Effect of airfoil with spoiler drop [110]

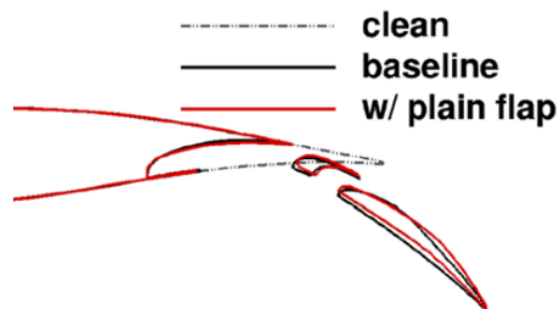


Figure 2-101 Schematic sketch of the plain flap design [103]

In a nutshell, there's a lack of flow-field data of the double-slotted flaps to the public due to security issues from aircraft manufacturers. Hence, providing fundamental research using the latest technologies and providing a database for the aerodynamic and aeroacoustic results might give us a further understanding of how to improve single-element flaps.

2.10 ICE SHAPE TESTING

Airplanes flying in ice conditions might influence aircraft safety since airfoils with ice will affect aerodynamic performance. After decades of research, some of the most detailed parameter studies on different ice shapes and aerodynamics especially in 2D icing have been completed. This leads to an accurate prediction for ice accretion using LEWICE, a computational tool that simulates ice growth on surfaces. However, the understanding in 3D ice shapes is rare and the prediction from LEWICE3D is inaccurate. Fig. 2-202 shows a comparison of ice accretion from LEWICE3D and experimental results from NASA Glenn Icing Research Tunnel by Fujiwara¹, et al. [111]. The results show that LEWICE3D cannot simulate most ice accretions accurately, especially for horn ice and in the inboard section. The following article provides an introduction to the current understanding of different icing shapes, the effects, and the study interests for future research.

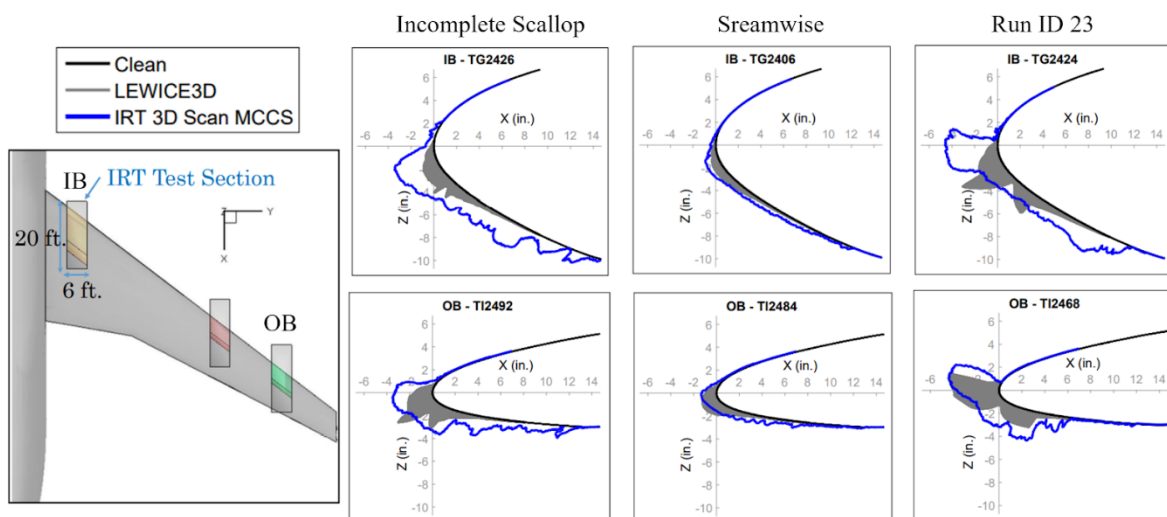


Figure 2-102 Comparison of ice accretion between numerical simulations (LEWICE3D) and experimental results [111]

For the shape of ice, four different representative geometries have been classified: ice roughness, horn ice, streamwise ice, spanwise-ridge ice. Ice roughness occurs at the initial stage of the ice accretion process before it turns into a significant ice shape. Roughness is characterized by ice height, density, surface location, and has been defined into three main zones: smooth zone, rough zone, and the feather region. Fig. 2-203(a) shows a concept of ice roughness features. The parameters of the roughness elements in each zone are caused by different freezing fractions and accumulation parameters affect airfoil performance a lot. Current studies mainly focus on the roughness effects on 3D swept wing performance (Fig. 2-203(b)). However, more data in the flow field are needed to understand the effect of roughness size, location, separation that interacts with the spanwise flow and concentration on swept-wing aerodynamics [112][113][114].

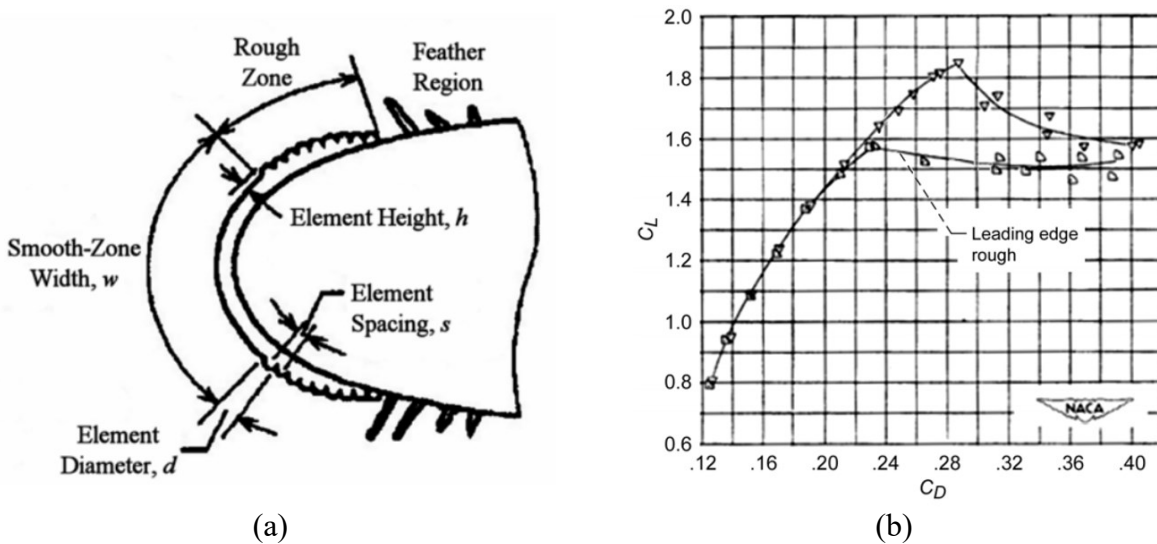


Figure 2-103 (a) Ice roughness features (b) Effect of roughness on lift [113]

Horn ice is a horned like shape ice accreted at the leading-edge (Fig. 2-104(a)). When flow flows through the top of the horn, the boundary layer separates and forms a shear layer. The shear layer then transitions to turbulent flow, influencing the aircraft performance. Fig. 2-105 shows the

velocity vector field downstream of the horn ice and the effect of horn ice on lift. For a 2D single horn, the characterization of the shape is its height, the angle between the chord line, and its location indicated by s/c , as shown in Fig. 2-104(a). For 3D horn ice, its geometry and flow field is way more complicated than 2D horn. Fig. 2-106 shows the three main subcategories (complete scallops, incomplete scallops, and no scallops) for 3D horn ice and Fig. 2-107 shows the difference of a separation bubble between 2D and 3D simulation horn-ice accretion using surface flow visualization. Fundamental aerodynamic understanding such as laminar separation bubbles, transition, reattachment, and separation region for 3D horn ice with complete scallops is unknown. This makes the simulation of 3D horn ice accretion less accurate, for example, the difference between simulated ice shape from LEWICE3D and the actual geometry in Fig. 2-104(b) [112][113][114].

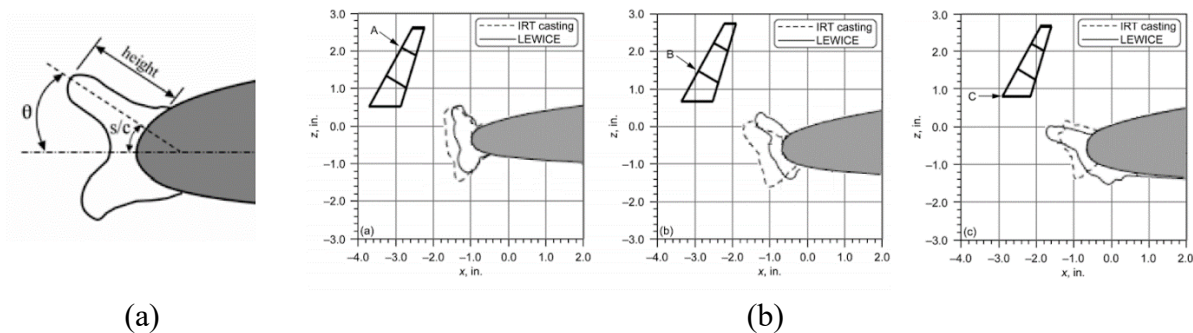


Figure 2-104 (a) Geometry of horn ice shape 2 (b) Comparison of horn ice shape from numerical simulations (LEWICE3D) and experimental results (IRT) [113].

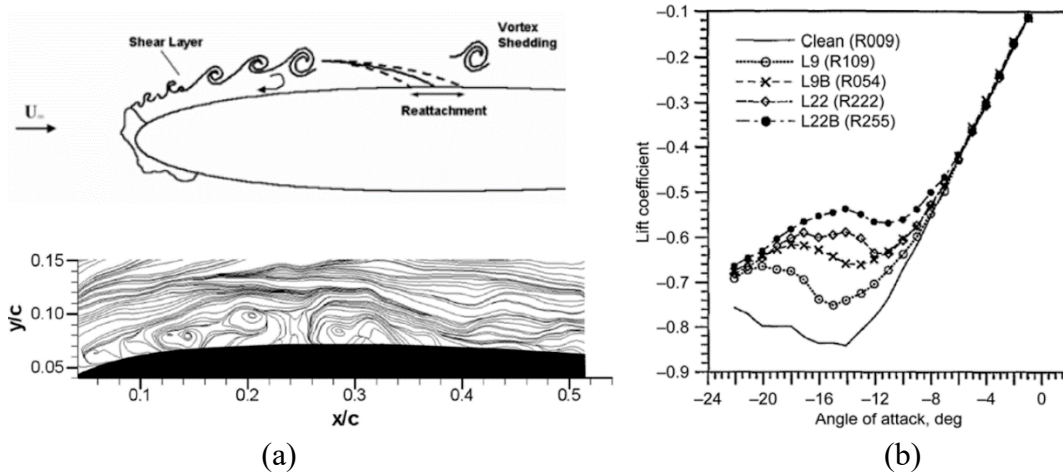


Figure 2-105 (a) Velocity vector field downstream of the horn ice [112] (b) Effect of horn ice on lift [113].

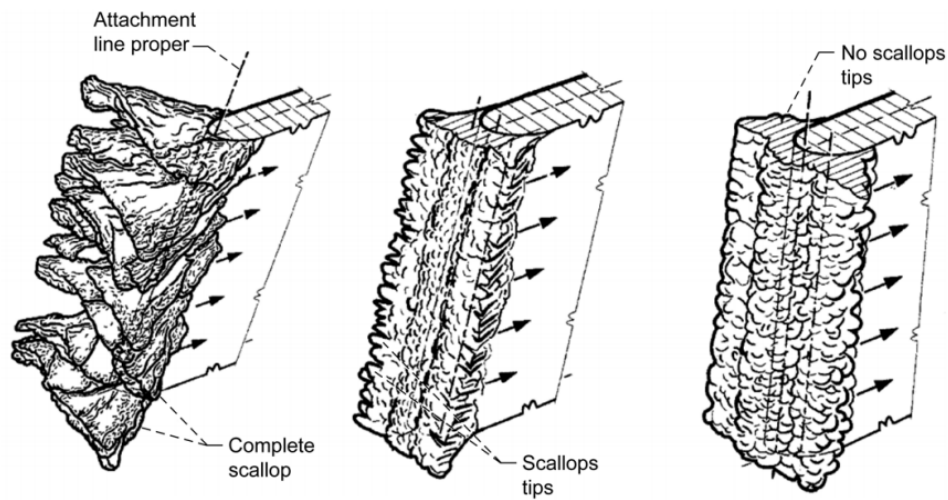


Figure 2-106 Configurations of the subcategories of horn ice [113].

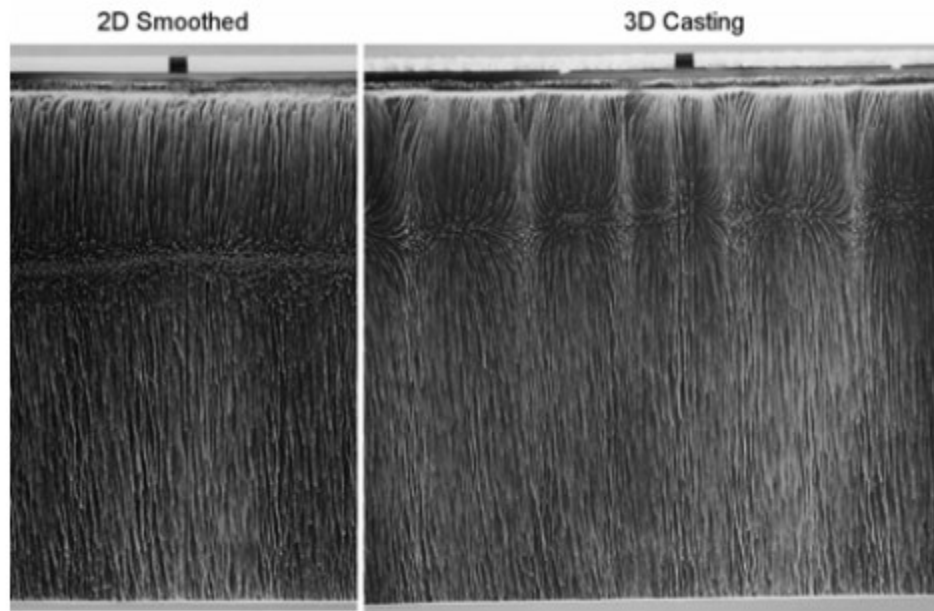


Figure 2-107 Difference of a separation bubble between 2D and 3D simulation horn-ice accretion [112].

Streamwise ice often forms at cold temperatures when the incoming droplets freeze on the surface at impingement. Fig. 2-108 displays the geometry of streamwise ice on different wing locations from IRT ice accretion and LEWICE simulations. In the past, since the impact of streamwise ice on aerodynamics is low than other types (Fig. 2-109), the understanding of streamwise ice is way lesser compared to other geometries. Despite that, both figures show that the simulation from LEWICE 2D provides good agreement with experimental results. In the future, more aerodynamic data for streamwise ice in 3D wings is needed in order to improve the code in LEWICE3D [113].

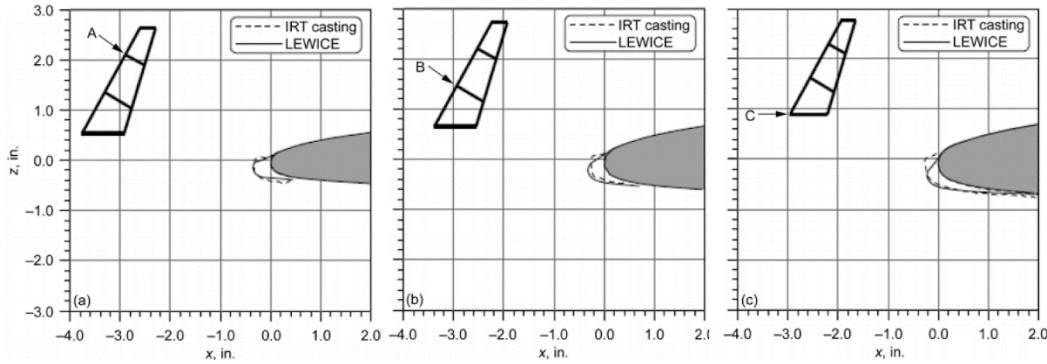


Figure 2-108 Geometry of streamwise ice on different wing locations from IRT ice accretion and LEWICE simulations [113].

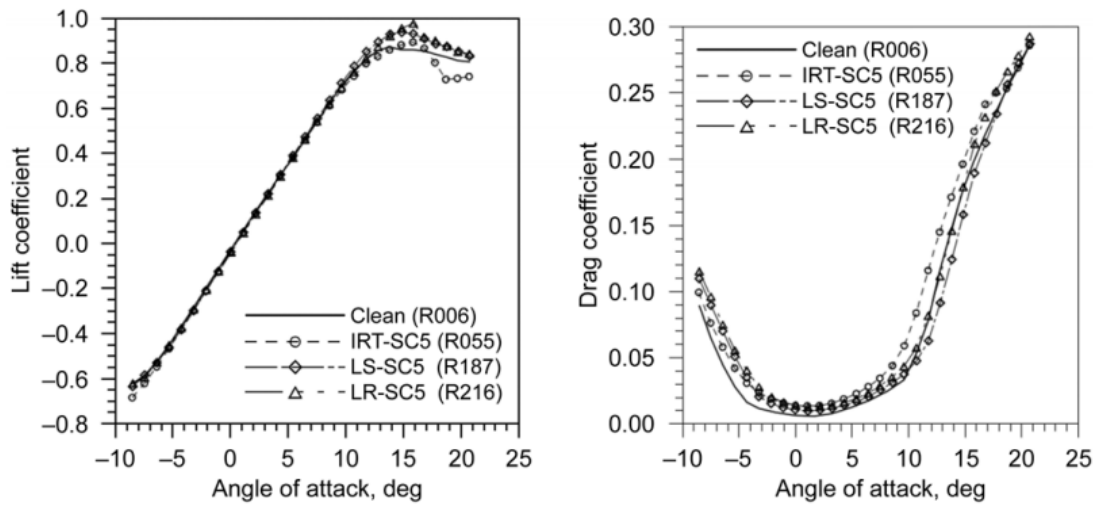


Figure 2-109 Effect of streamwise ice on aerodynamics (LEWICE: LS-SC5, LR-SC5) [113].

Previous ice shapes have certain configurations and are located at the leading edge. Spanwise ridge ice are “free” ice shapes that are further downstream of the leading edge, shown in Fig. 2-210(a). These spanwise ice will cause the boundary layer to separate and form an instability thin shear layer (Fig. 2-210(b)). The vortices in the shear layer will then roll up and generate a large separation region. Hence, the separation region relies on the geometry of the ice shape. Fig. 2-211(a) shows a classification of different simplified ice shapes used for simulation in the past, and Fig. 2-211 (b) shows the effect of different ice shapes on the performance. However, spanwise ice can also increase lift coefficient by entraining higher-momentum fluid into the

boundary layer, just like vortex generators. Fig. 2-211(c) shows the result of simulated spanwise ice located on a different airfoil but with the same Reynold number in the results in Fig. 2-211(b). Previous studies focused on 2D ridge ice shape and the aerodynamic performance of the wing, the flow field behind 3D spanwise ice is not understood. More data with flow field investigation from high fidelity 3D ice shape is needed to study the effects of ridge ice height, shape and wing geometry [112][113].

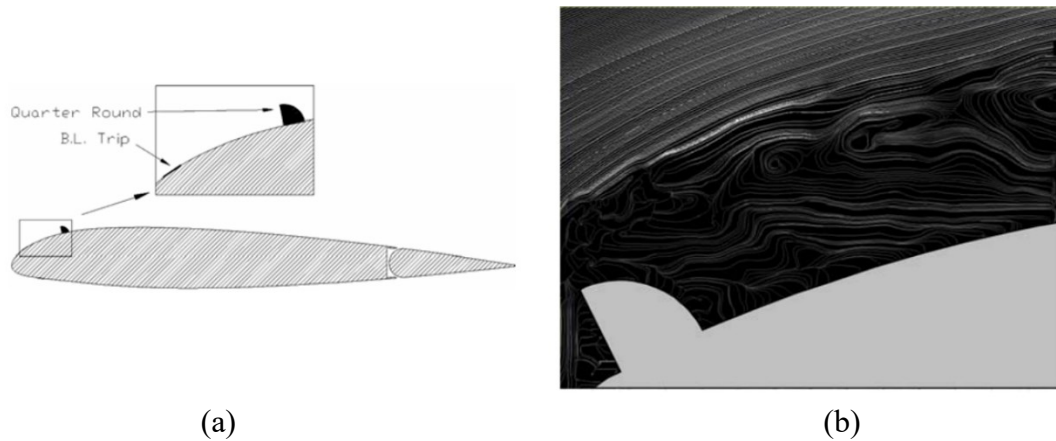


Figure 2-110 (a) Schematic drawing of spanwise ridge ice accretion (b) Flow streamlines over spanwise ridge [112]

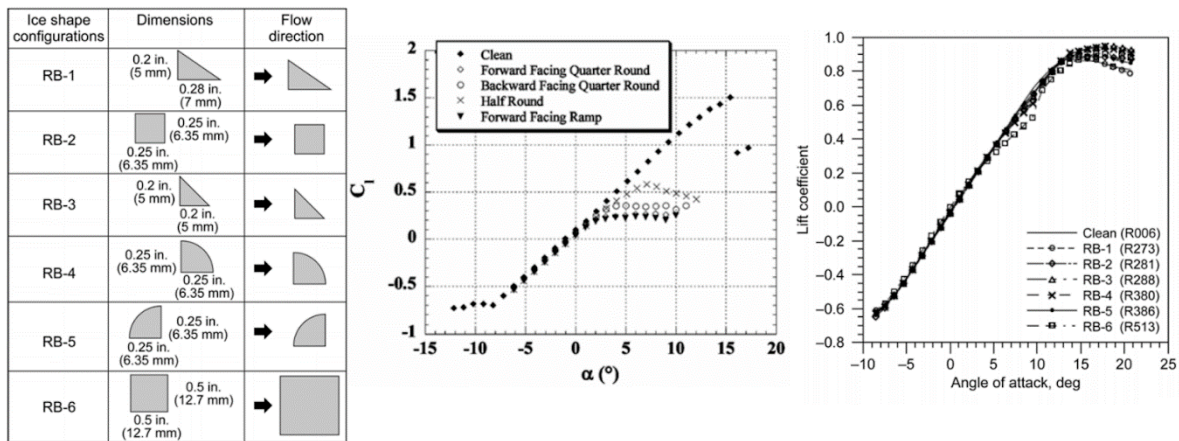


Figure 2-111 (a) Classification of different spanwise ice shapes [113]. (b) Effect of spanwise ice shape on a NACA 23012 airfoil for $Re = 1.8 \times 10^6$ [112]. (c) Effect of spanwise ice shape on a GLC-305 swept airfoil for $Re = 1.8 \times 10^6$ [113].

Supercooled large droplet is a typical example of spanwise-ridge ice. The SLD condition is defined as droplets that are ten to hundred times larger than normal rain drops that exist in a liquid form at air temperature below 0°C . It is a well-known issue that caused the ATR-72 commuter aircraft accident in 1994. In recent years, a number of studies focused on SLD conditions. Zhang et al. [115] listed the achievements on SLD conditions made by previous researchers and listed some missing parts regarding SLD conditions on high-lift configurations. Previous research distinguished icing characteristics of SLD against normal icing features. However, the flow field and the aerodynamic performance degradation of SLD on high-lift configurations, especially under critical near-freezing SLD conditions are neglected [115][116]. Fig. 2-112 shows the velocity magnitude around cleaned and iced airfoil. Secondly, there's a lack of research into the icing feature in the takeoff and landing conditions, and also the distinction between SLD and non-SLD icing for swept wings [115].

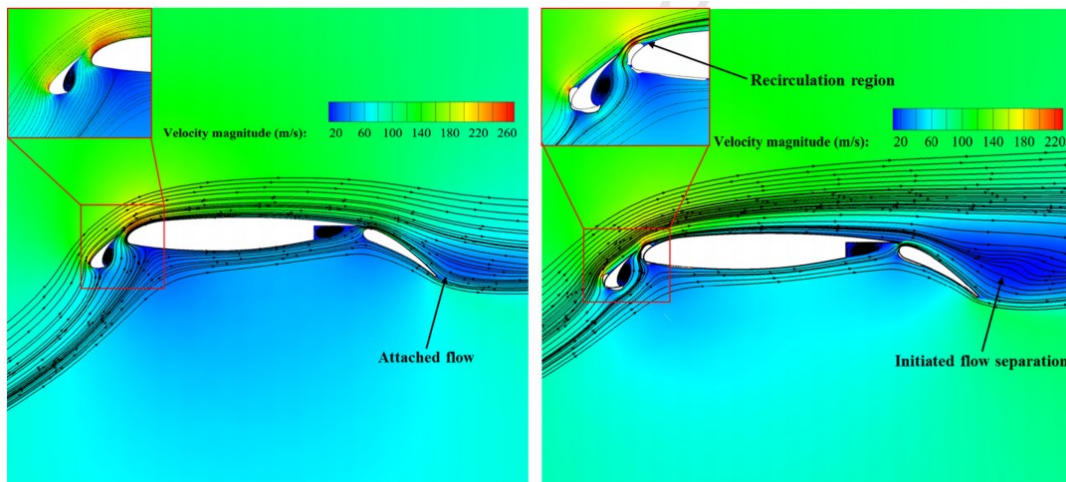
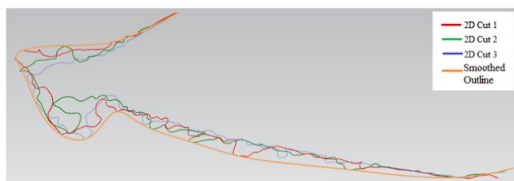


Figure 2-112 Velocity magnitude around cleaned and iced airfoils [116].

Camello [117] studied the influence of ice shape fidelity on aerodynamic performance. Fig. 2-113 and Fig. 2-114 shows a comparison between high-fidelity and low-fidelity maximum scallop configurations. For the ice shapes, low fidelity ice shapes use a 2D smooth and 2D smooth with

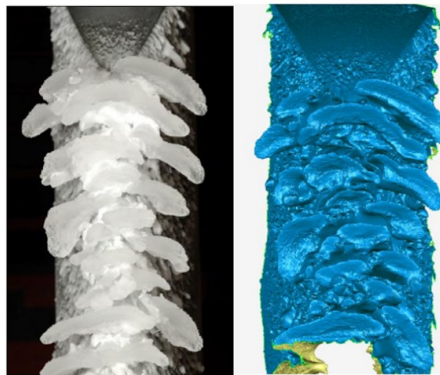
grit roughness ice-shape based on the high fidelity ice shape geometry. Fig. 2-114(a)(b)(c) shows the shape of the high-fidelity artificial ice shape used in the research. The remaining plots in Fig. 2-114 shows the aerodynamic results from force balance. The results show that the trends for all three cases are similar, the performance between the low-fidelity versions are alike, and the high-fidelity shape cannot produce the same aerodynamic performance as the low-fidelity versions. The difference between the performance of high-fidelity and low fidelity shape can be explained by the flow visualization results shown in Fig. 2-115. For the high-fidelity configuration, almost all of the tufts on the wing were oriented in the spanwise direction, indicating that flow separation happens all over the wing surface. However, for the low-fidelity model, separation happens at about 20% of the span location.



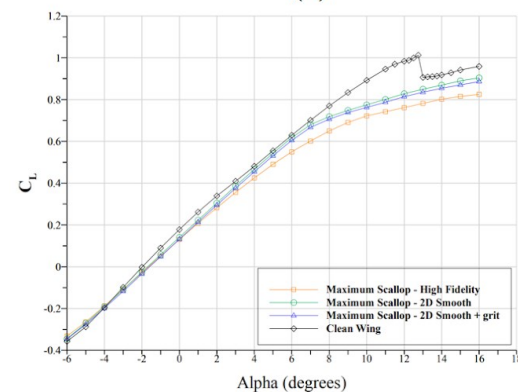
(a)



(b)



(c)



(d)

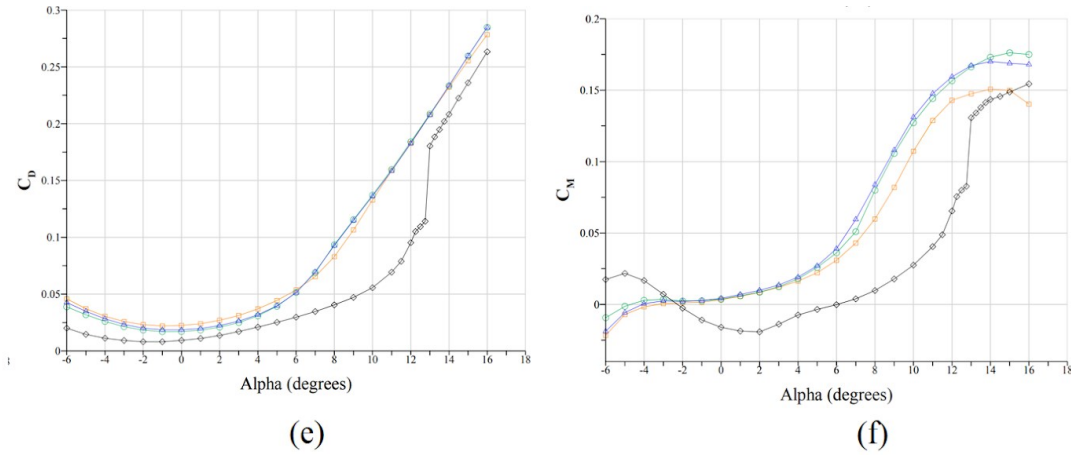
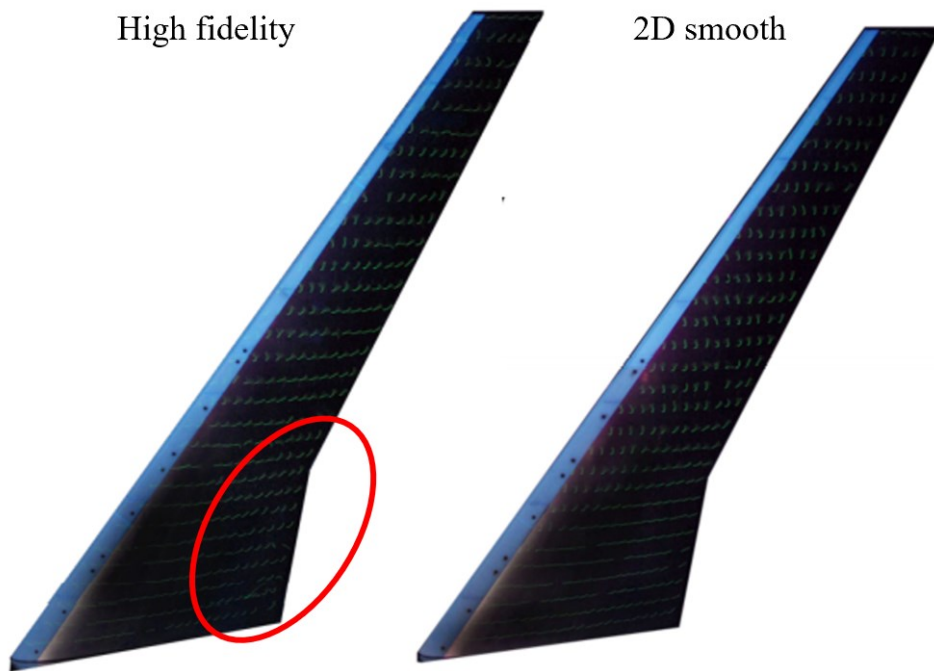


Figure 2-113 (a) Configuration of the 2D smooth low fidelity ice shape (b) Photograph of the applied 2D smooth ice shape with grit roughness (c) Comparison of real ice shape and high-fidelity artificial ice shape (d) Effect of ice shape fidelity of maximum scallop on lift (e) Effect of ice shape fidelity of maximum scallop on drag (f) Effect of ice shape fidelity of maximum scallop on moment coefficient [117].



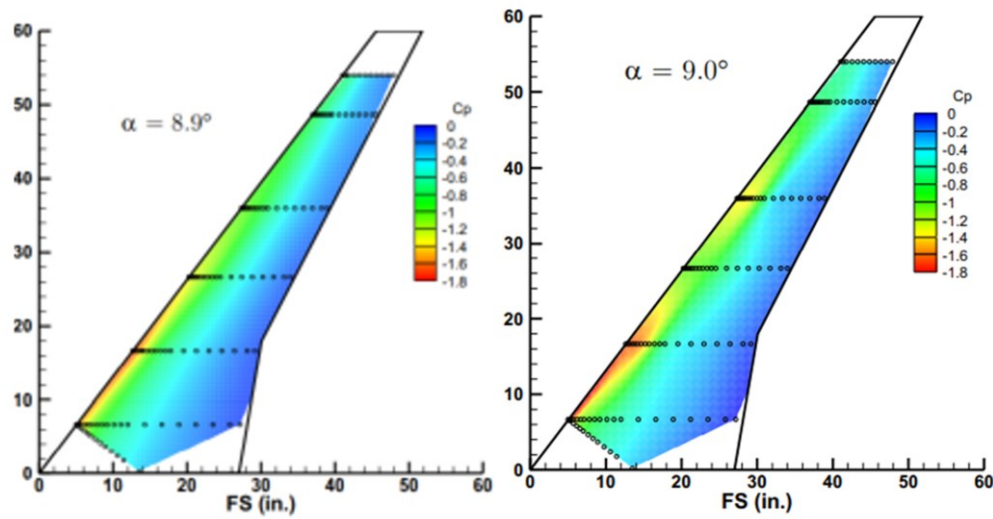


Figure 2-114 Comparison of the tuft visualization results and pressure distribution between the high-fidelity and low-fidelity cases [117].

The effects of Reynolds number and Mach number on airfoil is a well-known effect for clean airfoils, but for iced airfoils, they are altered by different ice shapes and roughness. For current understanding, Reynolds number does not influence iced-airfoil performance a lot since the boundary layer separation governed by ice shape reduces the effects of Reynolds number. Fig. 2-115 demonstrates the effect of Re on maximum lift coefficient on an airfoil with different leading edge roughness. However, there are still some phenomena that needed to be explained. For instance, there may be some small increases in C_{Lmax} as Re increases up to 3.5×10^6 , but no change for larger Re for large ice shapes. Additionally, an iced airfoil with streamwise ice have a better AOA performance than a cleaned airfoil at $Re = 1.8 \times 10^6$, but a worser performance at $Re = 15.9 \times 10^6$. Moreover, the roughness level needed to be maintained in streamwise/ice shapes in order to let low Re data to extrapolated to high Re . For Mach number, the effects of Mach number in the lower subsonic range are well understood, but not for high Mach number.

One example is that the separated flow region after a large ice shape tends to increase at a large Mach number, and the reason for this is not clear [112].

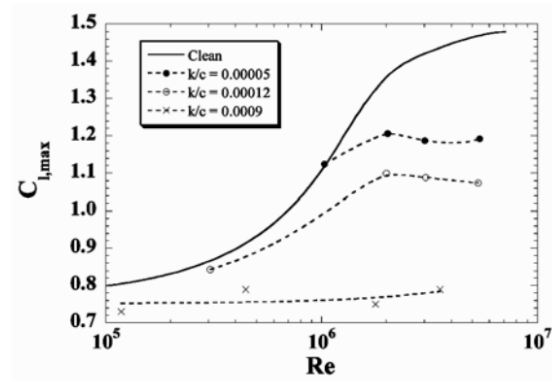


Figure 2-115 Effect of Re on maximum lift coefficient on an airfoil with different leading edge roughness [103].

Other effects that haven't been listed in the report by Bragg, et al. [112], and Broeren, et al. [113][114] are ice accretion on engine spinners and cold soak fuel frost. While aircraft operating in cold weather, SLD might freeze on aero-engine components like fan blades, inlet tips and spinners. Ice shedding from these parts might damage the fan and cause stall and surge. Among these components, the spinner of the engine is more sensitive to icing because the rotation speed is relatively low and a larger imping is for SLD. Fig. 2-116 shows photos of final ice buildup on the engine fan model with different spinners. For engine icing, most previous studies focused on the prediction of ice shape, heat transfer process, and the performance of anti/de-icing systems. Hence, more research is needed to understand the dynamic icing accretion process over the fan components and evaluate the icing influence on the aero-engine fan performance [118].

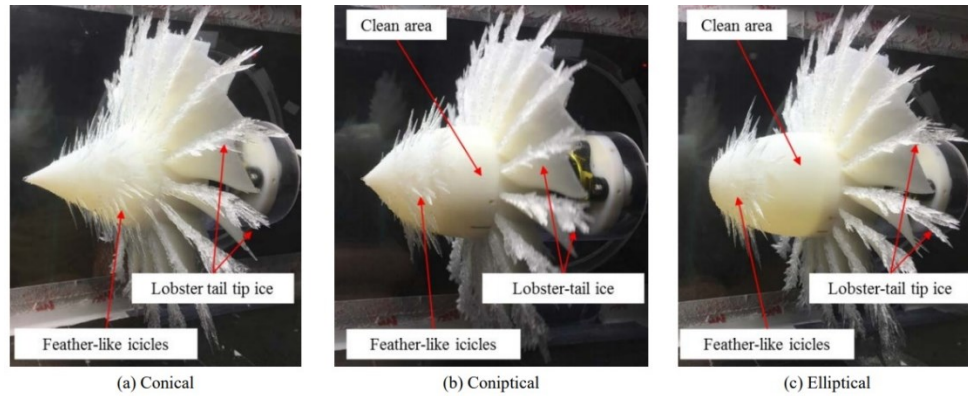


Figure 2-116 Final ice buildup on engine fan model with different spinners [118].

Cold Soaked Fuel Frost (CSFF) is formed on aircraft wing tank area on ground when the aircraft has been at high altitude where the wing structure and fuel have been cooled down. This phenomenon is an interest to airlines and aircraft manufacturers because for aircrafts at the next take-off after a stopover, the Federal Aviation Administration of the United States requires a clean wing concept on the upper wing surface. However, the amount of published studies with real frost related to aircraft is extremely scant. Koivisto [119] published a literature review on CSFF formation based on studies in the refrigeration industry. Soenne and Rosnell [120] measured the frost geometry and its corresponding aerodynamic effect on a HLCRM model and showed that the frost may evaporate during take-off. Beyond that, no consensus on the correlation form of CSFF, and no closed form solution for frost growth, is presented.

In conclusion, current understandings of the shapes and flow field of ice-accretion airfoil are based on straight wings, but the flow field of a swept wing is significantly different. Previous studies focused on the overall effect of 3D ice shape on aerodynamic performance, with a lack of research in the flow-field and corresponding performance penalties associated with ice accretion on various parts of the aircraft. More research is needed to be done on the ice accretion and its effect on 3D configurations, these data could be used for making LEWICE3D to predict more

accurate ice shapes. The study of icing accretion requires an icing research tunnel that can simulate elements of the atmospheric icing environment. Since UW KWT is only a subsonic wind tunnel that doesn't have the capability to provide icing conditions, studying icing accretion on the HL-CRM at UW KWT is unavailable.

2.11 ACTIVE FLOW CONTROL

Flow control is an important topic in the aerodynamic field, aiming to have a large engineering benefit by implying a small change in the flow field distribution that could increase the efficiency thus reducing fuel consumption of the flying vehicle. The purpose of flow control is to achieve different engineering goals like: transition delay, lift enhancement, heat transfer enhancement, noise reduction, separation postponement, turbulence augmentation, drag reduction, and skin friction reduction. There are mainly two methods for flow control: passive and active. Passive control uses different geometric shaping to manipulate pressure gradient, and active flow control aims at changing a natural flow into a more desired path by adding auxiliary power into the flow. Passive flow control techniques have been applied to current commercial airplanes to delay flow separation like VGs on the wing, or nacelle chines on the engine. On the other hand, using AFCs to improve aerodynamic performance is still in the research phase. The main interest of AFCs on aircrafts is to increase the lift of a simple hinged flap in order to replace the conventional flap, delay the nacelle induce wakes, and to reduce the size of the vertical tail. Since the application of AFCs on nacelles and vertical tails have been discussed in Chapter 2.03 and 2.05. This article focuses on the installation of AFCs on a sweeping wing [121][122].

Piezo-electric actuators and fluidic actuators are the two groups of AFCs that are most widely used. Piezo-electric actuators generate blowing flows or pulsed suction into the flow that

does not require a fluidic source, like synthetic jets, and plasma actuators. In comparison, fluidics actuators are those which require air supply such as sweeping jets [40].

Synthetic jet actuators (SJA) generate vortex structures by oscillating the diaphragm (Fig. 2-117(a)). Previous research shows that synthetic jets could inject additional momentum into the boundary layer and affects the instability of the separated shear flow. Tang et al. [123] studied the effect of SJA arrays on a low-speed small UAV wing model. Two SJA arrays were installed in the wing model; force balance, and PIV velocity field measurements were done in the experiment. Aerodynamic force measurements in Fig. 2-118 showed that SJA actuation could increase the lift coefficient, reduce the drag coefficient, and delay stall angle of attack. Actuators in the front arrays operating at high frequency have the highest improvement in maximum lift coefficient. Fig. 2-119 presents the time-average and instantaneous velocity vectors with and without SJA actuators. The time-average plot first shows that the low speed flow near the wing was energized as the velocity in the region increased. The instantaneous plot then reveals that the extra momentum from SJA actuators does not “eliminate” the shear layer, but alters the local flow field.

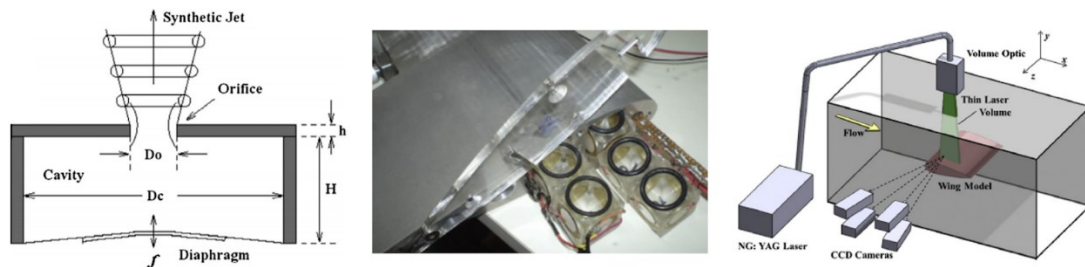


Figure 2-117 (a) Schematic of the SJA (b) Photograph of two SJA installed in the wing model (c) Experiment setup of the PIV measurements [123].

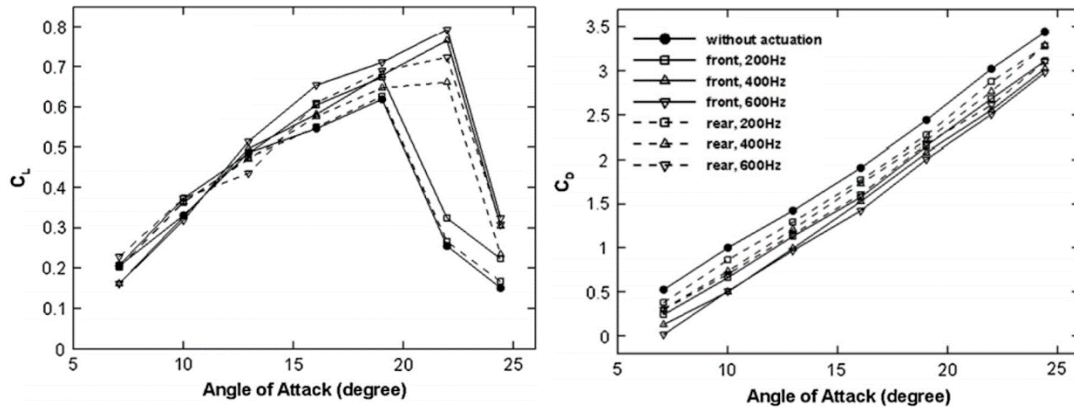


Figure 2-118 Effect of SJA arrays on airfoil [123].

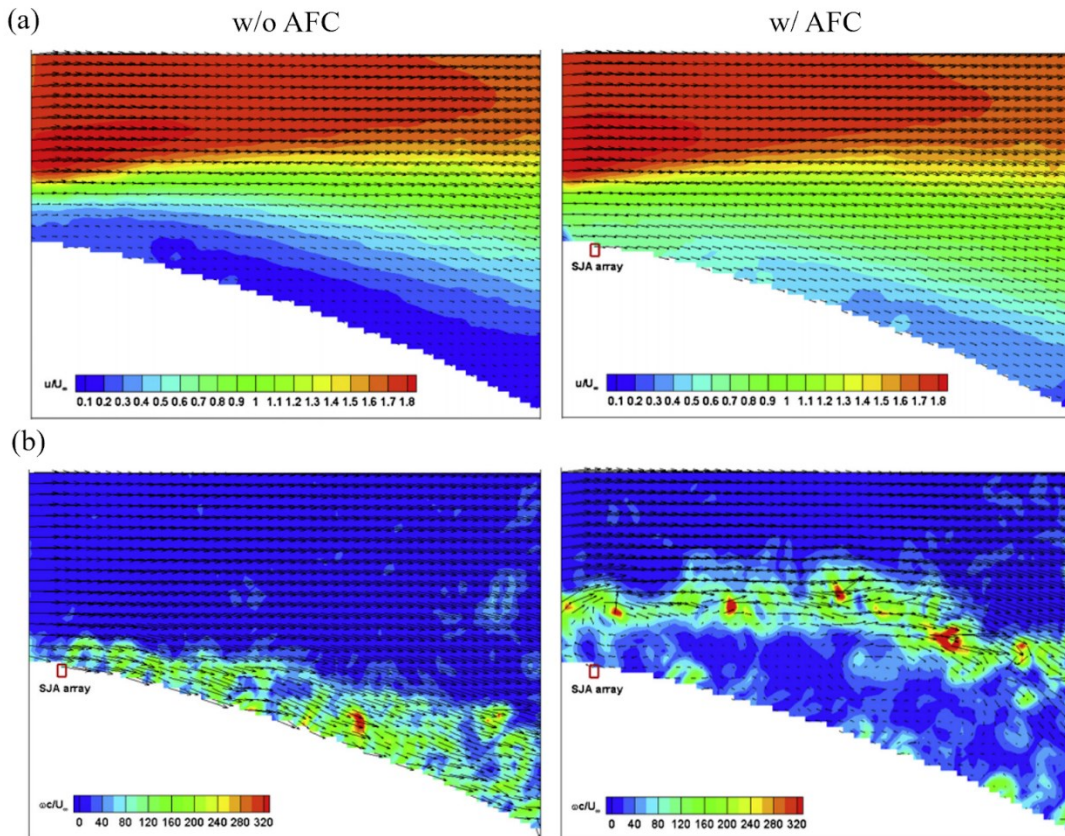


Figure 2-119 (a) Time-averaged (b) Instantaneous velocity contours with and without SJA arrays [123].

Plasma actuators transform momentum to the flow by ionizing the air between electrodes using a high-voltage signal. Fig. 2-120(a) provides the schematic of a dielectric barrier discharge

(DBD) plasma actuator. The advantages of plasma actuators such as simple construction, lack of moving parts and ease of installation provides them the potential to be used to delay separation on airfoils. Little and Samimy [124] tried to use a single plasma DBD actuator to control the boundary layer separation. Experiments are done on a simplified high-lift version of the NASA Energy Efficient Transport airfoil with a trailing edge flap. Fig. 2-120(b) shows the effect of DBD actuators on the airfoil at different flap angles δ_f . The flow structure of the most effective case (blue line) was then analyzed using PIV measurement. The streamlines and the time-averaged normalized vorticity of the baseline and the DBD controlled case are displayed in Fig. 2-121(a) and Fig. 2-121(b). The streamlines on the flaps shows that DBD actuators increase suction on the flaps and causes greater acceleration at the leading edge and the flap shoulder. The enhanced circulation then leads to a delay in separation and lift enhancement. Moreover, the time-averaged spanwise vorticity fields corresponding to the streamlines are shown in the below figure. It is obvious that size and the area of the recirculation region is reduced by the effect of plasma actuators.

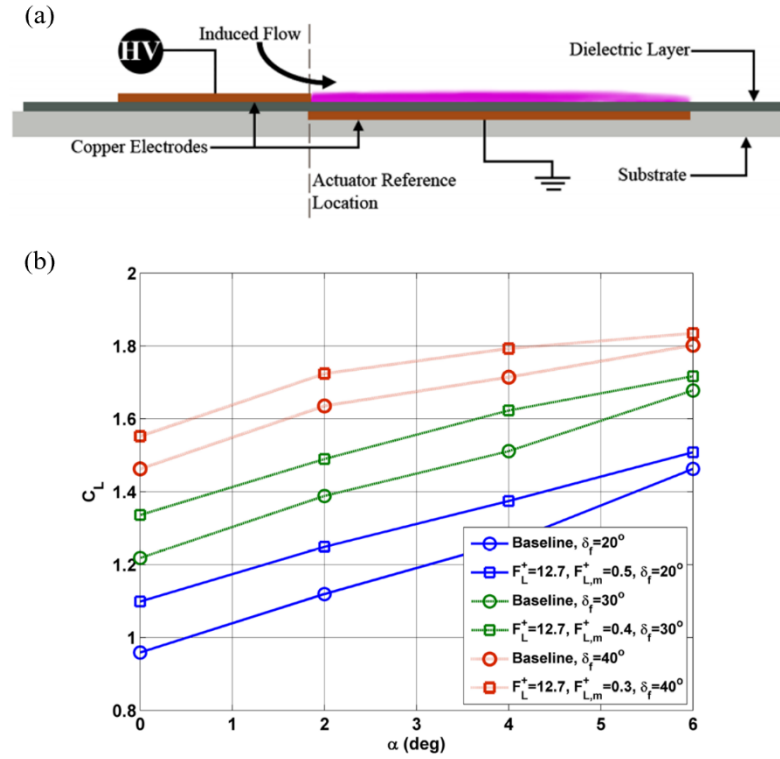


Figure 2-120 (a) Schematic of the DBD plasma actuator (b) Effect of DBD actuators on the airfoil at different flap angles ($Re = 240k$) [124].

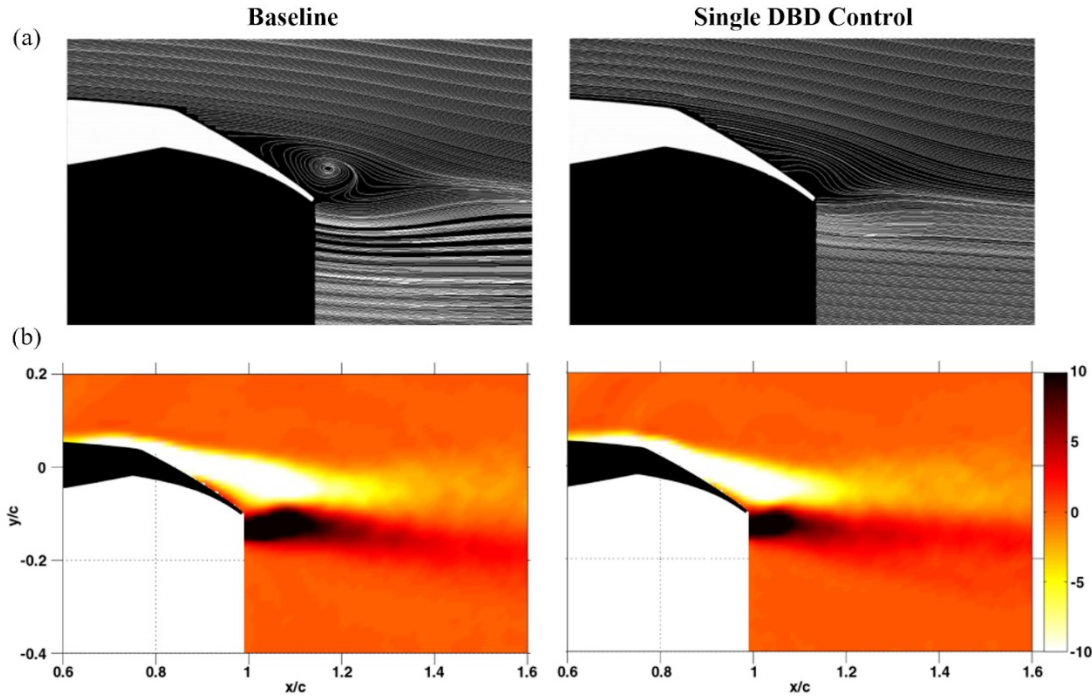


Figure 2-121 (a) Streamlines of the baseline and plasma control airfoil (b) Time-averaged normalized vorticity of the baseline and plasma control airfoil [124].

Sweeping jet actuators (SWJ) inject an oscillating jet into the air from a supplied pressurized fluid. The advantage of SWJ is that it does not require any moving parts to refill: the supplied air rushing to the outlet will increase the pressure at the inlet to the upper feedback channel, and creates flow in the feedback channel that pushes the entering jet back to the opposite surface and repeats the process. Melton, and Koklu [125] studied the effect of SWJ on a NACA 0015 semi-span model (Fig. 2-122(b)). Fig. 2-123 shows the results of SWJ at different operation conditions, where C is the moment coefficient of the SWJ actuator. The lift polars show that SJA are able to increase the lift coefficient, and the lift coefficient increases while the momentum provided from the actuator increases. The mean velocity contours reveal the mechanism of SJA. The large circulation region on the baseline case was first reduced by small momentum $C_\mu = 1.2\%$ provided by the SWJ. While the injected momentum increases, the sweeping jet produces a

large positive vortex that alters the original negative vorticity. Finally, the large C_μ levels eliminates the circulation region on the flap.

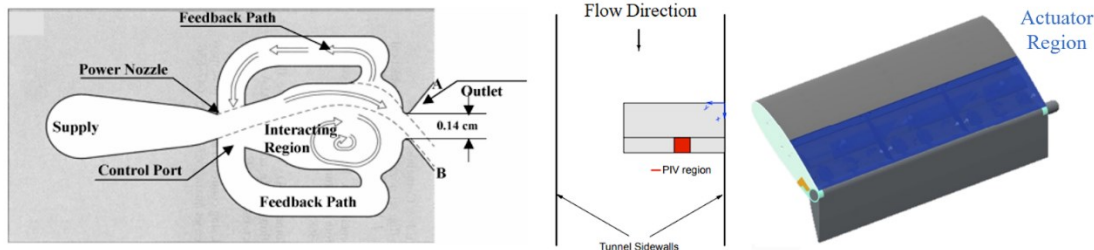


Figure 2-122 (a) Conceptual design of SWJ [69] (b) Setup of the experiment [125].

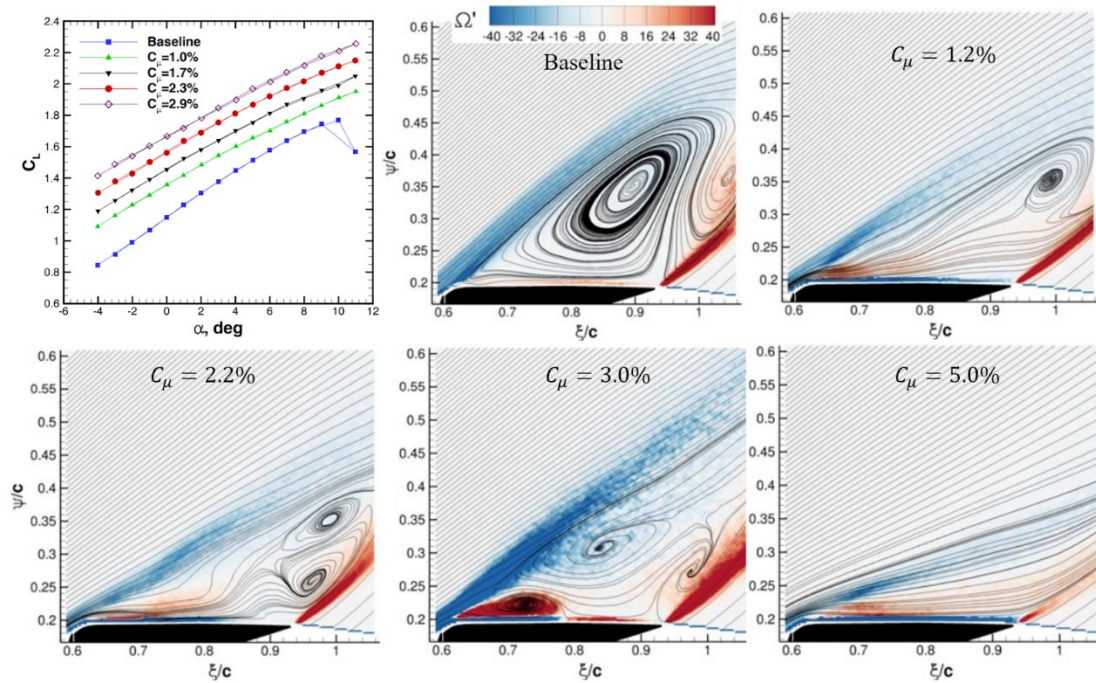


Figure 2-123 Effect of SWJ on lift ($Re = 750k$) and mean vorticity contours from PIV on the flap ($Re = 500k$) with $\delta_f = 40^\circ$ [125].

Yet the application of AFCs on airfoil is not limited to a single type, a combination of different AFCs might have more improvement in the performance. In the NASA HL-CRM project, the effect of a simple hinged flap airfoil with different AFC devices were compared with a conventional flap configuration (Fig 2-124). Three types of AFC devices (Table 6) were

investigated: the DRSWJ actuator used two rows of sweeping jets, the APJ used two rows of pulsed jets, and the HELP actuator combined a row of sweeping jets, and a row of pulsed jets together. First, by comparing only the results from the CRM-SHL-AFC (Fig. 2-125 (a)) shows that the effect of AFC is obvious, especially the HELP actuator. Two rows of different actuators operating together produces larger lift than the sum of each row actuating individually. However, when taking conventional flaps into consideration. The results in Fig. 2-125(b) shows that the L/D from simple hinged flaps is still not enough, despite that the HELP configuration could produce equal lift as the conventional flap at the stall condition. It is a pity that flow visualization measurements are not included in this experiment to investigate the difference between both configurations [30].

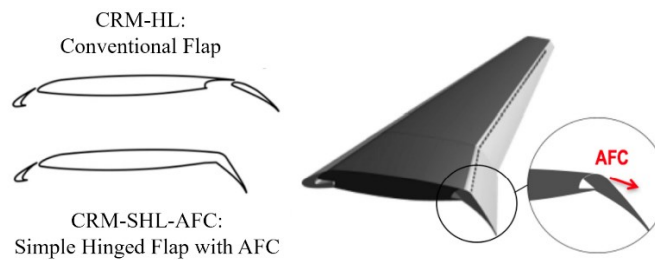


Figure 2-124 Concept of testing configurations [30].

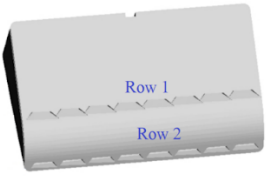
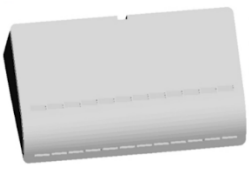
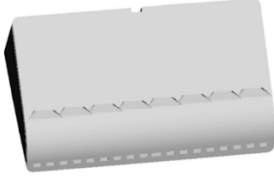
	DRSWJ Actuators	APJ Actuators	HELP Actuators
			
Row 1	Sweeping Jet	Pulsed Jet	Sweeping Jet
Row 2	Sweeping Jet	Pulsed Jet	Pulsed Jet

Table 6 AFC cartridges [30].

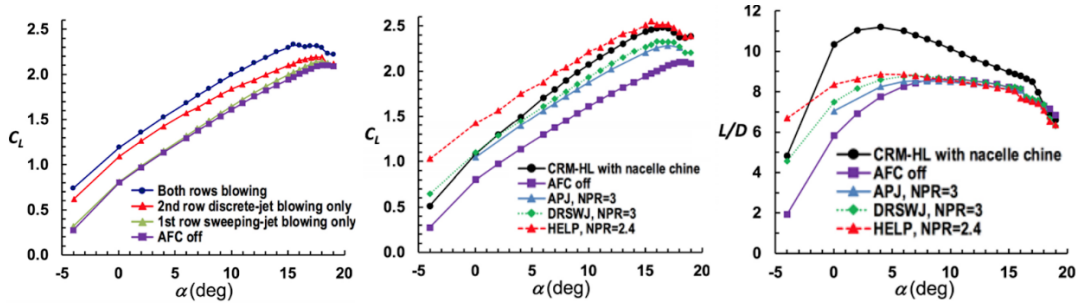


Figure 2-125(a) Effect of AFC on simple hinged airfoil (b) Effect of AFC on simple hinged airfoil compared to conventional flap [30].

Other aerodynamic effects are viscous wakes, shedding vortices, tip leakage vortices and turbulence fluctuation. When bursting viscous wakes from the slat passes through the high pressure gradient area, the maximum lift will be limited due to the rapid spreading of the viscous wake. Fig. 2-126 shows the total pressure field on the airfoil w/ & w/o AFC actuation on the slat. The results indicated that AFC actuation reduces the sizes of the wake at the trailing edge [126]. In addition, since the vortices of the bursting viscous wakes separate at a certain frequency, resonance will occur if the shedding off frequency matches the natural frequency of the object [121]. Another possible effect is tip leakage vortex (TLV), which is unsteady flowfields origin from the tip region of the fan blades. TLV is an important issue related to the design of turbomachinery since its unsteady behavior of vortex wandering, splitting and breakdown will influence flow feature. Despite that, the effects of TLV on AFCs are still unknown [127].

The above paragraphs demonstrate the overall effects of AFCs on airfoil. However, to implement AFCs on future aircrafts, more understanding of the design parameters, the interaction of AFC with flowfield, and the effects of AFC is needed. For instance, in Kim and Kim's [128] research, the lift performance of a synthetic jet AFC has no significant differences between different frequencies, but the flow features were completely different, shown in Fig. 2-127. The resonance characteristics in the cavity of the synthetic jet AFC are not clear, and the injection speed of synthetic jet is too low for high speed flow control. [128][129] The flow physics associated with thin bounded viscous layers, separated flows, vortical elements, systems of shocks, and expansion waves at actuators are still unknown [40]. For the interaction effects, the wall jets interact with high frequency AFC jet flows and outer flow turbulent and will affect the separation location, and this interaction is influenced by AOA, airfoil circulation, pressure distribution at the Coanda flap, and the leading edge geometry [130].

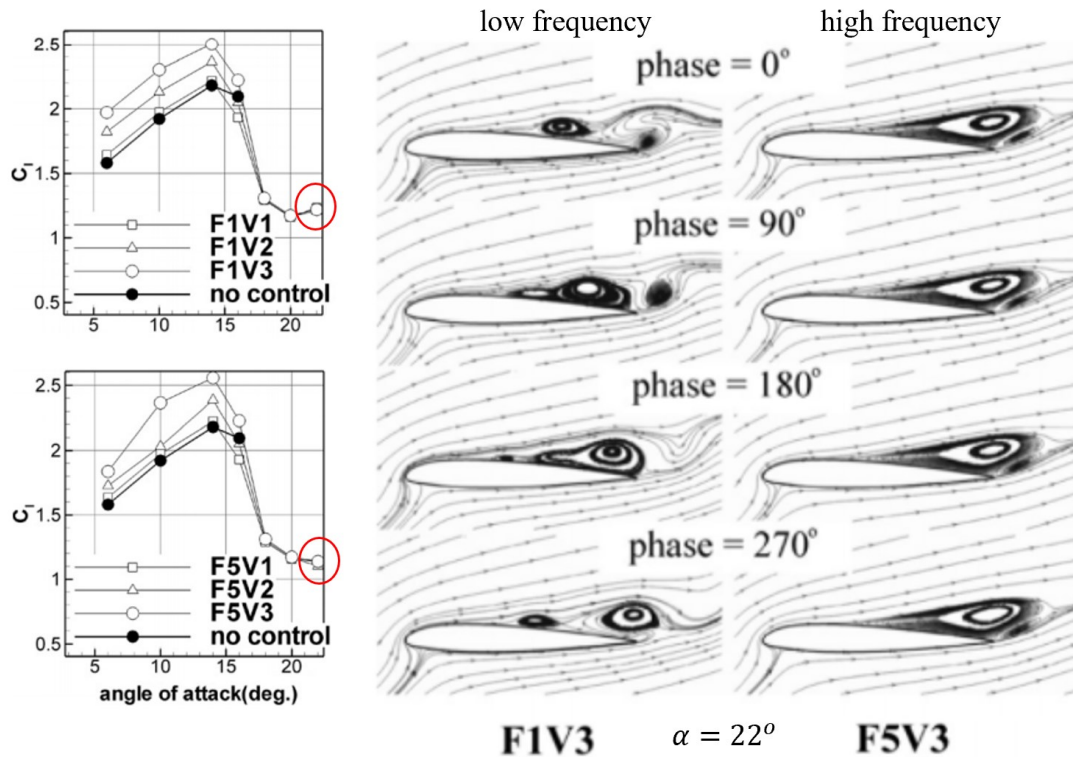


Figure 2-127 Effect & flow features of AFC w/ different operation frequencies [128].

The last step is to consider real life operation conditions during the development of AFC. For instance, the gap between wind tunnel to flight, Reynolds number and Mach number effect is important to all models tested in the wind tunnel. For AFCs, Reynolds number and Mach number effects relate to the input magnitude. Contamination such as rain, ice, and insects might enter the system. Simulated noise source in ultrasonic range is needed when testing since a flow control system should not generate additional contribution to aircraft noise [131].

In general, the potential for using AFCs to increase the performance is obvious, but the experiment in the HL-CRM, shows that replacing conventional flaps by simple hinged flaps with AFC is unavailable. This provides various research topics for studying the AFC on a HL-CRM like the mechanisms of AFC and the interaction in the flow field by flow visualization techniques, the influence of different design parameters by force measurements, AFC operation during

contaminations, and the aeroacoustic effects. There's still a long way to transfer laboratory work into real world applications.

2.12 LOW REYNOLDS NUMBER COMPUTATION FLUID DYNAMICS VALIDATION

The purpose of building a HLCRM is for CFD validation, because CFD workflows contain considerable uncertainties. These unqualified numerical uncertainties come from many sources including mesh generation, flow modeling (RANS/LES/DNS), boundary conditions, and residual convergence. Hence, the CRM and HLCRM model is carried out to acquire high-quality test data in 3D configurations to verify numerical results for commercial transport high lift configurations in the public domain since geometry and test data for current aircrafts is unpublished by aircraft manufacturers [1][132]. Fig. 2-128 shows the impact of CFD at Boeing. The green areas are the ones that have strong CFD penetration, violet areas have some knowledge, and the red areas are the target for future research. For instance, Fig. 2-85 shows good results for wingtips (green) from numerical computation. In addition, for icing in the violet area, CFD performs well in 2D, but has poor results in 3D icing. An example in Fig. 2-102 shows that the results in streamwise ice from LEWICE3D (3D icing software) act pretty well, but the configurations at incomplete scallops are bad.



Figure 2-128 Impacts of CFD from Boeing [133].

High lift devices are the main issue of HLCRM, and the complex flow near the leading edge and trailing edge is still unknown, especially in the near-stall conditions during landing and takeoff. In other words, the ability to predict the flow over high lift devices is poor. For slats, the separation behind slat tracks was usually wrong even when including brackets. CFD generated too much separated flow behind the outermost slat brackets, and other slat brackets on the outer part of the wing [134]. For flaps, numerical and experimental results match well between different CFD methods for low AOA up until stall, where there is a great difference. All CFD methods under-predicted inboard separation and over-predicted outboard separation occurring in the post-stall region [135]. The nacelle/pylon installation is an investigation done by Airbus-Deutschland in the EUROLIFT project and JAXA on the JSM model, aiming to understand the effects of an aircraft model with and without a nacelle/pylon. In the numerical side, CFD value agreed very well with no-nacelle/pylon experimental lift curve data, but not with the nacelle/pylon case [134]. There is a lack of accurate prediction of the stall characteristics, and high angle lift. This might be due to

the miss-prediction on the flow transition and separation at surface flow patterns and inboard wing fuselage, especially a lift breakdown observed downstream of nacelle. In Fig. 2-129, results showed that RANS code had too much separation near wingtip, but not enough separation near the wing root [136][137][138]. Fig. 2-130 shows the under-estimated separation in the inboard wing fuselage region.

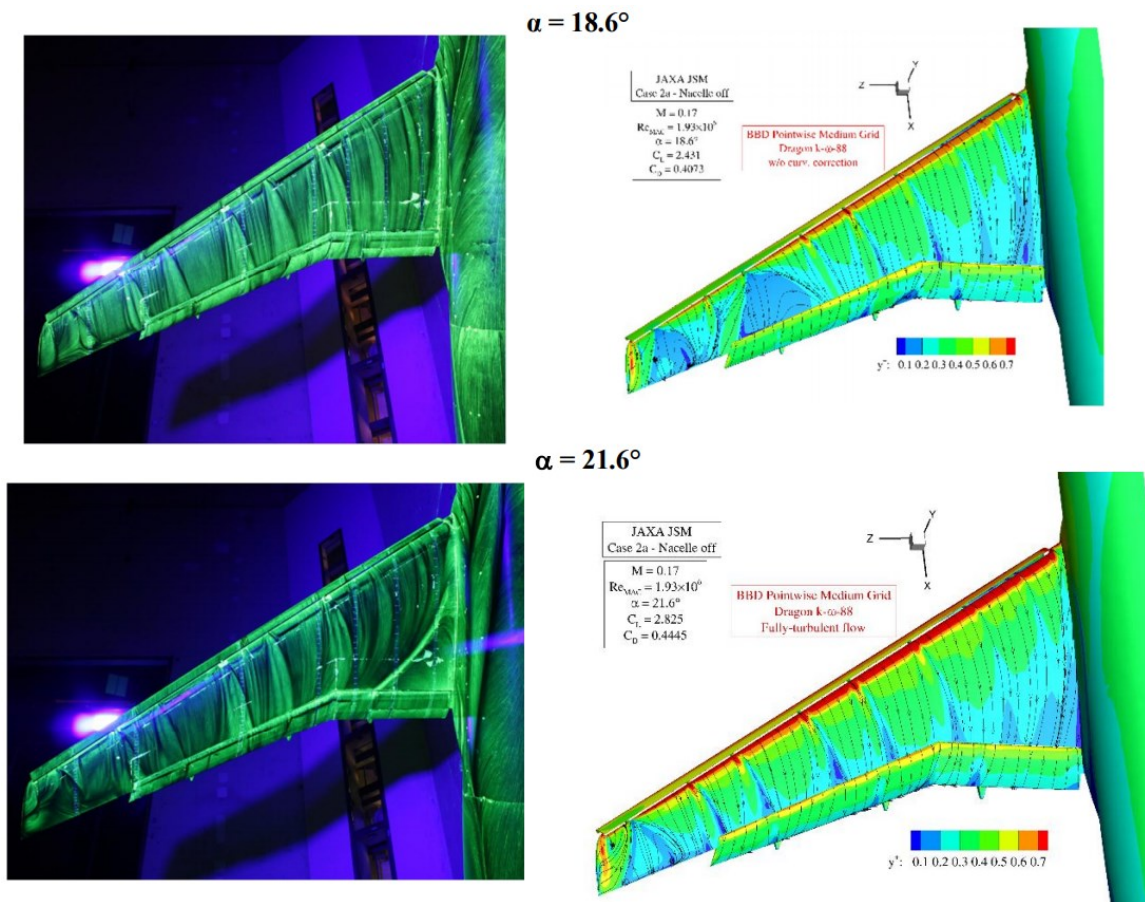


Figure 2-129 Experimental and numerical results of surface flow patterns [137].

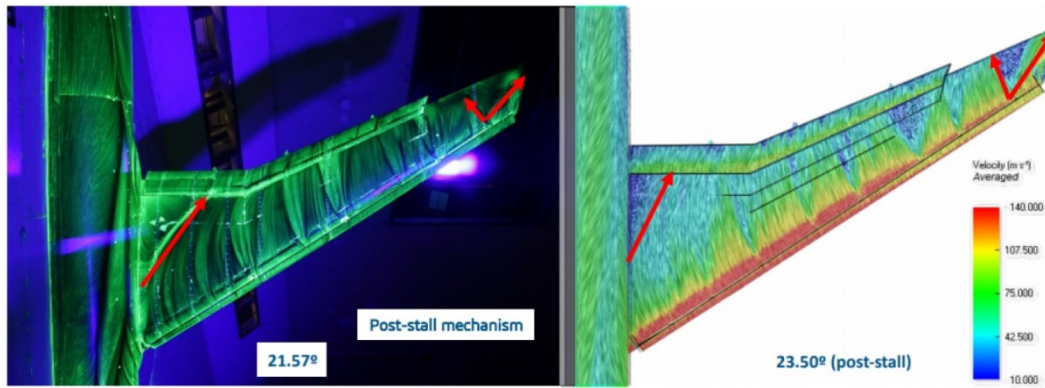


Figure 2-130 Separation in the inboard wing fuselage region from experimental and numerical results [137].

The reason for the inaccurate prediction to transition and separation, and turbulence is due to the lack of a theoretically based, hybrid RANS-LES turbulence simulation capability. Current transition process is estimated by empirical equations or coupled PDE solutions of stability equations. A generalized transition prediction capability is needed to fill the missing transition models in the current process. Additionally, more specific testing data from a fidelity geometry is needed to validate and improve turbulence prediction. Besides fluids, the ability to couple CFD with other analysis such as acoustics, structures, heat transfer are limited by a lack of general methodologies [132]. Lastly, the gap between CFD and experimental results is not fully due to the lack of knowledge in aerodynamics. Previous CFD simulation uses free-air CFD run with corrected semi-span data might lead to a considerable uncertainty. Future models should include wind tunnel testing geometry and boundary conditions [133].

2.13 CAPABILITIES FOR HINGE MOMENT MEASUREMENTS

For aircraft control systems, hinge moment is the moment that needs to be overcome by either the pilots or by other methods like the control system. (Fig. 2-131). For aircraft design, the hinge moments used for the early design phase to design control surface actuators are from

appropriate results obtained from CFD, which were inaccurate and overestimated (Fig. 2-132). These estimated values will lead to an overestimated control surface actuator, increasing the size, weight and cost. This is because accurate hinge moment can only be measured by strain gauges on large scale wind tunnel models, which is usually built in the late process. Hence, the interest in hinge moment relies on fundamental studies on the parameters that influence hinge moments, and develop new methods that could measure hinge moment on small aircraft models [139].

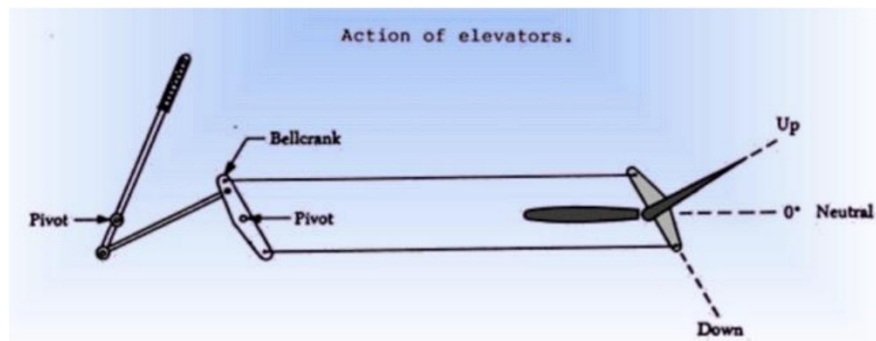


Figure 2-131 Illustration of the relationship between hinge moment and control system [140].

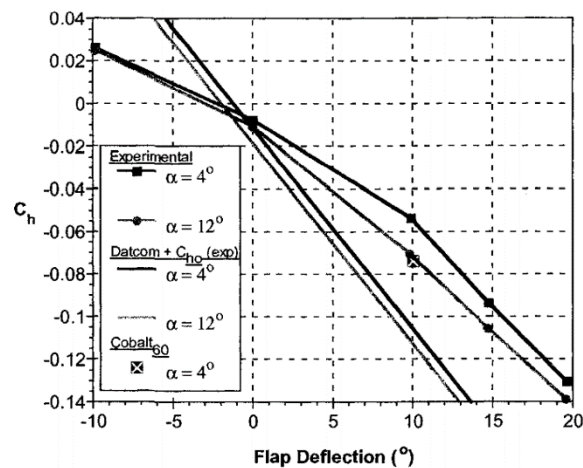
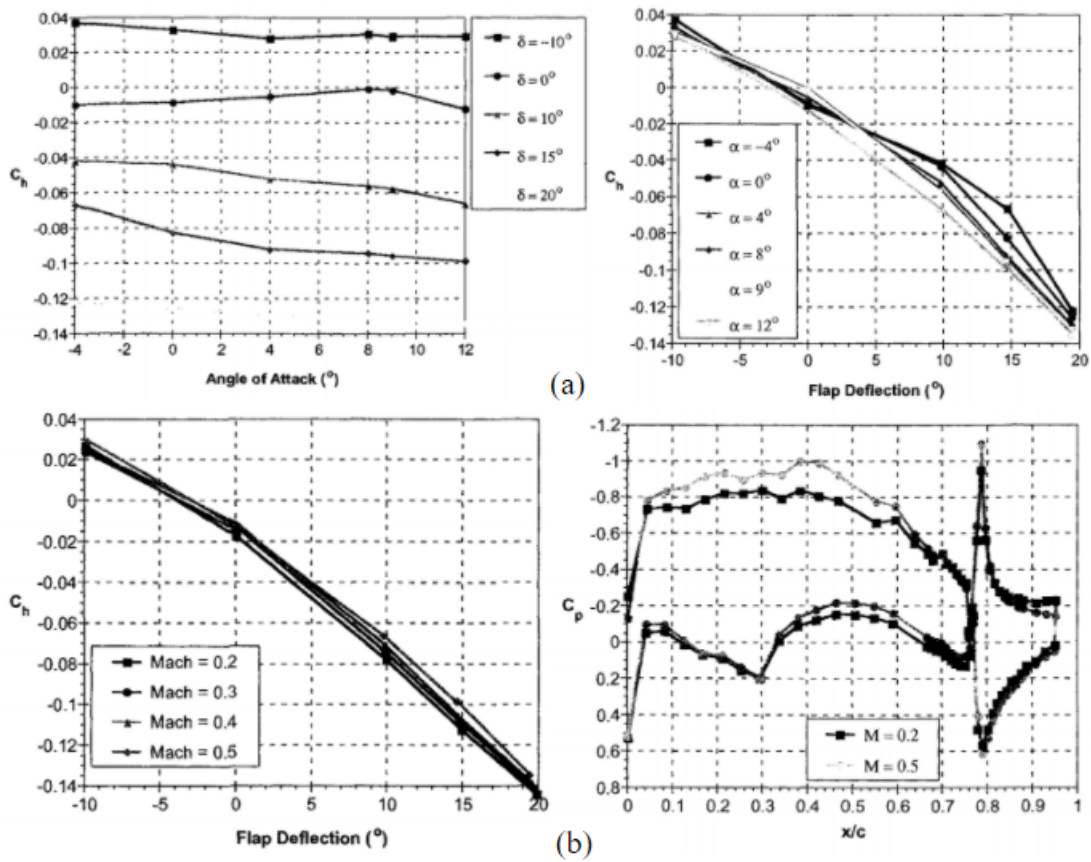


Figure 2-132 Comparison between measured and calculated C_h (Datcom, Cobalt) [141].

Grismer et al. [141] tried to improve the calculation by investigating the factors that influence hinge moment. A 2D wing was placed in the wind tunnel in different conditions. Surface pressure and hinge moments were measured by pressure tapes and a torque cell. Fig. 2-133(a)

shows that flap deflection has a major influence on hinge moments, where the angle of attack only slightly decreases the hinge coefficient as the angle of attack increases. Next, Fig. 2-133(b) shows that C_h increases as Mach number increases. This might be due to separation on the wing surface. In Fig. 2-133(c), by comparing the hinge moment at different flap deflection, and angle of attack, and the pressure distribution of different flap gaps shows that the spacing between main wing and trailing edges have nearly no influence on the hinge moments in most cases except at low angle of attacks.



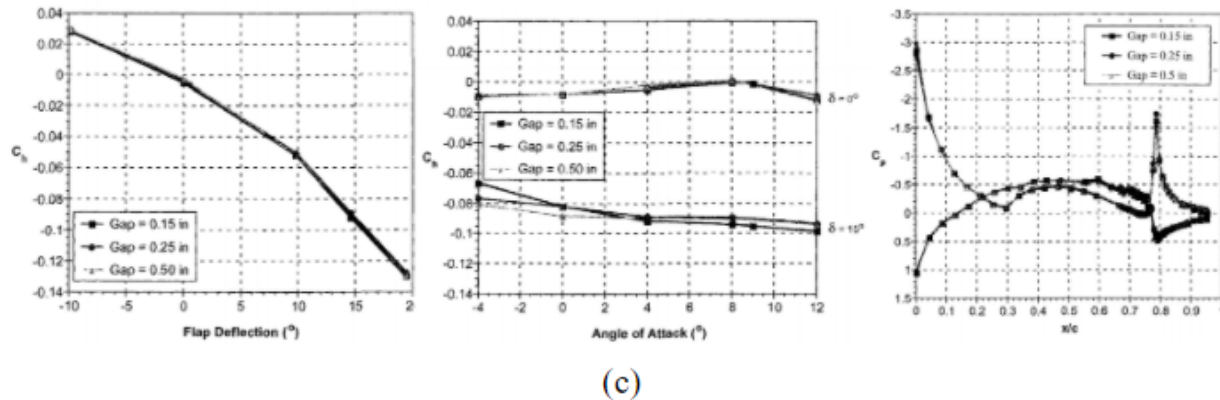
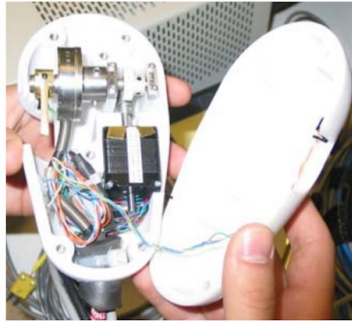
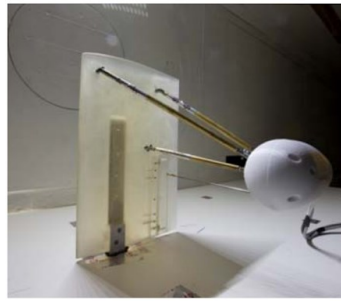


Figure 2-133 Effect of (a) angle of attack & flap deformation (b) Mach number (c) flap gap on hinge moments [141].

The Conceptual Aircraft Hinge Moment Measurement System (CAHMMS), developed by the California Polytechnic State University at San Luis Obispo is one way to measure hinge moments for small scale aircraft models. Figure 2-134(a) shows the CAHMMS measurement system: a torque transducer was suspended below the aircraft model by an adjustable tripod. Hambrik and Thomason [139] validated and verified CAHMMS using CFD and wind tunnel data from Lockheed Martin and found out that the CAHMMS system has an 5% error for small control surface deflections, and 12% error for large deflection. An example of the comparison of the measurement results is shown in Fig. 2-134(b). Despite the fact that this method can measure hinge moments for small scaled aircrafts, it is not perfect. Future work such as improving the error and to understand the aerodynamic effect of the tripods on the test model needs to be done in order to improve the method.

(a)



(b)

		C_l	C_d	C_h
Exp. Using Force Balances →	TN 2080	0.38	0.06	-0.33
Computed →	DATCOM	0.420	0.082	-0.426
CFD →	CFD wing only	0.576	0.069	-0.441
	CFD with CAHMMS	0.102	0.063	-0.260
Exp. using CAHMMS →	LTWT with CAHMMS	0.322	0.163	-0.306

Figure 2-134 (a) Photos of the CAHMMS system (b) Comparison of the results [139].

Moiré interferometry can be used to measure deformation; the principle can be explained by Fig. 2-135 (a). The reference ruling was first visualized by the stripes on the surface at position S_1 , when the surface moves to position S_2 , the shift will lead to a change in the interferogram of the surface. Deformation of the surfaces can then be calculated by the transition of both rulings. Baumann and Bütetfisch⁴ tested the Moiré interferometry system on the wing of a hypersonic vehicle (Fig. 2-135 (b)) with $\alpha = 7^\circ$, and $\delta = 10^\circ$, the results were verified using strain gauges and a force balance. The sketch in Fig. 2-136 displayed an example of the topology of an interference surface. The results in Fig. 2-137 shows that the change in angle of attack and flap deflection matches the data from force balances and strain gauge very well, especially at low speed.

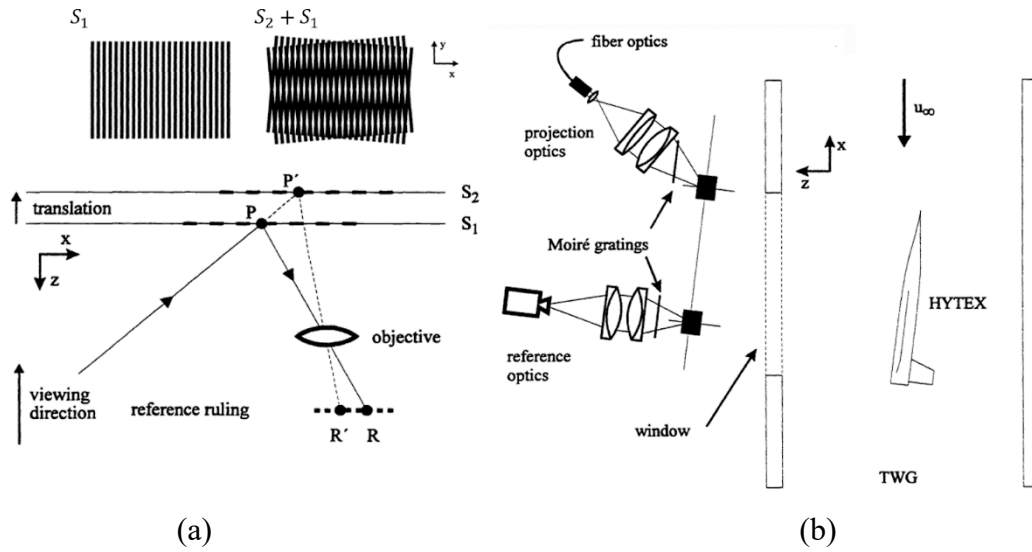


Figure 2-135 (a) Principle of using Moiré interferometry (b) Setup of the experiment [142].

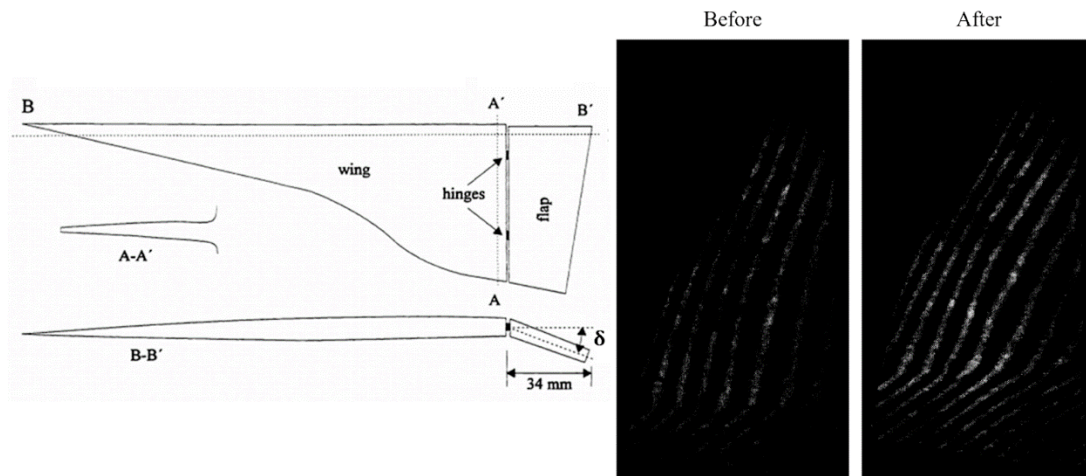


Figure 2-136 Interferograms of the testing model [142].

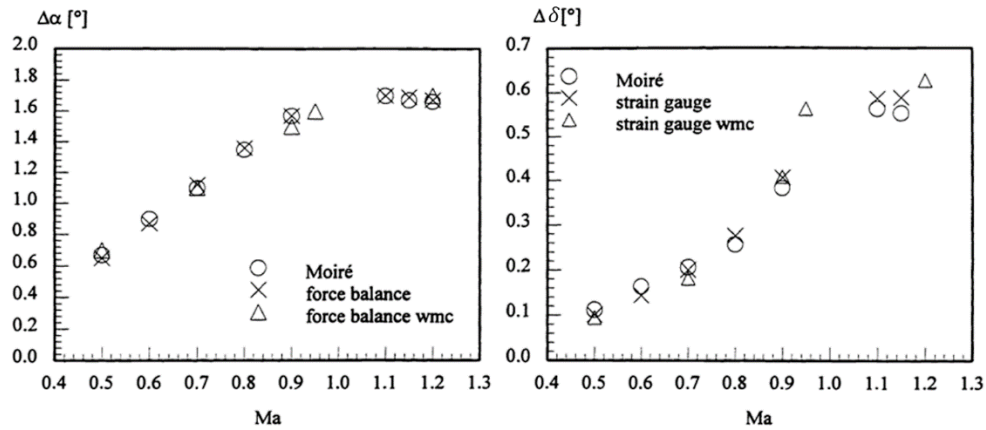


Figure 2-137 Comparison of Moiré's result to force balance and strain gauge [142].

With the development of science and technology, piezoelectric sensors, which is a MEMS (Micro-Electro-Mechanical Systems) technique, have the potential to measure hinge moment. While force or momentum is applied to the sensor, the piezoelectric quartz wafer will respond to electrical charge and generate output voltage. Hence, with the output voltage, hinge moment can be obtained from the relationship between the output voltage and the input moment (Fig. 2-138). The piezoelectric sensor has only been calibrated by Ren et al. [143] by applying a static force on a small airfoil (Fig. 2-139). More investigations needed to be done to confirm the availability of using piezoelectric sensors to measure hinge moment.

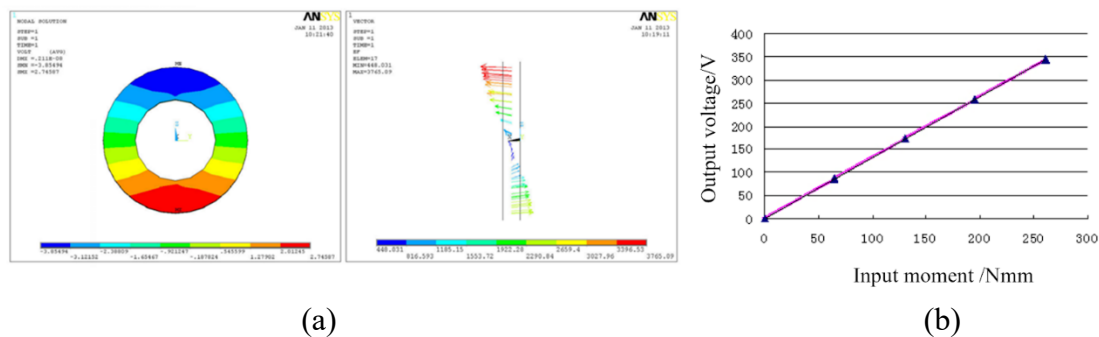


Figure 2-138 (a) Potential distribution of the electric field intensity and the effect of moment
(b) Relation between the input moment and the output voltage [143].

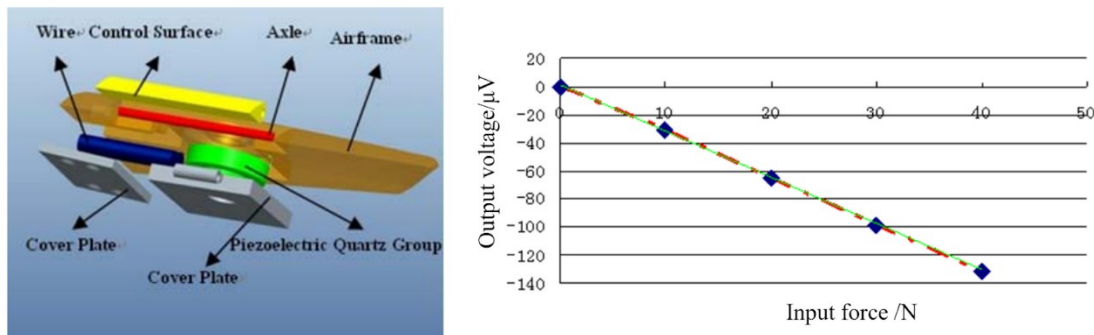


Figure 2-139 Setup and the calibration results of the piezoelectric sensor [143].

Using hinge moment to predict stall might be another application for hinge moments. Ansell and Bragg [141] observed large increase in hinge moment unsteadiness while studying experimental hinge moment data from a NACA 3415 airfoil model approached to the stall angle of attack under clean and iced configurations. From Fig. 2-140, large increases in C_h occurs at stall for the simulated glaze ice and simulated rime ice case, this indicates a correlation between flow separation and the increase in hinge moment unsteadiness. A stall prediction algorithm system was then invented to provide stall warning before stall happens by a combination of three methods (moment-based detector, derivative-based detector, and spectrum-based detector). From Fig. 2-141, the output of these three methods for the clean and iced cases shows that with an increasing AOA, the output display level increases monotonically, and remains consistent on a given preset. This allows the use of a single threshold method to predict the wing stall.

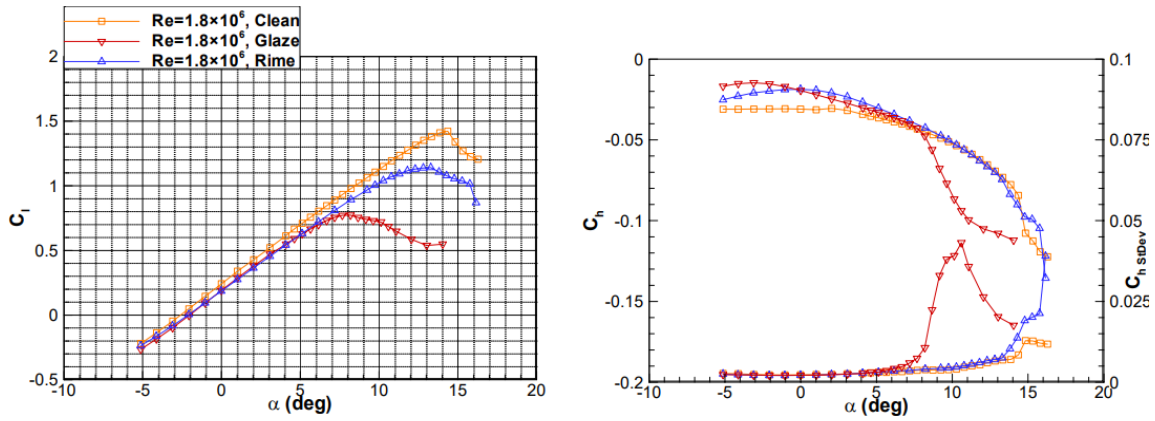


Figure 2-140 NACA 3415 performance with simulated ice contamination [144]

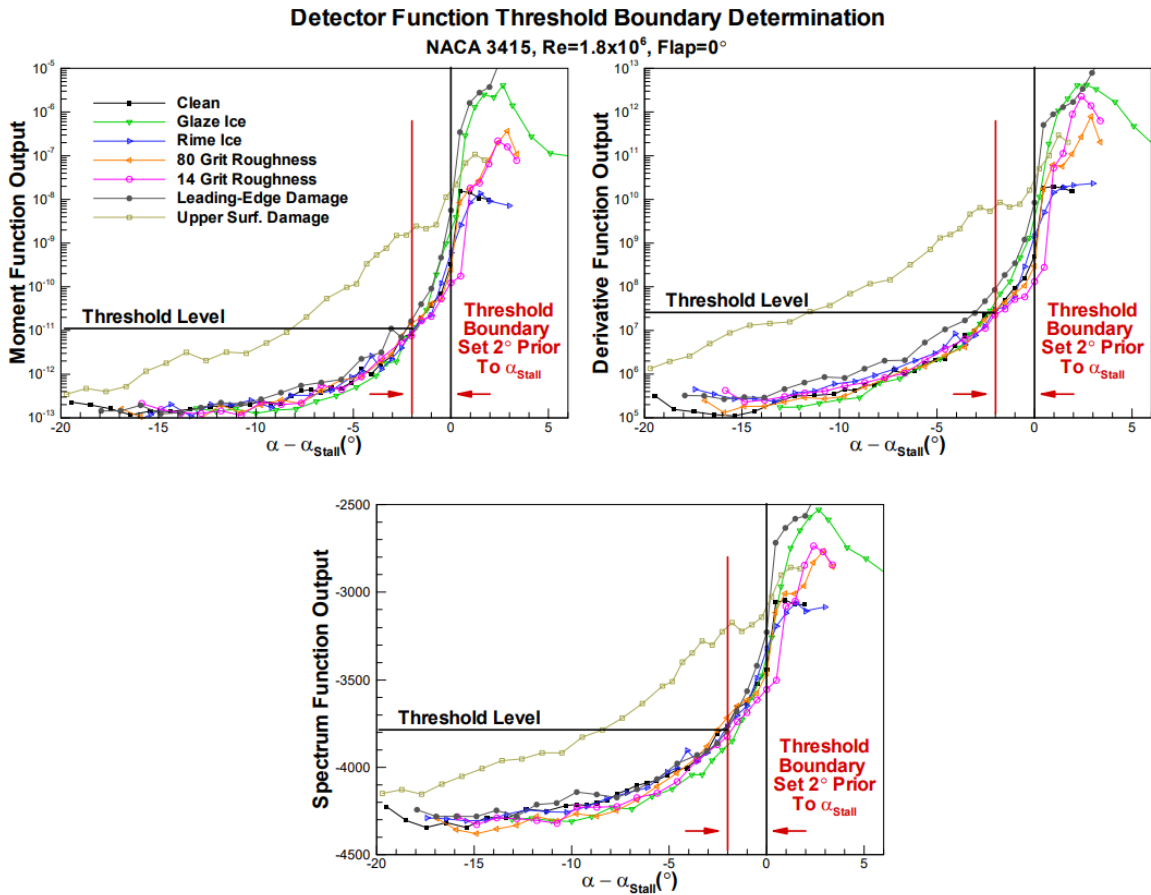


Figure 2-141 Detector methods output for NACA 3415 at different configurations [144].

Hinge moment calculation is an important topic in aircraft design. However, not much research has been done to improve the accuracy. Recent research is mostly based on numerical

simulations with a lack of comparison to measurements from strain gauge, and the investigation done by Grismer et al. [141] only includes C_h and C_p with missing flow field information. Moreover, the use of new developed techniques is rare and large errors still exist. The method from Ansell and Bragg [144][145] to predict stall is a great idea, and has been proved on a NACA 3415 airfoil. However, there are still some expectations while applying the system to another airfoil [145]. These expectations are probably due to different mechanisms that cause the stall, like different airfoils configurations, contamination, and 3D effects [144]. More research like obtaining 3D airfoil, flow structure and dynamic/contamination data needed to be done to improve the system [146][147]. The UW KWT does not have the ability to measure hinge moment. However, this is a very important topic and not much research has been done to predict hinge moment accurately. Therefore, the author believes that the ability to measure hinge moment accurately should be added to the UW KWT.

2.14 AIRFRAME NOISE TESTING

With the invention of bypass turbofan engines, the noise from engines has then been reduced, making noise originating from the turbulent flow around the airframe during approach and landing conditions more important. For current civil aircrafts, major airframe sources can be listed as follows: high lift systems (slat, track, and flap side edge), landing gears, and interference effects (jet-flap, gear wake flap, etc.) [148]. Fig. 2-142 is an overview of noise generating mechanisms of conventional mid-range transport aircraft with turbofan engines, and Fig. 2-4 depicts the noise breakdown of the airframe noise components [149]. Since the impact of landing gears on noise contribution is discussed in Chap. 2.04, this article introduces the effects of high

lift systems, the interference between different components, and other effects like engine shielding, and scaling effects.

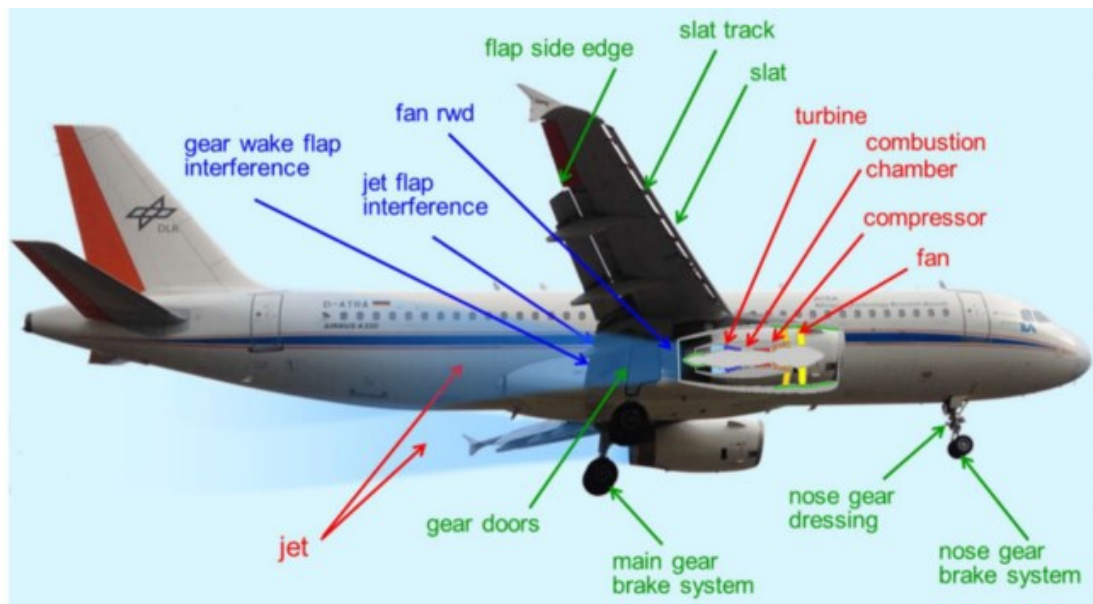


Figure 2-142 Depiction of the most relevant noise sources on civil aircrafts [149].

From Fig. 2-142, leading edge acoustics are related to slats and slat tracks. The influence of slat tracks has been mentioned in Chap. 2.02. For slat noise, acoustics originate from two characteristics: high frequency noise caused by vortex shedding at the trailing edge of the slat, and low-mid frequency noise from the oscillation of the cove vortex in the cavity (Fig. 2-143(a)). Complex flow behavior in the slat core such as vortex merging, secondary vortices, reattachments makes an accurate simulation of the slat noise challenging [150]. Some examples of simulating slat cove flow and noise were shown below. Choudhari and Khorrami [151] provide a 3D model of the slat cove flow using large eddy simulations shown in Fig. 2-143(b). The vorticity structure in the 3D simulation agrees pretty well with the PIV results: vortex merging and secondary vortex from the slat cusp is clearly shown, but the recirculating vortices were overemphasized. Housman et al. [152] simulated slat noise using different structure grids. The results in Fig. 2-144 shows that

the noise from 1-3 kHz were captured by numerical results, in spite of some fluctuations. The simulation of the peak at 25 kHz was shifted a bit left and the results were amplified.

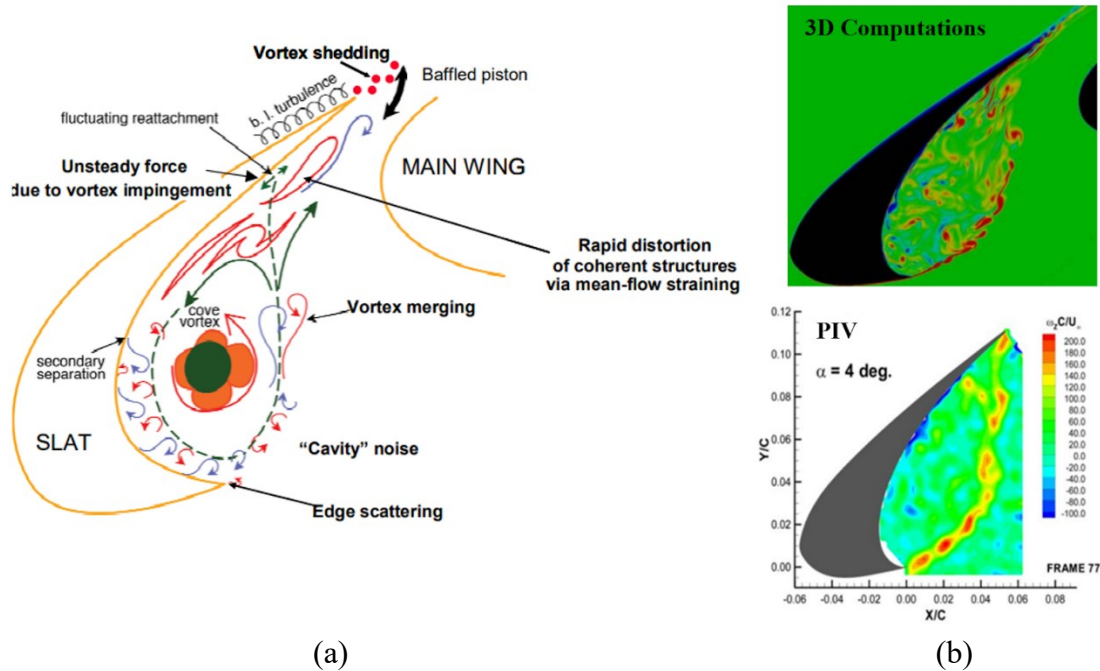


Figure 2-143 (a) Concept of the slat noise mechanism (b) Comparison of the computed instantaneous velocity field to PIV measurements [151].

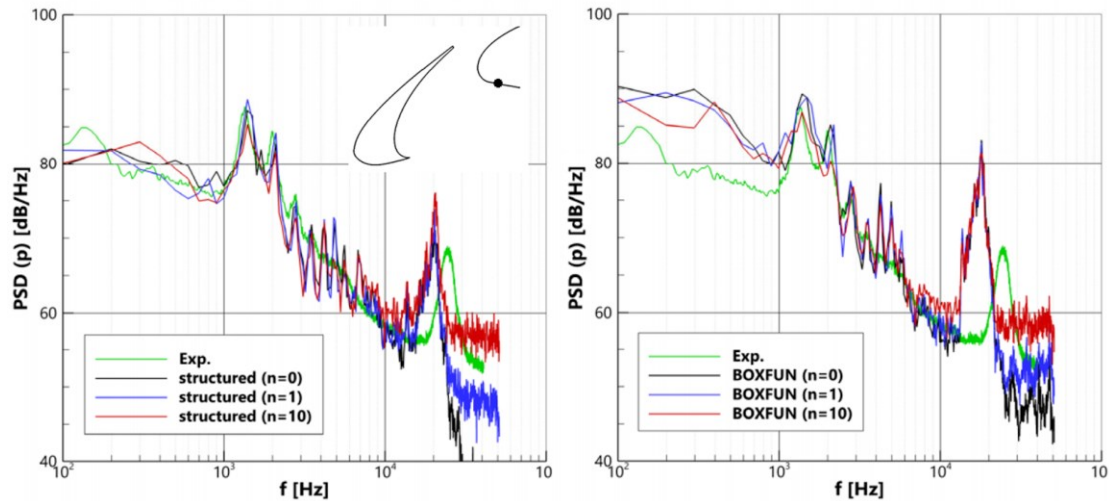


Figure 2-144 Power spectral density of the surface pressure from CFD using different grids [152].

After understanding the mechanisms of slat acoustics, active and passive noise reduction techniques have been tested to reduce noise. Silva [153] studied the noise reduction effects of plasma actuators on slats. In his study, applying DBD actuators on the cusp of a slat produces the best results. In Fig. 2-145, 3.35 dB in the overall sound pressure level, and 12 dB in the main peak were reduced when DBD-actuator was operating. Smoke flow visualization shows that the ionized air was able to shift the reattachment position. Zhang et al. [154] tested the effect of three passive control methods: slat extension, gap filler, and cove filler. Fig. 2-146 indicates that all three configurations of slat extensions have only a slight impact on noise cancellation. The results from gap fillers (Fig. 2-147) shows that the device could suppress the peaks for $St_s < 3$, especially at low angles of attacks, but it starts to lose control after $St_s > 3$. In Fig. 2-148, results shows that the cove filler is effective during low angle of attacks, when angle of attack and Reynolds number increases, the broadband noise level increases, and cause additional noise contribution for $St_s > 3$.

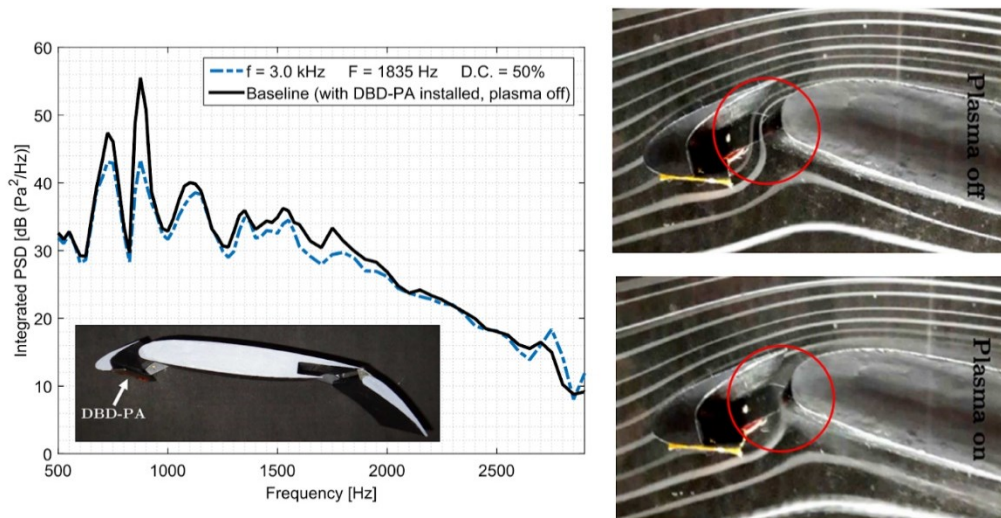


Figure 2-145 Effect of plasma actuators on noise reduction [153].

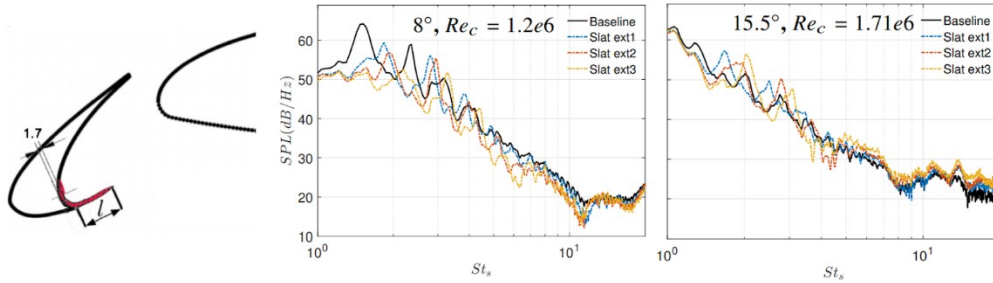


Figure 2-146 Effect of slat extension [154].

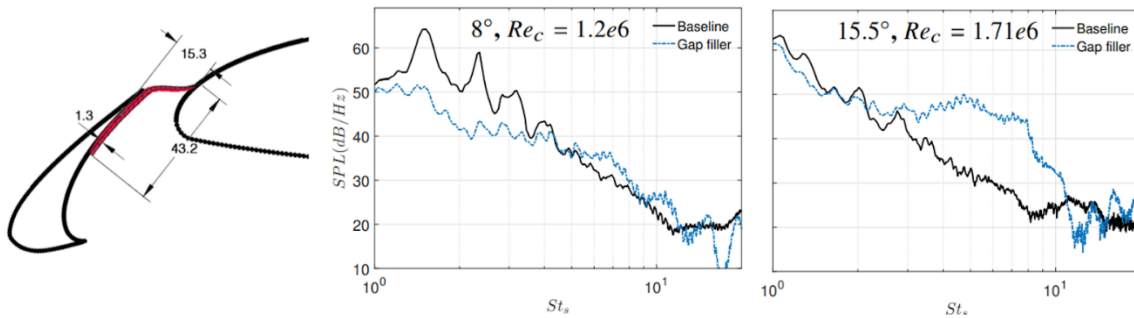


Figure 2-147 Effect of slat gap filler [154]

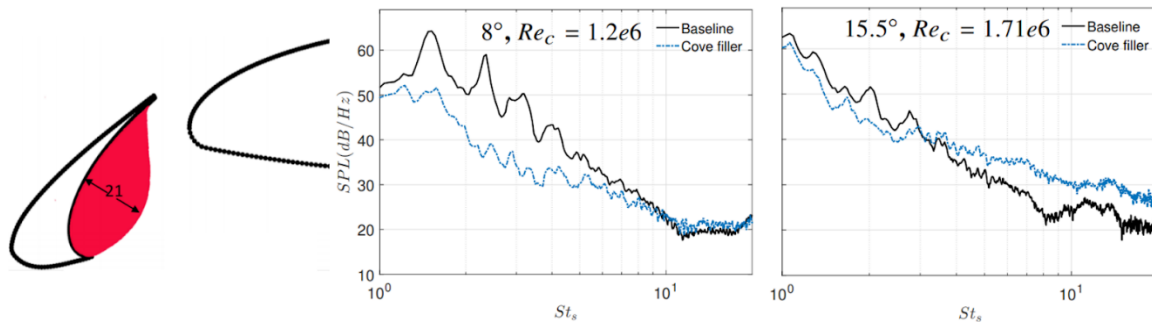


Figure 2-148 Effect of slat cove filler [154].

On the other hand, the main interest from flap noise is flap side-edge acoustics. Flap side edge noise is related to two sources: fluctuations and instabilities located at the front of the side ridges and in the shear layer, and large eddies merging at the midway and at the rear of the side edge. As shown in Fig. 2-149, side-edge fluctuations are induced by the shear layer at the pressure side of the flap edges that travels over the side edge generating vortices that are the main contributors to the mid and high frequency noise. The primary and secondary vortex then merges at the downstream, generating low-mid frequency noise. Despite that these complex vortices make

accurate sound source simulation challenging, current understanding still allows us to perform good numerical results. Fig. 2-150, 151 shows a flap side-edge noise simulation using RABS-based source modelling done by Kolb et al. [155]. Pressure distribution and velocity components show similar trends compared to the measurements, even though the error increases at the flap trailing edge where separation happens. Furthermore, the SPL spectrum shows pretty good agreement, especially in the 60 m/s case with the wind tunnel measurements and another semi-empirical prediction.

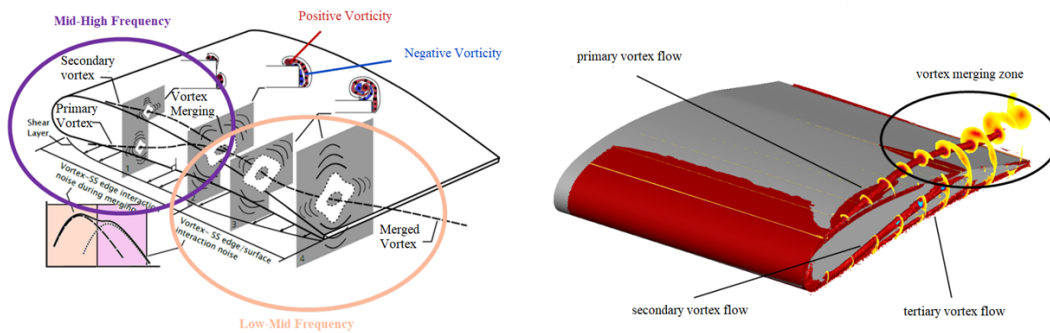


Figure 2-149 Overview of the flap side-edge noise source mechanism [155].

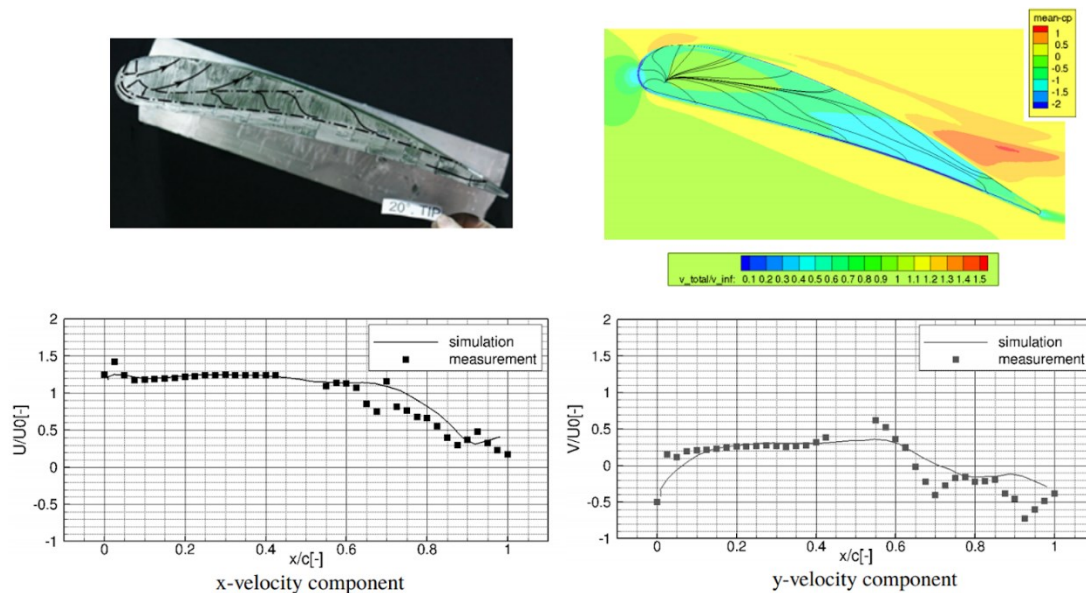


Figure 2-150 Comparison of the computed surface pressure distributions and velocity components with experimental data [155].

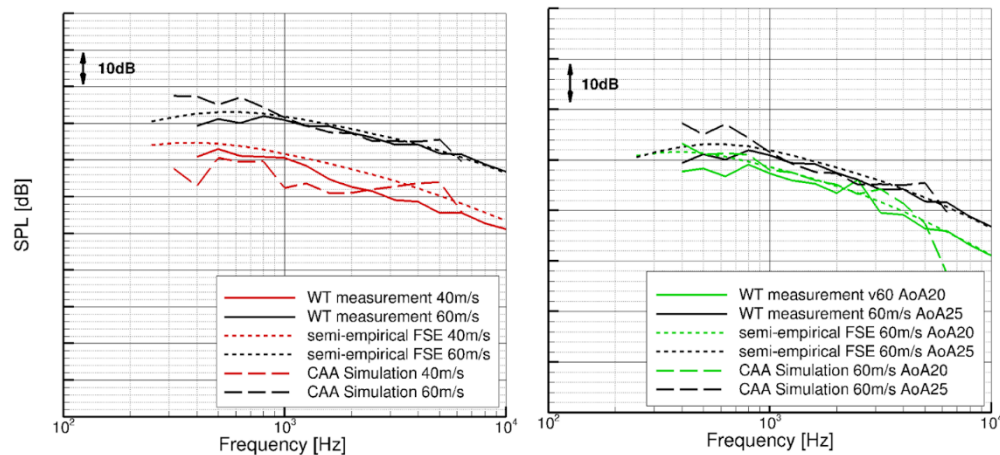


Figure 2-151 Comparison of the simulated SPL spectrum with wind tunnel measurement [155].

For noise cancellation on flaps, JAXA achieved great progress by applying Protruding Rounded Lower edge (PRLE) and small barriers/vortex generators on flaps. PRLE can reduce the noise generation by removing turbulence on the flap side edge and shifts the vortex from the upper side-edge away (Fig. 2-152). Flight test results show that PRLEs have a significant impact on noise contribution [156]. The idea of putting small barriers or vortex generators on the upper surface near the flap leading edge tip is to weaken the roll-up vortex from the flap side-edge by generating vortices from the obstacles on the flap tip. Fig. 2-153 displays the results of applying small barriers on airfoil from Isotani et al. [157] Wind tunnel results show that small barriers can have about 1-2 dB noise reduction.

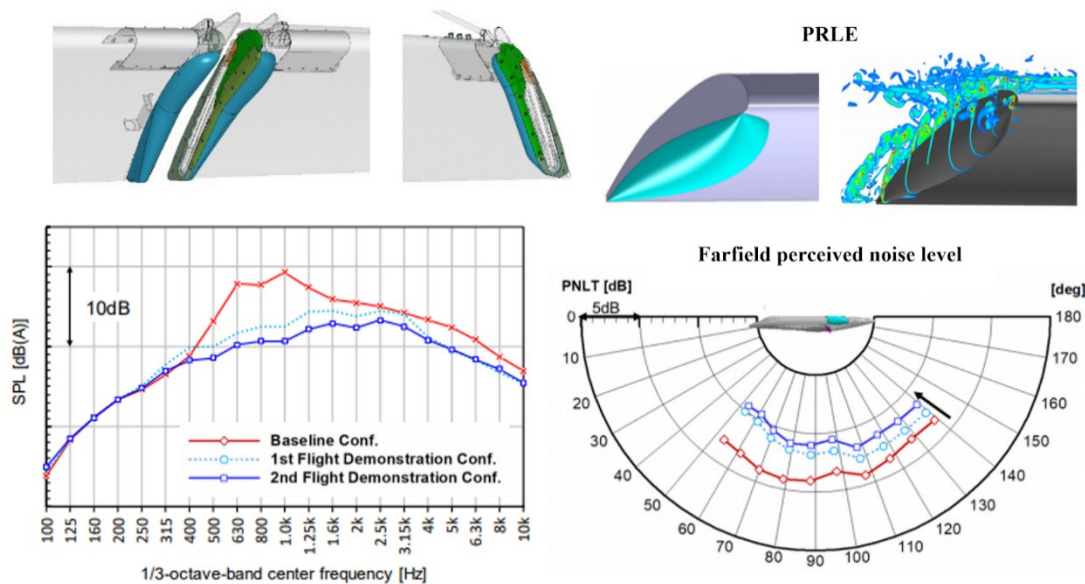


Figure 2-152 Effect of applying PRLE to flaps [156].

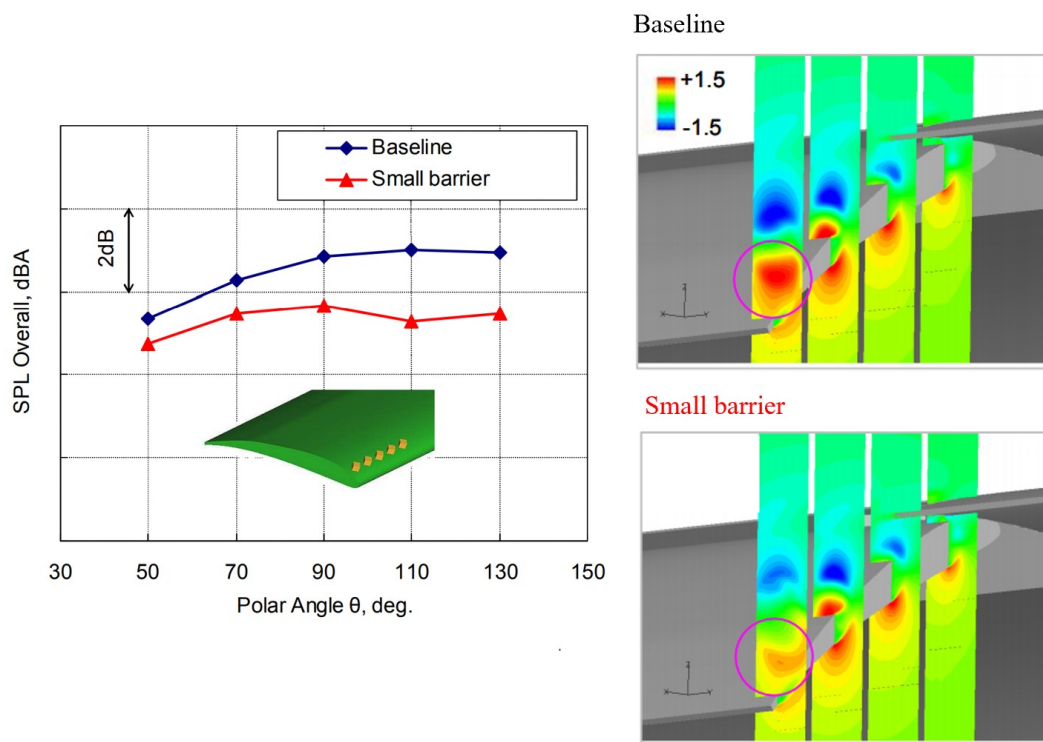


Figure 2-153 Effect of applying small barriers to flaps [157].

Interaction effects between each individual noise source will directly influence the noise generation, resulting in generating new noise sources or zooming the original noise source. An

example of the interaction effects between engine and flaps is shown in Fig. 2-154. The total noise level of the flap wing near the jet increases significantly compared to the noise from a jet only. Moreover, by comparing the narrowband spectra results shows that the jet-wing interaction effect is producing medium and high frequency noise [158]. The Joint DLR & TU Delft Aviation Noise Workshop have identified important interaction effects of noise sources that scientists have a low theoretical understanding. For instance, engine pylons might have interaction with aircraft wings due to different pylon design and the engine location, jet flow might react with flaps due to different pylon design/position, flap-jet distance, and Mach number, and the flow conditions around flap and slat might be influenced by spoilers and landing gears [159].

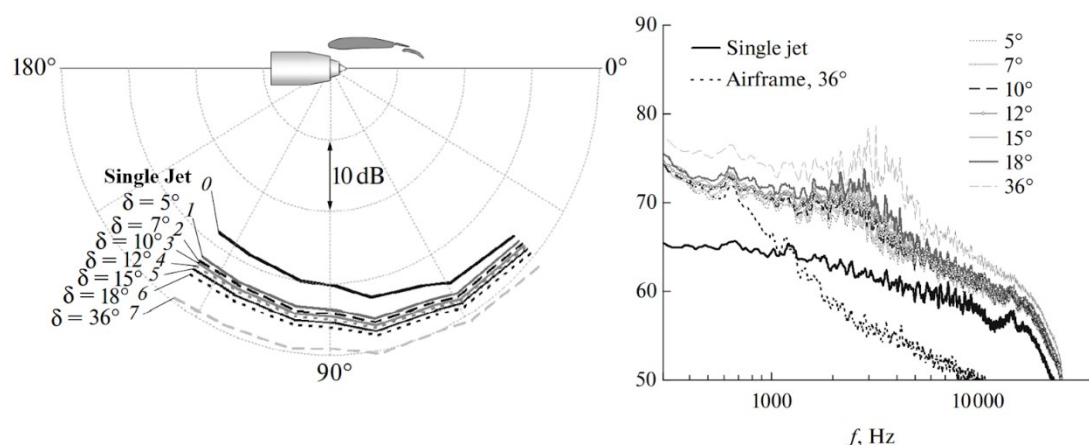


Figure 2-154 Comparison of isolated jet noise and jet-wing interaction noise [158].

For engines, although the noise of aircraft jet noise nowadays has fallen to a level comparable to aircraft noise, aerodynamic sound related to engine is also an issue [160]. Engine shielding effect is the effect of noise from engine placed, since acoustic waves generated from the engines will be refracted by the mean flow past the moving aircraft [161]. By placing engines at different locations, the noise measured from the far-field varies. Fig. 2-255 shows a comparison of noise spectrum for engines placed at different locations [162]. Despite that lots of research have

been focus on shielding effect experimentally and numerically, the theoretical understanding of shielding effect is still low [151].

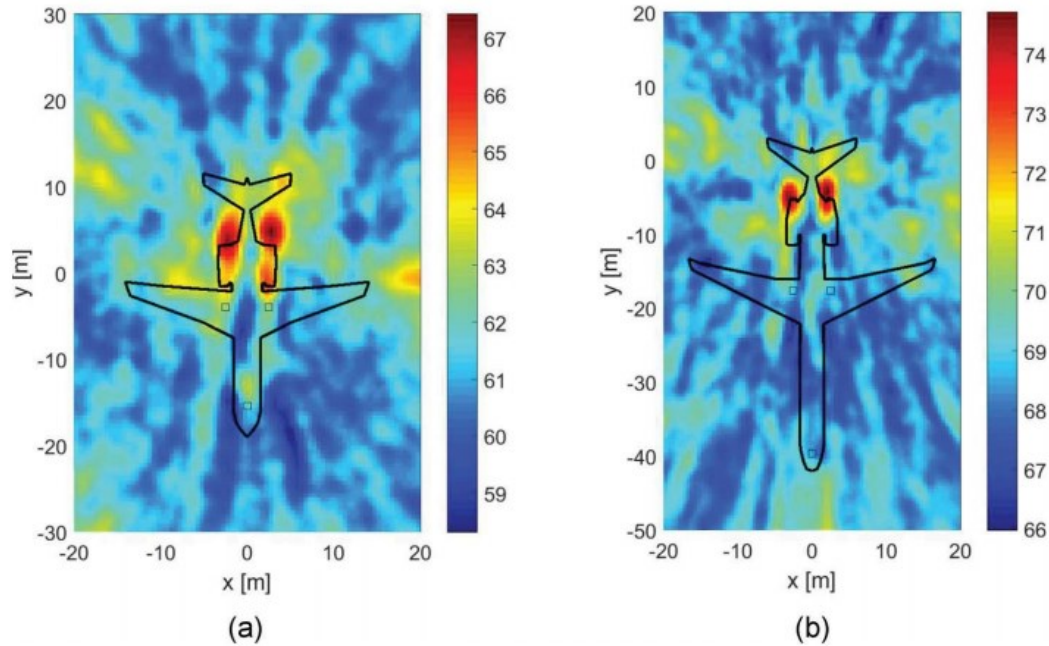


Figure 2-155 Noise spectrum for engines placed at different locations [162].

Lastly, another main influence to the whole aircraft acoustic performance is the scale effect. Experimental results showed that noise generated from the aircraft are different from aircraft models and full-scale flight tests. Fig. 2-156 showed an example of the noise source locations of a Boeing 777 aircraft in flight and comparisons with past scaled model results. It is obvious that noise distributions are different for the flight test and scaled models at both low frequency noise and high frequency noise [163]. The cause of these results is still unknown, which means the scale of the HL-CRM is important for this issue.

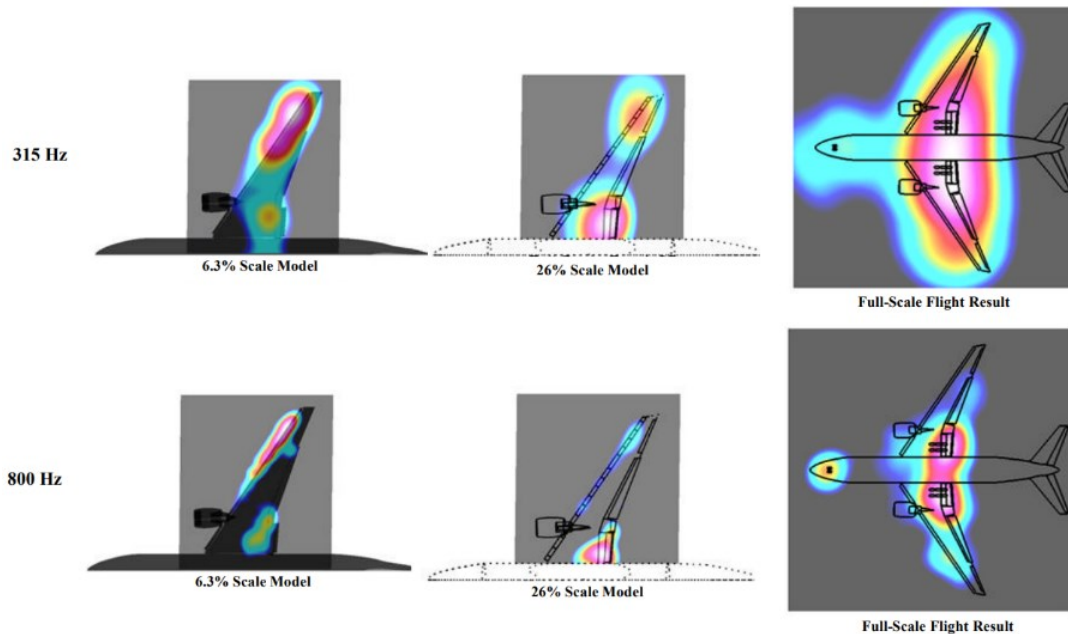


Figure 2-156 Comparison of airframe noise from model and full-scale aircraft [163].

In general, the knowledge in slat and flap aeroacoustics allows CFD to perform good numerical results. More investigation needed to be done, especially in the vortex shedding in slats, and the vortex merging and separation on the flap trailing edge in order to provide a more accurate model and better high-lift design. The effect of reducing slat noise is very poor, only the cove filler appears to provide good noise suppression in low angle of attacks. More experiments need to be done to study the flow field in the leading and trailing edge, the interaction effects, and to find ways to reduce slat noise by using flow visualization techniques to acquire data in the flow field, and microphone arrays to obtain acoustic data.

2.15 CAPABILITIES FOR LEADING AND TRAILING EDGE POSITIONING STUDIES

High lift devices are the components at the wing leading edge and trailing edge that could provide additional lift during takeoff and landing. Fig. 2-157 shows different types of high lift devices that have been frequently used in current civil aircraft. The difference between these

devices will not be discussed in the thesis. This chapter will only focus on a few of the important parameters that influence their performance characteristics. According to Balaji et al. [165], one percent increase in the maximum lift coefficient could provide additional payload of 22 passengers. Hence, increasing the efficiency of the high lift system is extremely important. Apart from using flow control devices to increase the lift, operating slats and flaps at the optimum position is also a way to provide more lift, less drag, and less noise in flight. The following article explains the aerodynamic and aeroacoustic effect of different leading and trailing edge positioning and some effects that need to be considered while deciding the best position for flaps and slats (the riggings for the NASA HL-CRM is listed in Chap. 2.01).

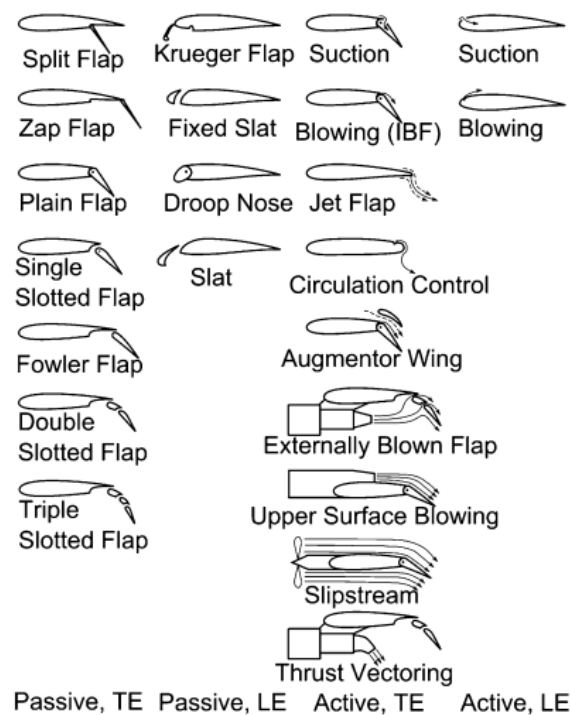
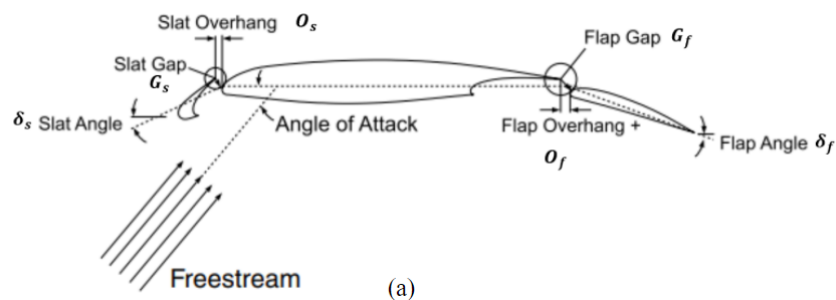
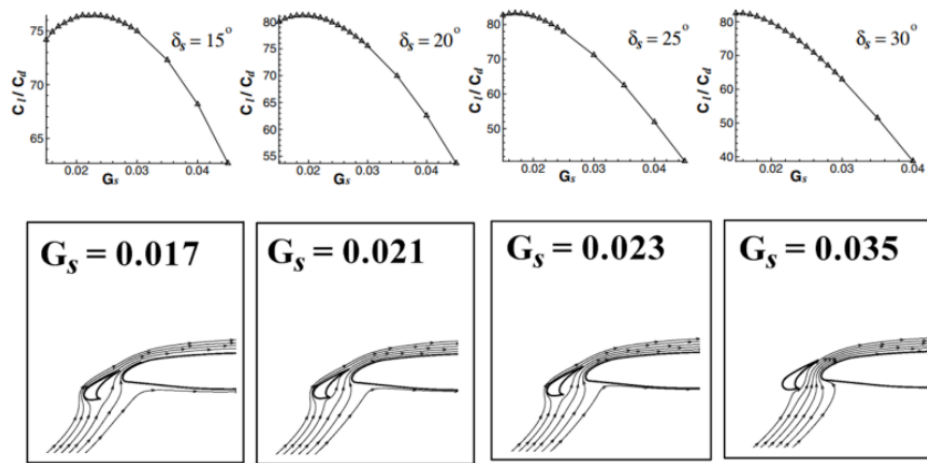


Figure 2-157 Different types of flaps and slats [164].

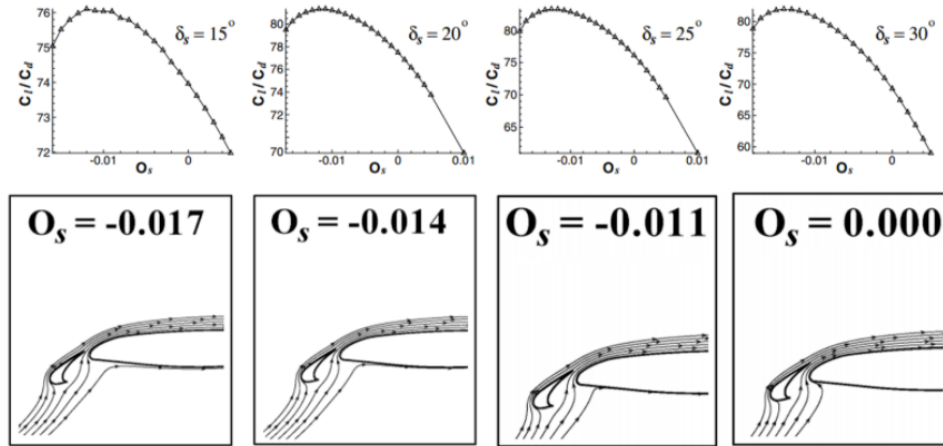
In order to achieve the above goal, the aerodynamic effects related to leading edge and trailing edge positioning must be studied. Balaji et al. [165] studied the effect of slat and flap positionings on lift and drag using RANS simulation. The definition of the model is shown at Fig.

2-158(a) and the remaining plots show the results. The effect of slat riggings can be explained by the L/D plot and fairfield streamlines shown at Fig. 2-158(b) and (c). At low AOA, a large slat gap and overhang allows accelerated flow flows through the slat slot, but when the gap/overhang is too large at high AOA, a huge gap results in lift loss. For flap riggings in Fig. 2-158(d) and (e), when flap gap or overhang increases, the amount of air rushes through the cove of the main element increases, which decreases the pressure in the trailing edge of the main element. Although the trend in these four plots looks similar: as δ increases, L/D reach their optimum value at a smaller gap/overhang, their effect on airfoil is different. According to the author, the slat gaps mainly affect the loading at the main-element first then the slat and the slat overhang changes the loading on the slat and main element equally. Flap gap influences the peak of the flap suction significantly while the flap overhang has only a slight impact on it. The last plot in Fig. 2-158 shows the effect of slat/flap deflection angle: both plots show that as δ increases, the drag increases faster than the lift.

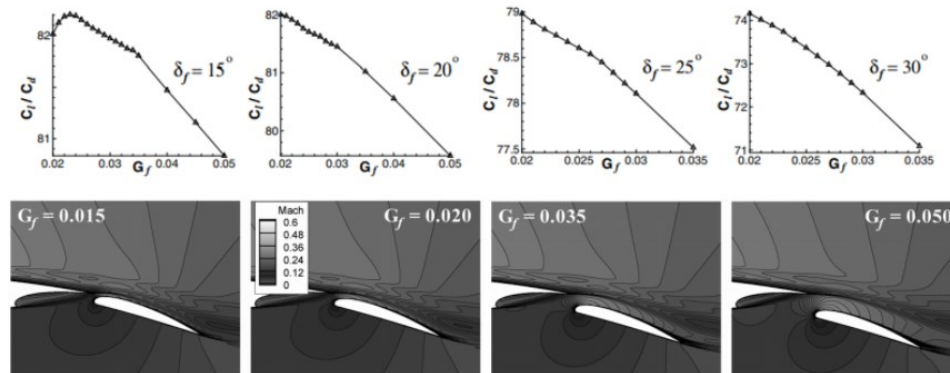




(b)



(c)



(d)

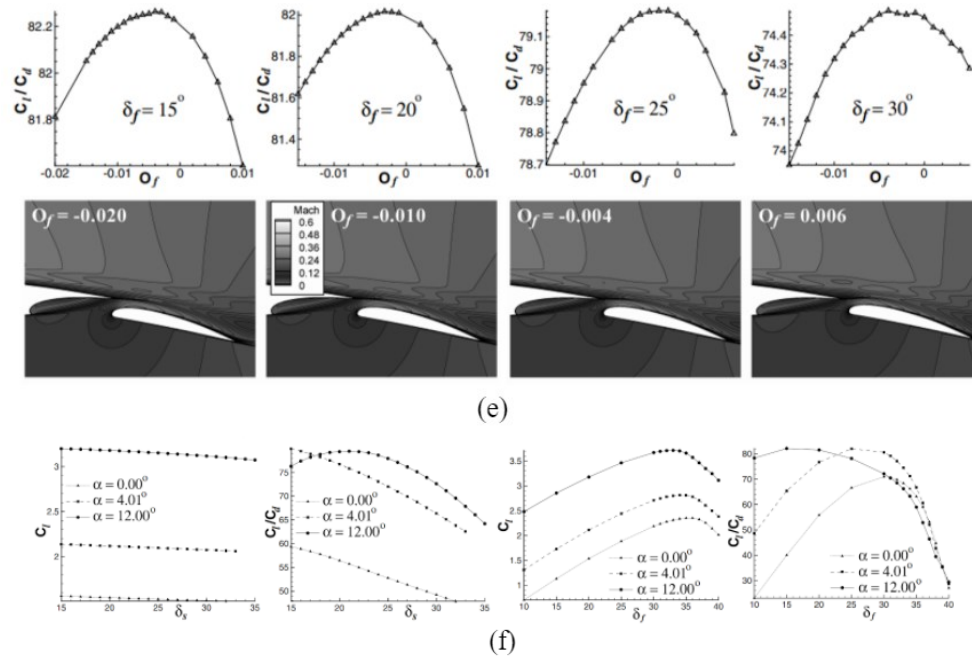
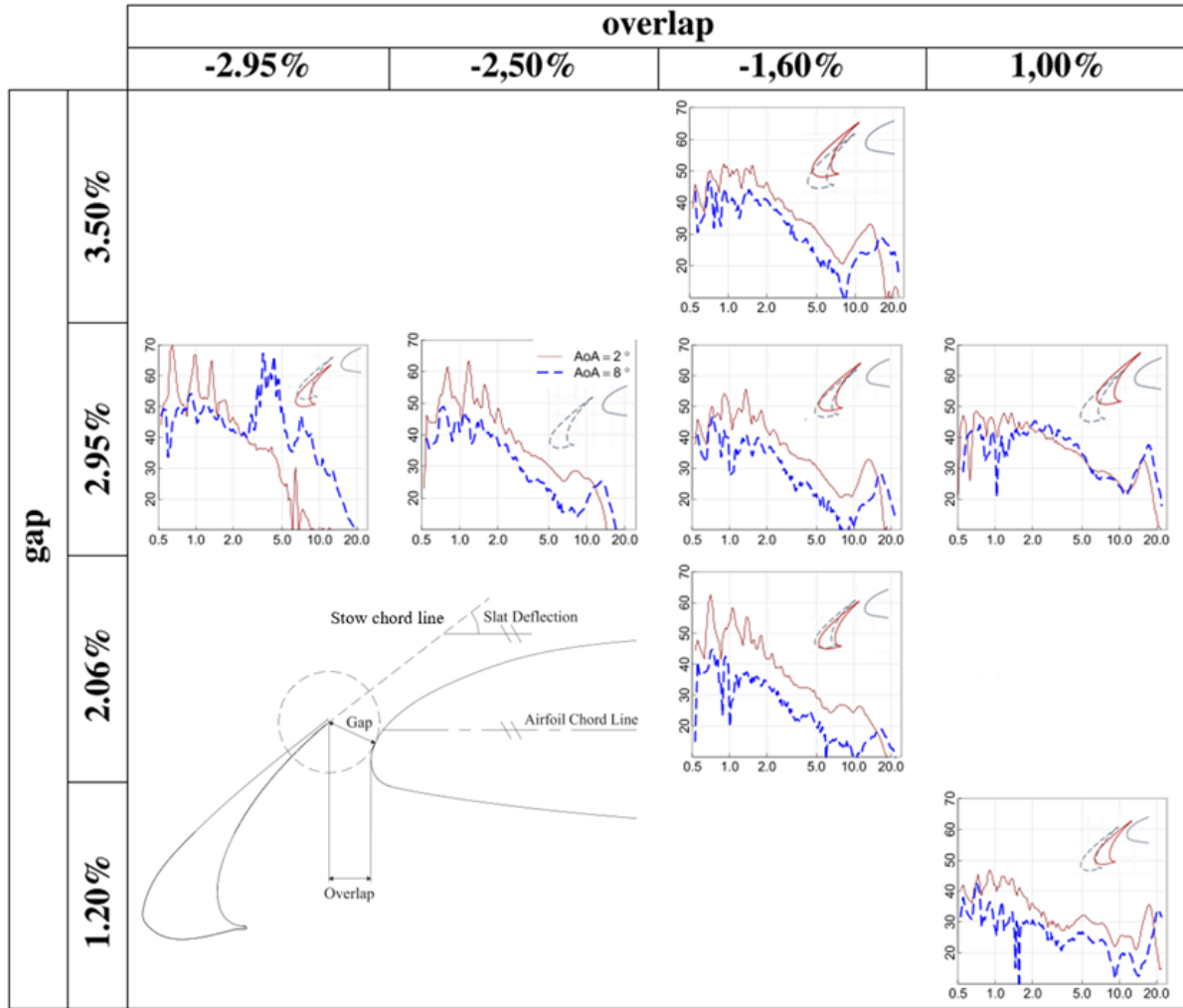


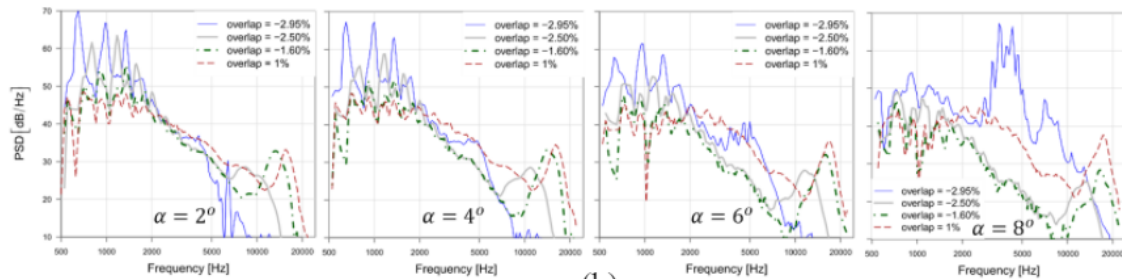
Figure 2-158 (a) Definition of the airfoil (b) Effect of slat gap on L/D and the far-field streamlines through slat slot (c) Effect of slat overhang on L/D and the Mach contours through slat slot (d) Effect of flap gap on L/D and the Mach contours through flap slot (e) Effect of flap overhang on L/D and the Mach contours through flap slot (f) L and L/D as a function of slat and flap deflection [165]

Pagani et al. [166] studied the effect of different slat configurations on noise. The definitions of the slat configuration are the overlap, the gap, and the slat deflection angle. Gap is the shortest length between the main wing and the slat. Overlap is the distance between the stowed chord line and the peak of the main wing. Slat deflection is the angle between the stowed and chord line. Fig. 2-159(a) shows the configurations and the results of the cases tested at $\delta = 30^\circ$. The case $g = 2.95\%$, $o = -1.6\%$ was then tested with the different deflection angles to study its effect, shown in Fig. 2-159(d). Fig. 2-159(b) and (c) shows the influence of overlap and gap on noise distribution. It is obvious that reducing the overlap also reduces the level of low-frequency noise, but the level of high frequency increases. Moreover, increasing the gaps slightly reduces the low-

frequency peaks, but increases the noise in high-frequency. Smaller slat deflection angle shows no impact in low-frequency noise, but generates lesser noise in high-frequency.



(a)



(b)

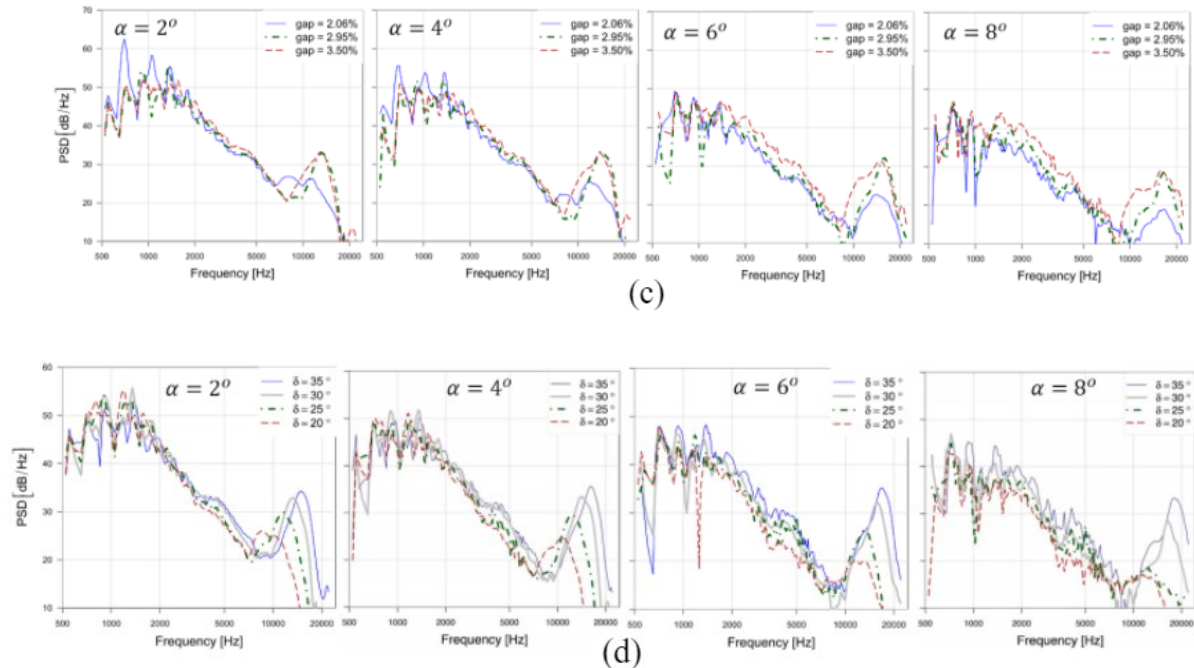


Figure 2-159 (a) Configurations and noise spectra for different slat configurations (b) Effect of slat overlap on noise (c) Effect of slat gap on noise (d) Effect of slat deflection on noise [166]

Leading edge and trailing edge positioning also related to Reynolds number effect and Mach number effect. Experimental results showed that lift curves of different slat gaps performed differently at low Reynolds number, this is probably due to flow separation occurring on the flap. Moreover, the sensitivity of the slat gap and flap overhang also depends on Reynolds number. Fig. 2-160(a) showed the effect of Reynolds on slat gap and Fig. 2-160(b) showed the effect of Mach number on lift curve [167]. Another effect that is found related to Reynolds number is vortex roll-up. As shown in Fig. 2-161, the flow in the shear layer from the slat cusp to the slat trailing edge is at laminar state during low Reynolds number. When Re increases, vortex roll-up might happen in the shear layer, providing additional noise contribution, and thus changes the optimum slat position [168].

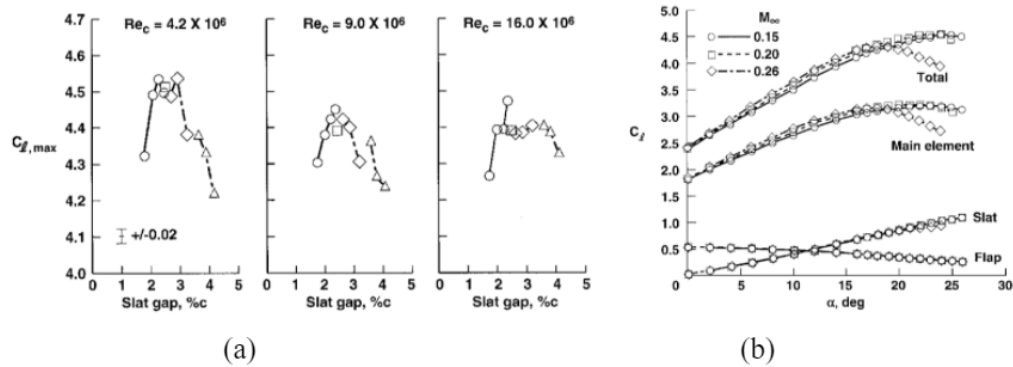


Figure 2-160 Effect of Re on slat gap (b) Effect of M on lift curves of configuration [167].

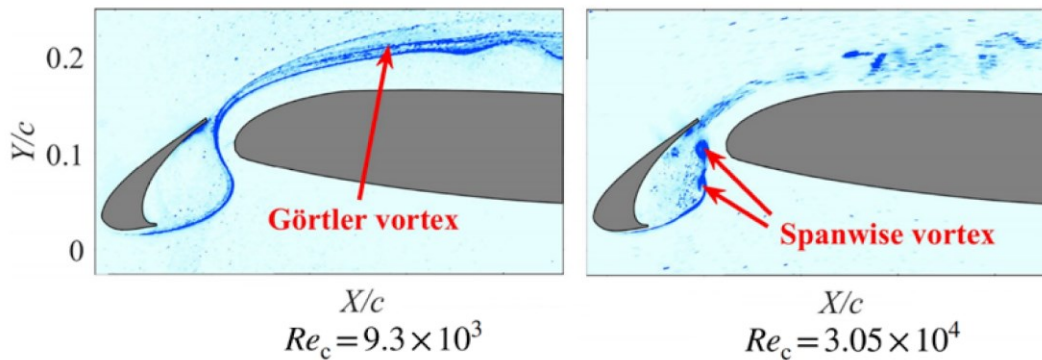


Figure 2-161 Side view measurement of slat wake [168].

Ground effect and buffets are the remaining two effects that are related to leading edge and trailing edge positioning but haven't been studied in the past. Ground effect is that the aerodynamics and flow field during take-off and landing are different from the unbounded flow during cruise. The aerodynamics and flow physics for a single-element airfoil have been studied widely, but not for multi-element airfoils. This is because previous research did not take leading edge and trailing edge positioning into consideration [169]. Buffets are the flow instabilities caused by shock/turbulence boundary layer interaction. These interactions produce oscillating aerodynamic forces that affect the performance of aircraft. One way to weaken the shock strength is installing buffet control devices like trailing edge deflectors and fixed bump. Fig. 2-162 shows a sketch of the trailing edge deflector and its effect. The positioning of the flap might influence the

location of the shock, which influences the position of the bump or deflector, but no study related to this has been done yet [170].

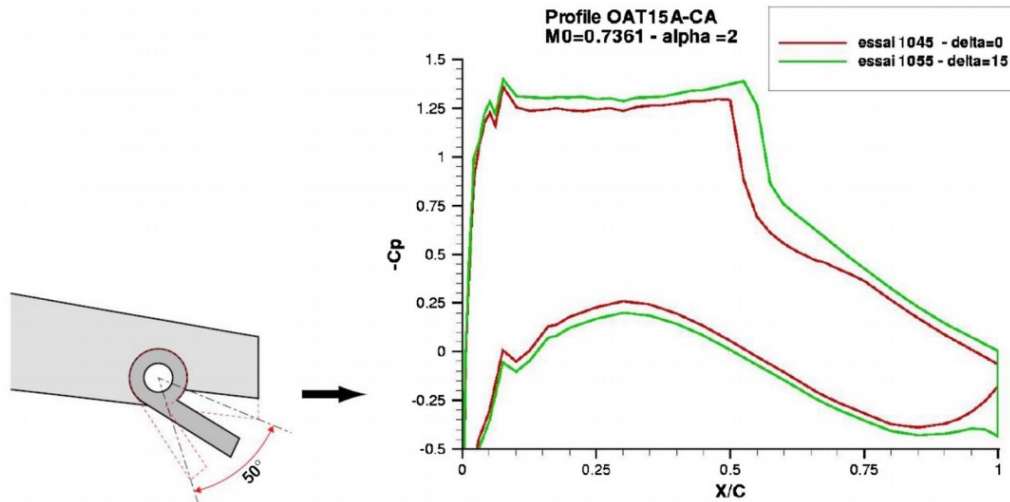


Figure 2-162 Sketch of a trailing edge deflector and its effect [170].

Previous studies focused on the overall effect of the riggings, lacking a comparison to the detail flow fields. For instance, no research studied the influence of trailing edge vortex shedding on the main wing at different slat gaps/overlap or the contribution of flap side edge noise at different flap gaps. Moreover, experimental data related to different slat/flap positionings are rare, even the NASA HL-CRM did not provide their data for deciding the leading and trailing positions. These data might be confidential for aircraft manufacturers. Hence, it is valuable to acquire more experimental data using force balances, flow visualization techniques, and microphone arrays to study the optimum positions for the UW-HL-CRM high lift system since they're not available in public.

2.16 WAKES STUDIES

Vortices produced by aircraft wings with long life-times wakes can endanger any aircraft following close behind (Fig. 2-163 (a)). To avoid such accidents, the Federal Aviation

Administration (FAA) of the USA and the International Civil Aviation Organization (ICAO) have established an instrumented flight rule to require following aircraft maintain a safety distance from a landing aircraft up ahead of them. However, these safety standards started to limit the capacity of many airports due to the strong growth of air traffic. Current separation standards are not fully adequate since they are largely empirical and lack full rationale (Fig. 2-164) [171][173]. Hence, if there is a predict system that can predict when and how long the corridor is free of vortices, air-traffic controllers might ease some of the regulations without loss of safety. In order to develop a predictive system, the knowledge of wake-vortex behavior under varying meteorological conditions is needed to solve questions like when will the crosswinds permit reduced separations? What separation is possible? How long will the conditions be appropriate for using reduced separations? What is the procedure if the winds shift or decrease in magnitude? Can one forecast when appropriate conditions will be available? [174][175] Current knowledge of vortex behavior can be classified into near field, mid/far field (in ground and atmosphere), as shown in Fig. 2-163 (b). The following paragraphs gives a brief introduction about vortex combination during near field, vortex motion and decay process in ground, and the conditions that influence the vortex at atmospheric conditions.

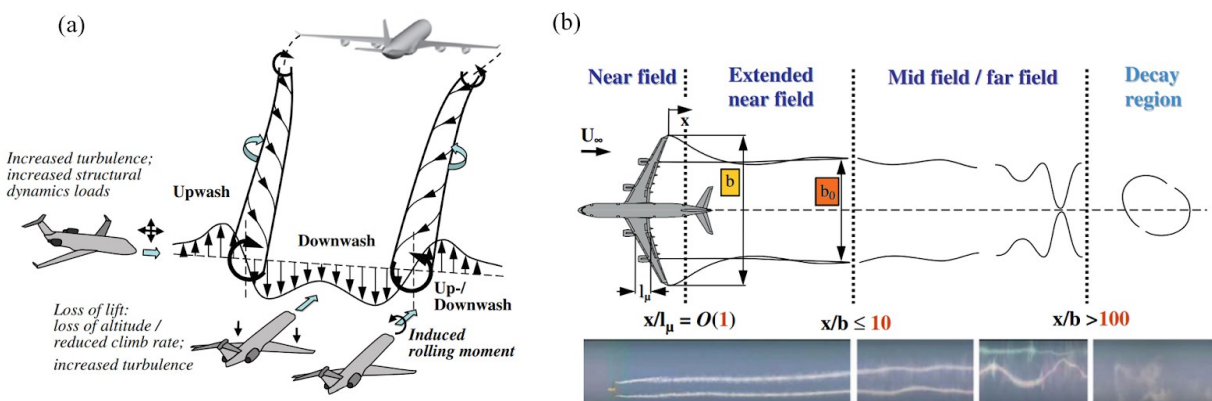


Figure 2-163 (a) Effect of wake vortices on aircrafts (b) Different stages of wake [172]

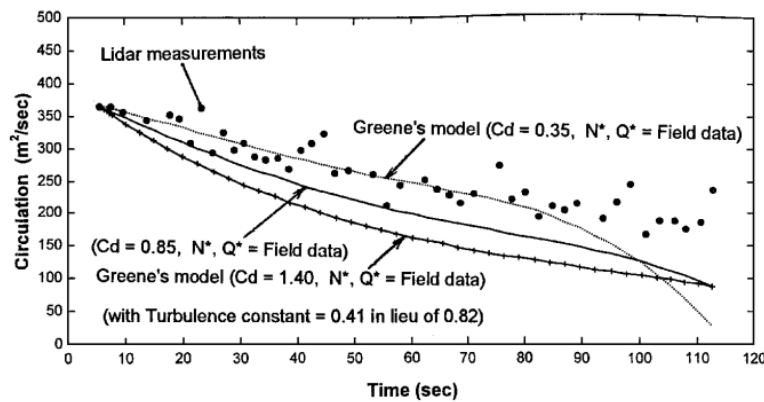


Figure 2-164 Simulated vortex circulation vs. time [173]

Aircraft wake origins from the vortices generated by different devices on the wing like wingtips, side edges of slat, nacelle, slat horns, and flap track fairings. Various types of near field vortices can be classified into six main vortices (Fig. 2-165): WTV (wingtip vortex), HTV, (Horizontal Tailplane Vortex), OFV (Outboard Flap Vortex), WFV (Wing Fuselage Vortex), ONV (Outboard Nacelle Vortex), and INV (Inboard Nacelle Vortex). The research interest in the near field/extended near field focus on the understanding of how each vortices roll-up and evolve into an elongate wake-vortex. Previous studies show that the mechanism of the roll-up process is different at low Reynolds number ($Re < 2000$) and high Reynolds number ($Re > 2000$). For low Reynolds numbers, the merging process is mainly quasi 2D. The vortices first rotate each other, the vortex sheets then deform and spread out wrapping around the vortices. The distance between both vortices decreases and finally the two vortices core merges into one as shown in Fig. 2-166(a). For high Reynolds numbers, the instability is rather 3D, although it also starts by both vortices rotating around each other. The main difference is that secondary vortices perpendicular to the primary vortex will be created during the wrapping process. These secondary vortices will accelerate the merging process transforming the initial vortex faster to a common vortex core (Fig. 2-166(b)) [172].

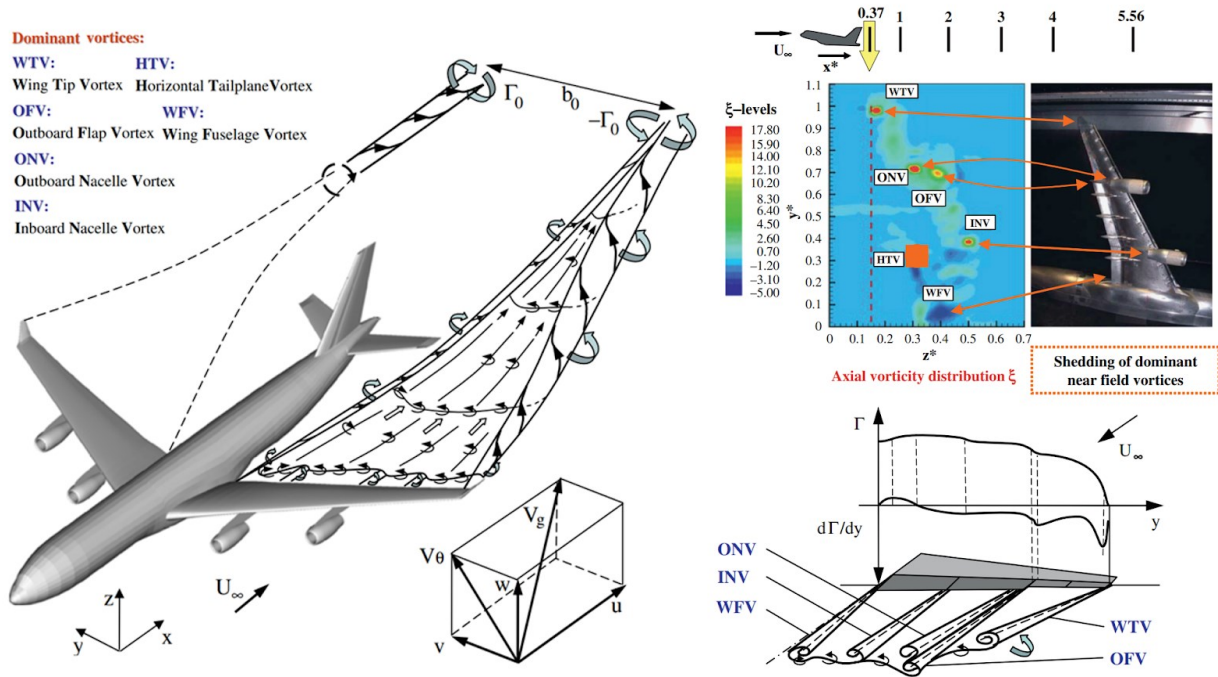


Figure 2-165 Concept of the roll-up process and sources of various near field vortices [172].

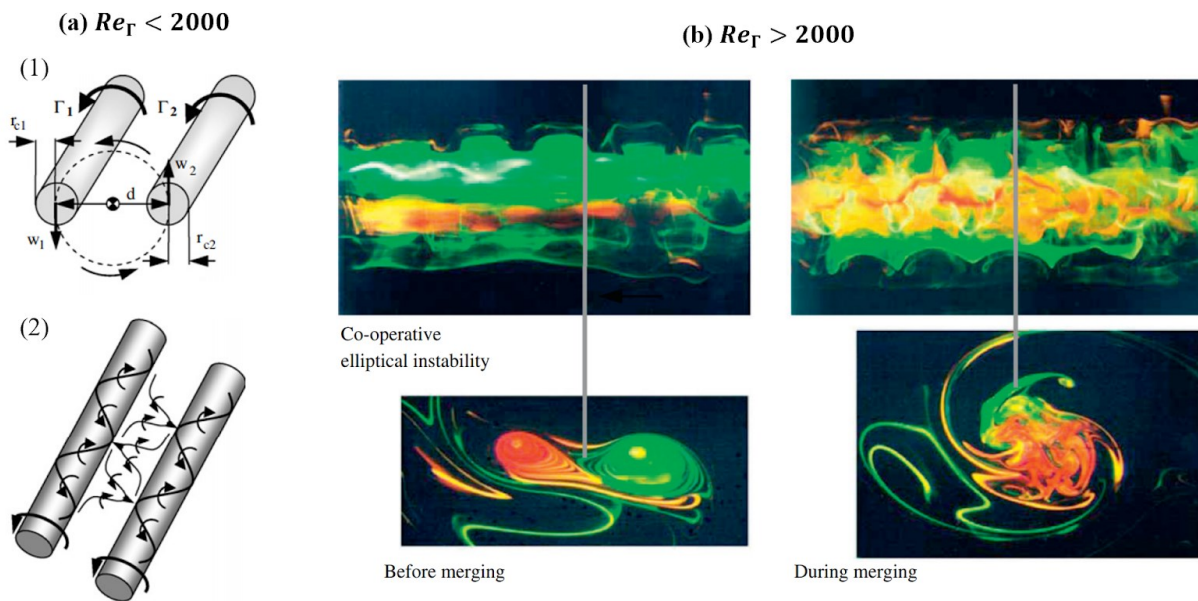


Figure 2-166 (a) Concept of the vortex merging mechanism at low Reynolds number (b) Experimental images of vortex merging at high Reynolds number [172].

The definition of wake vortex in ground conditions is usually two altitudes above ground ($z_0 = 2b_0$). 6 Jet vortices acting at the ground will induce a vortex boundary layer as it descends.

This boundary layer close to the ground will roll-up into a secondary vortex and then separates (Fig. 2-167). When encountered crosswind, elongated structures called streaks will generate from the high velocity in the spanwise direction from the crosswind. The streaks cause wind gradient regions at the upwind and downwind secondary vortex, influence the wakes, and reduce the circulation decay rate (Fig. 2-168). The team at DLR also studied the obstacle effect: wake vortices in ground with obstacles. Results in Fig. 2-169 shows that the secondary vortex of the wake changes significantly while approaching the obstacle, leading to a rapid reduction of the circulation. In Fig. 2-170, the wake-vortex evolution then provides a detailed analysis of the mechanism. First, an omega-shape vortex generated when the secondary vortex of the wake encounters an obstacle. Then the omega vortex extends and interacts with the primary vortex by self-induction. At $t^* = 1.35$, the stretched omega vortex around the primary vortex and travels along with the primary vortex. Last, both vortices transform into the same helix vortex which has the same rotations, helicity, and direction [177].

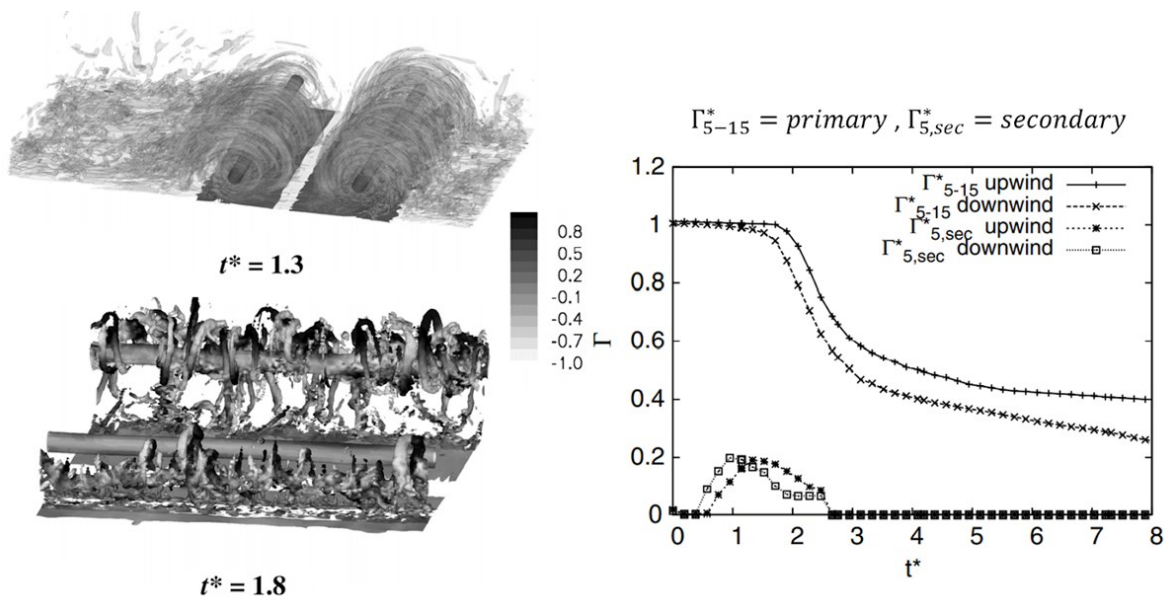


Figure 2-167 Numerical results of the secondary vortex situation and the circulations between the primary and secondary vortex [177].

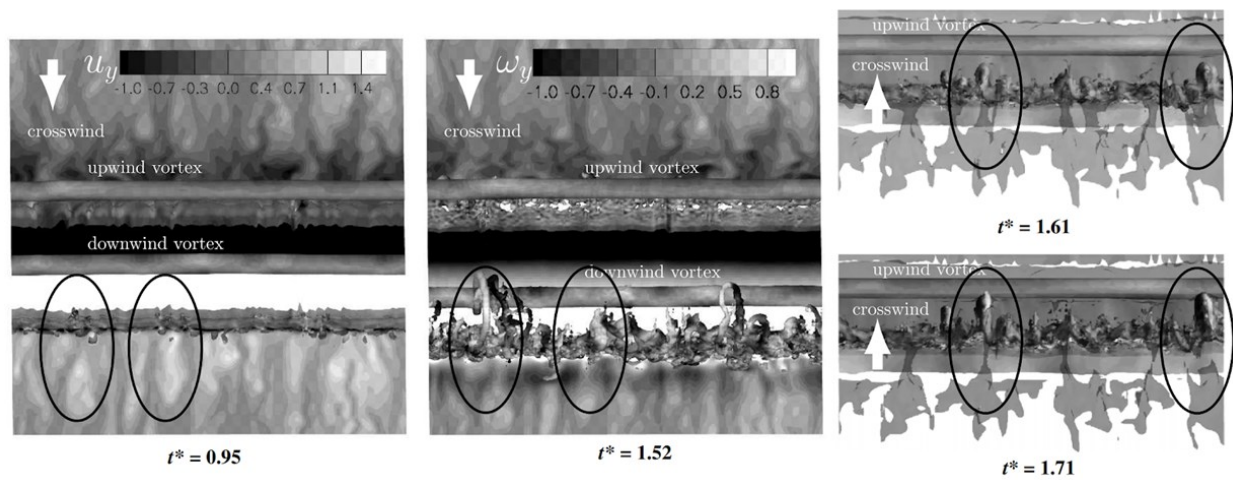


Figure 2-168 Effect of the crosswind on wake vortices in ground from CFD simulations [177].

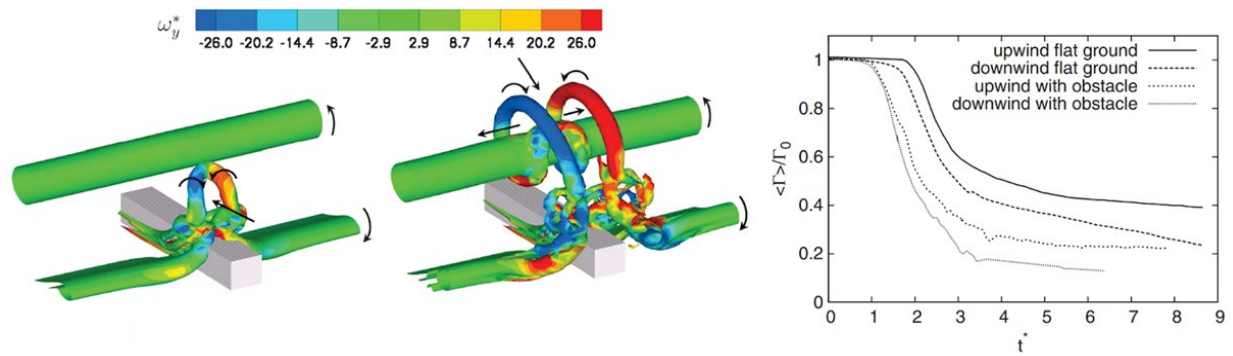


Figure 2-169 Mechanism of wake vortex in ground with obstacles and the effect of obstacle on ground wake [177].

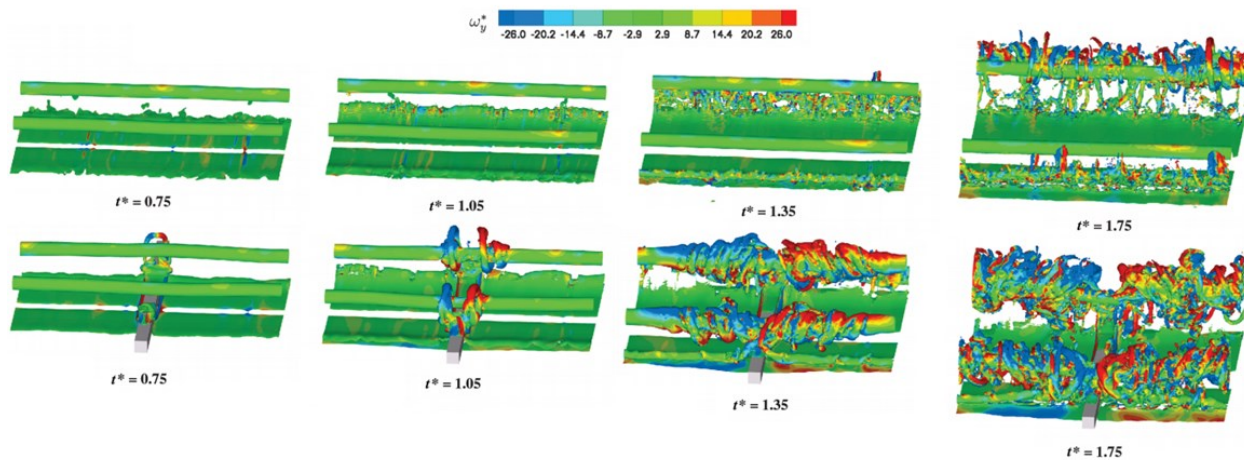
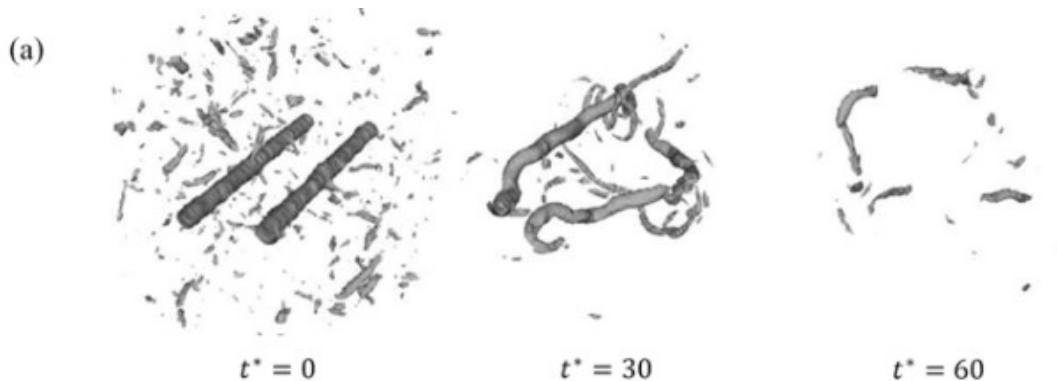


Figure 2-170 Evolution of ground wake vortices encounter crosswind and obstacles [177].

Wake vortex decay mechanisms in the atmosphere can be characterized into three conditions: in homogeneous isotropic turbulence (HIT) environment, in turbulent stably stratified atmosphere, and in weakly turbulent sheared environments. HIT is the effect of turbulence in ambient air on wake vortices. The ambient turbulence will squeeze and stretch the primary vortex and destroy it eventually (Fig. 2-171(a)). In Fig. 2-171 (b), for the HIT effect with a smaller vortex separation shows that the circulation shows a very different decay behavior. Next, stably stratified atmospheric is a boundary layer near the surface where the air temperature is colder. Numerical results show that turbulence in this case will be amplified, tilted then wrapped around the primary vortex. Shear flows in a turbulent environment will induce lateral velocities around the primary vortex and leads to a decay. As shown in Fig. 2-171 (d), vortex stretching of the primary vortex and ambient turbulence leads to a streak in the right vortex. The streak later extends through the streamline, expands to the adjacent vortex, and causes a rapid decay. The evolution of wake in different conditions is shown in Fig. 2-171 (e). For HIT, the types of the vortex (single or pairs) and the strength of different vortex separation (b_0/rc) plays an important role in circulation decay for small turbulence intensity ($I_a = 5.5\%$), but the influence of these parameters minimized for large turbulence intensity ($I_a = 524\%$). Interaction with a strong turbulent stably stratified atmosphere will lead to a delay that can reduce vortex lifetime depends on different stratifications (N^*). Last, shear environments can cause rapid decay in the wake lifetime [178].



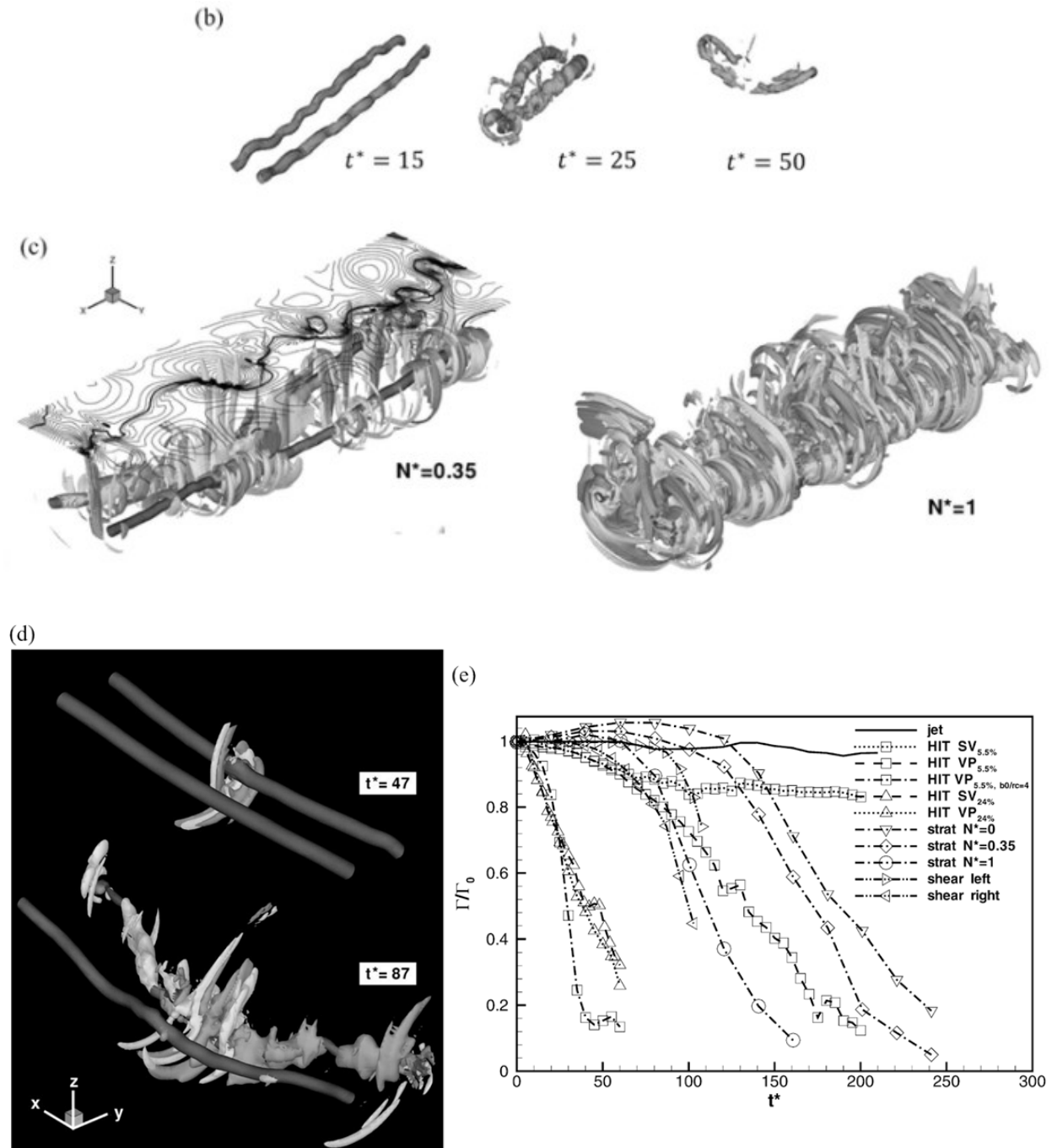


Figure 2-171 (a) Evolution of wake vortex in HIT condition (b) Evolution of wake vortex in HIT condition with a smaller vortex separation (c) Wake vortex under different stratified atmosphere (d) Wake vortex in shear environment (e) Time evolutions of wake vortices at various conditions [178].

The mechanisms of wake vortex in the nearfield, in ground and in the atmosphere have been briefly discussed. For vortex in nearfield, although only the combination of vortices is discussed in the chapter, the investigation in nearfield vortices is pretty mature. This is because that near field vortex can easily be captured in wind tunnels using flow visualization techniques. [172] provides more insights in the characteristics vortex nearfield interaction. In contrast, the understanding of mid and far field mainly relies on computational results. This is due to the difficulty of measuring wake properties in the atmosphere. Current method uses pulsed lidars to capture vortex configurations near ground, but this technique has a limit of 1500 feet at altitudes, and it's hard to track the propagation of vortices for long translational distances. Fig. 2-172 shows the concept of using pulsed lidar to collect data. Last, the industry still remains questions like, How near-field modifications translate into the far field? How relevant wing/aircraft parameters influence the decay of vortices? What is the procedure if the winds shift or decrease in magnitude? Can one forecast when appropriate conditions will be available? The research in wake requires further investigation, especially comparing wind tunnel and simulations results to flight data [171][174][175].

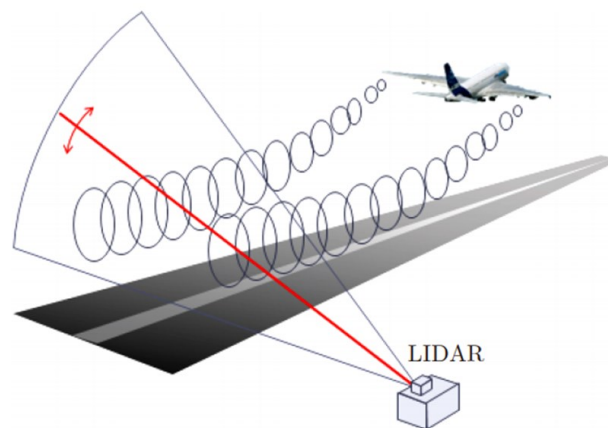


Figure 2-172 Concept of pulsed lidar [179].

2.17 TRANSITION STUDIES

Transition from laminar to turbulent flow on wing surfaces may increase the drag. Hence, maintaining laminar flow and delay transition and turbulence over wing surfaces to reduce wall friction, increase flight range and decrease operating cost is one of the main targets in aerodynamics. Yet an accurate mathematical description for transition is still missing, this leads to different numerical results [180]. Since current turbulence models don't include transition effects, the benefits of including transition models in computation are still uncertain. For instance, in Fig. 2-173(a), the results from Coder and Ortiz-Melendez2 shows that CFD models included transition have higher aerodynamic results than a fully turbulent model without transition. However, in Fig. 2-173(b), CFD results from JAXA showed models included transition does not have influence on global force [134]. Hence, one of the goals of HL-CRM is to provide research in predicting transition locations on wings, acquire high-quality experimental data, and delay transition on airfoils. Different types of transitions will be introduced in this chapter.

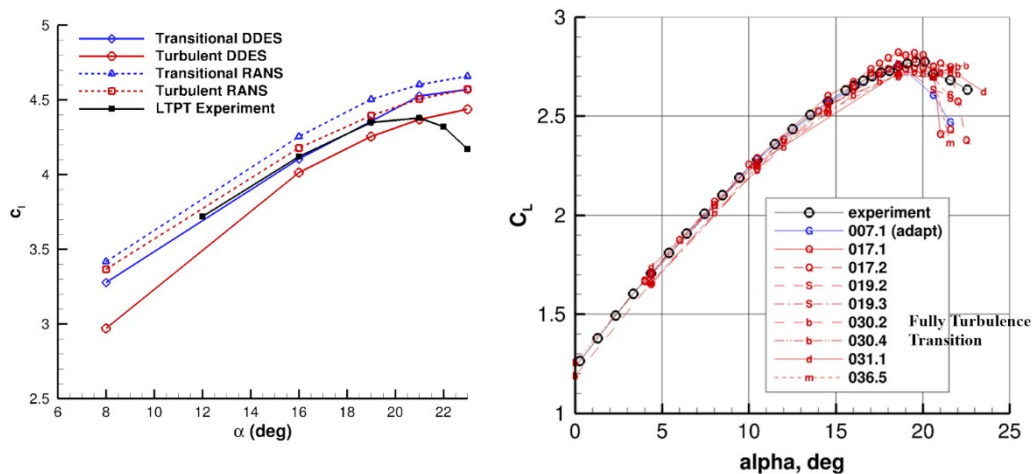


Figure 2-173 (a) CFD and experiment results from Coder and Ortiz-Melendez [181] (b) CFD and experiment results from JAXA [134].

In the European High Lift Programme EUROLIFT, a research project was carried out to acquire and provide a detailed database for developing numerical methods and for transition prediction tool developers. Because of different profiles and local conditions, different transition processes can be observed on each element, which makes it very difficult to evaluate the changes in Reynolds number. Therefore, transition research in the EUROLIFT project focused on the adverse Reynolds number effects (a sudden decrease of maximum lift with increasing Reynolds number) occurred during the extrapolation from wind tunnel to flight test results. Yet the adverse Reynolds number effect is not the only challenge in providing an accurate mathematical description for transition. Other mechanisms that trigger transition to turbulence such as Tollmien-Schlichting (TS) waves, short bubble transition, attachment line contamination, cross flow instability, and relaminization needed to be predicted accurately [182].

TS wave is a streamwise unstable wave and will generally trigger transition to turbulence. It starts with linear amplification waves in an undisturbed boundary layer. These waves then grow up to resonant amplified wave pairs due to the interaction with non-linear disturbance from the leading edge, which eventually break down to turbulence. The mechanism of how the induced waves develop into turbulence is not fully understood. More insight may be gained by obtaining more experimental data. However, TS-waves are hard to measure since their amplitudes are well below 1% of the freestream velocity [183]. Moreover, slat track waves from the leading edge also makes the measurement difficult. In addition to this, the geometry used for calculation might also be a reason for the gap between experimental results and numerical results. For instance, NASA's high lift trap wing model includes supporting slats and flap brackets, but the pressure distribution taken from CFD results on a model without supporting slats and flap brackets [184].

Attachment line contamination is due to the propagation of turbulence from the fuselage or wind tunnel floor along to the attachment line of the wing, as shown in Fig. 2-174. The flow above the wing surface will become turbulent when attachment line contamination occurs. Attachment line contamination is characterized by a parameter $\bar{R} = W_e / \sqrt{v(dU_e/dX)X_a}$, where W_e is the spanwise velocity component. The critical value for the flow being contaminated is around $\bar{R} = 250$, and the values of $\bar{R}$ on civil aircraft wing root areas range from 400 to 700 [182][186]. In order to prevent contamination, an anti-contamination device (ACD) has been studied. Fig. 2-175 showed that ACD has the ability to prevent leading edge contamination by placing ACD at the leading edge [186]. However, the aerodynamic performance of the ACD has not been tested in the previous research.

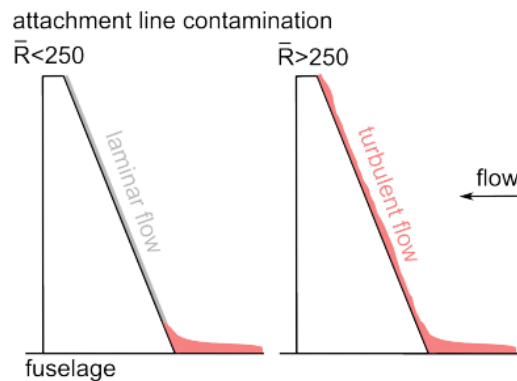
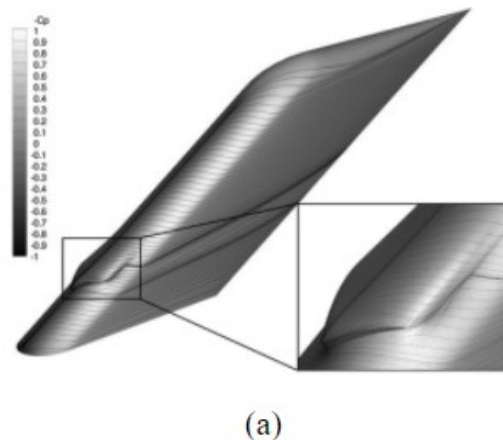


Figure 2-174 Concept of attachment line contamination [185].



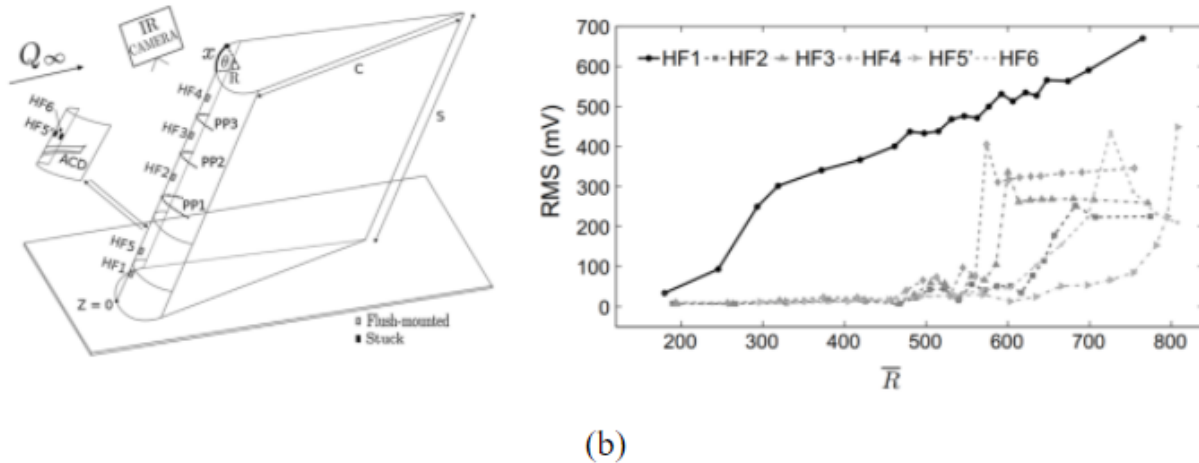


Figure 2-175 (a) Profile of ACD (b) Effect of ACD at different locations on the leading edge [186].

Laminar separation bubble (LSB) is a separation area with a moderate recirculation flow and constant pressure. The existence of LSB is due to several reasons: a strong negative pressure gradient, a turbulent flow that could change the inside separation layer, or a turbulent reattachment. A scheme of the nature of LSB on a turbine blade is shown at Fig. 2-176. LSB will lead to an increase in airfoil drag and extensive pressure fluctuations. Hence, with an understanding of LSB and its behavior can help engineers design an optimal airfoil for low Reynolds number. Studies in LSB starts from the influence and behavior at the stall situation of airfoils later on to carrying out empirical models. However, these semi-empirical models cannot foresee the behavior in all situations. Current studies focused on the characteristics of the unsteady bubble, and the influence of up-flow aggravations. Somashekar and Immanuel Selwyn Raj [187] listed some research gaps in the development of LSB. For instance: the relation of the lift with vortex circulation along with position and size of vortices was not investigate, not enough research found on the LSB characteristics, the behavior of quick bubble transition can be described by Kelvin-Helmholtz instability but the manner of long bubble transition is currently unclear, and the lift and drag performance is different depends on different Reynolds number.

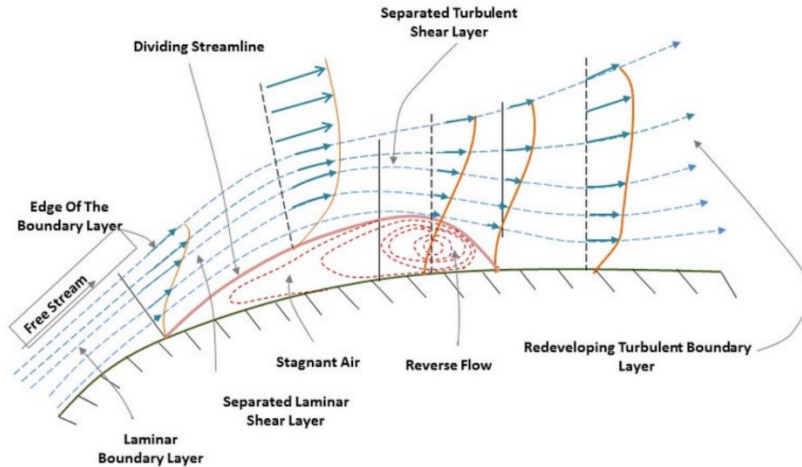


Figure 2-176 Nature of laminar separation bubble on a turbine blade [187].

Moreover, Olson et al. [188], listed four challenges in experimental characterization of LSB. The first is the uncertainty due to determination of angle of attack. The accuracy of the angle of attack is related to chord length: if the chord length accuracy is desired to be better than 1% in determining the separation location, the determination of the angle of attack must be better than ± 0.1 . This level of accuracy is hard to accomplish for asymmetric airfoils. Another discrepancy is the uncertainty due to near-wall measurement resolution. The separation point of LSB is chosen to be between the location where the velocity first reverses and the one immediately upstream of it. Yet the flow reversal region near the wall is extremely thin in the wall-normal direction, which needs sufficient spatial resolution to identify reverse flow. In addition to this, the reattachment location is found to have a huge difference within small changes in Reynolds number, shown in Fig. 2-177. The last one is the sensitivity of the LSB to freestream turbulence level. It is difficult to compare results obtained in different experiments because of the variation of freestream turbulence intensity. LSB can be reduced by applying an external body force. In Fig. 2-178, a simulation done by Aholt and Finaish [189] showed that by placing a plasma actuator in the optimum location, the external body force from the actuator can eliminate LSB.

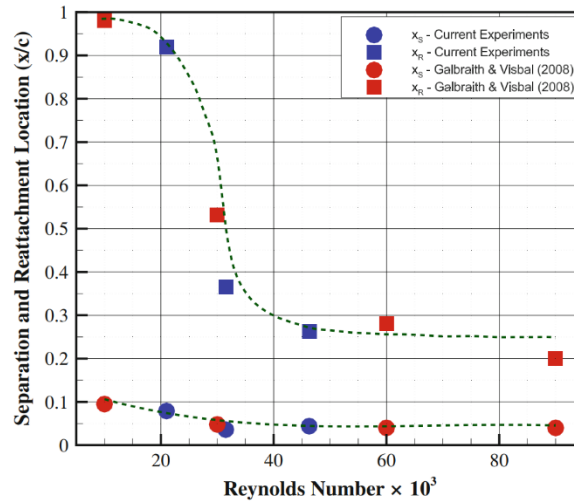


Figure 2-177 Dependence of the separation and reattachment location on different Reynolds number of a micro air vehicle [188].

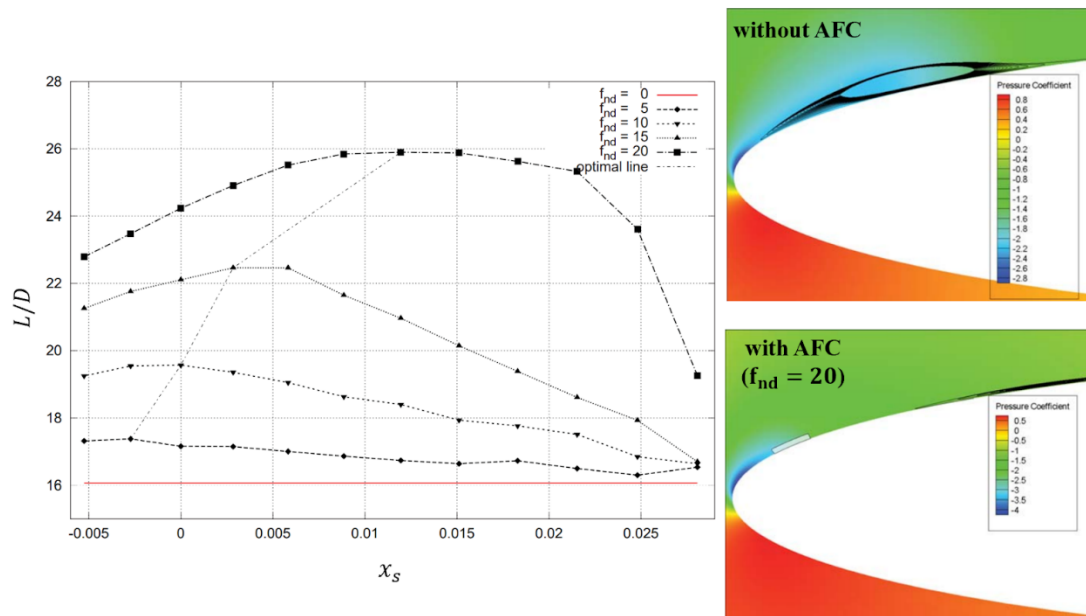


Figure 2-178 Effect of reducing LSB by applying plasma actuators [189].

Crossflow triggered transition is mainly driven by surface roughness. Natural-surface roughness generated stationary crossflow features, these small stationary disturbances then later transfer momentum and produce a large integrated effect in the streamwise flow. With significant progress in measurement techniques and numerical tools in the past decades, importance factors

such as environmental conditions on the appearance of stationary and traveling waves, different crossflow modes (primary crossflow instabilities and secondary crossflow instability), sensitivity of the stationary disturbance to leading-edge, and nonlinear effects and modal interaction have been identified. Moreover, the mechanisms of the primary crossflow instability have been fully understood, like the details of the nonlinear saturation of the dominant stationary mode and the growth of harmonics [190].

Secondary crossflow instability is a mechanism acting on the boundary layer, modified by the primary stationary crossflow. In the past, the structures, behaviors, locations and growth rates of the secondary instability have been studied. However, some of these investigations cannot be verified experimentally since the full spatio-temporal development of the instability cannot be measured directly by current measurement techniques such as hot-wire measurements or PIV [191]. By understanding the behaviors of crossflow instability, transition can be delayed by installing laminar flow control techniques on swept wings. This can be accomplished by changing the surface roughness such as distributed roughness, holes, or glow discharge applied at the leading edge with. Fig. 2-179 showed an example of a laminar flow control technique: discrete roughness elements. In Fig. 2-179(b), separation can be delayed by spray coating discrete roughness elements (a) on wing surfaces [192]. Note that the effect of changing surface roughness to delay transition may differ with different Reynolds numbers. Fig. 2-180 shows the effect of roughness diameter on the transition location with different Reynolds numbers [192].

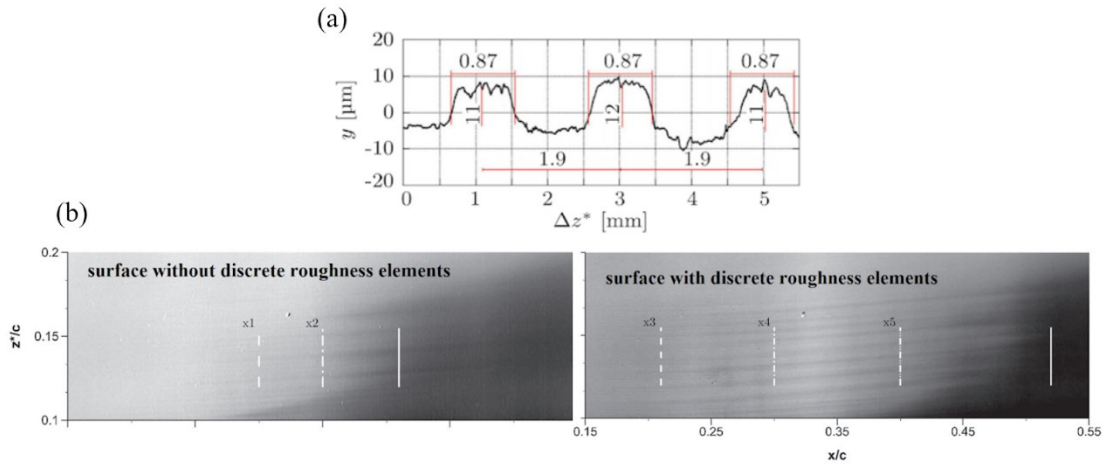


Figure 2-179 (a) Profile of discrete roughness elements (b) Effect of discrete roughness elements [191].

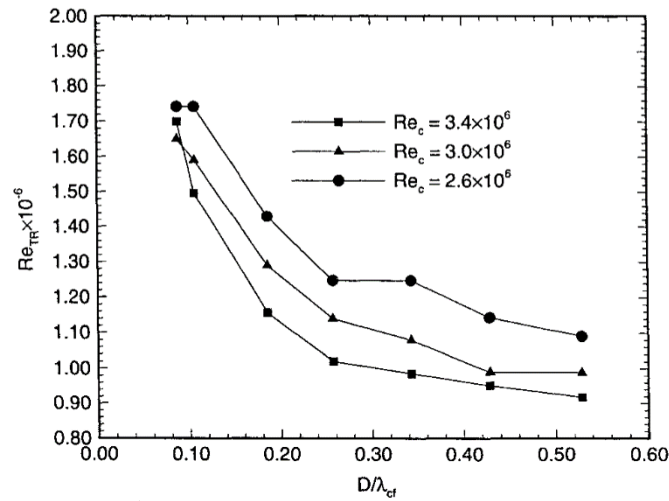


Figure 2-180 Effect of roughness diameter on transition location (Re_{TR} represents the transition Reynolds number, transition location x_{tr}/c can be obtained by scaling Re_{TR} by chord Reynolds number) [193].

Relaminarization happens when the attachment line moves to the pressure surface at high incidence, and the streamwise boundary layer experiences strong acceleration when passing over the wing leading edge. This may lead to some recovery in the lift. Research of relaminarization starts from a flat plate, and an acceleration parameter $K = (\nu/U_e^2)(dU_e/dx)$ has been defined to

indicate the onset of relaminarization. These criteria and predictions are based on two-dimensional flat plate experiments [194]. Viswanath, et al. [195] found that different levels of relaminarization occurs on a swept wing with different K variants. These results mean that present empirical relationships must be updated by taking the potential role of surface curvature into account. Moreover, more fundamental parameters may be required to identify relaminarization, and more data on boundary-layer thickness and Reynold stresses is needed to investigate relaminarization.

The five main types of transition are introduced in the chapter. Among previous high lift projects, only the EUROLIFT project studied the transition locations of different mechanisms by hot film and infrared images. Due to the importance of transition, the author suggested that the UW-HL-CRM should provide further research in the mechanisms of transition and the effect of transition reduction devices using various techniques like fluid visualization.

Chapter 3. CONCLUSIONS AND FUTURE STUDIES

Present investigation provides an overview of the current achievements and the latest research interest of the low-speed aerodynamics of commercial aircrafts. A comparison between the research topics in the previous high lift projects and the these important issues from UW & Boeing are listed in Appendix A. Previous high lift project EUROLIFT and JAXA JSM focused on using flow visualization to study the boundary layer at takeoff and landing configurations and using passive vortex control device nacelle chine to eliminate nacelle-wake separation. The latest NASA HL-CRM focused on applying AFC devices on flaps, but the boundary with and without nacelle chine are also studied. Among all of the important issues, only a few of them have been mainly studied by previous high-lift projects. Hence, there is definitely a need for more experimental and/or CFD research to better understand the effect of various parameters like winglets, landing gears, etc. on the behavior of such vehicles. This investigation will definitely help optimizing the aerodynamic behavior of current aircraft. The review from each issue can be used to decide the primary functions and measurement capabilities of the UW HL-CRM. This final chapter summarizes the research interest and the priority of each topic and the next step of designing the HL-CRM.

3.1 CONCLUSIONS

All issues have been characterized into three groups by the author depending on their priority, shown in Table 7.

Group A	Group B	Group C
2.01 Cruise, Takeoff, and Landing 2.04 Landing Gear Noise 2.05 Horizontal and Vertical Tails 2.06 Flow Visualization 2.07 Boundary Layer Study 2.11 AFC 2.12 CFD 2.16 Airframe Noise 2.17 LE/TE positioning 2.19 Transition Studies	2.03 Nacelle/Pylon 2.08 Wingtips 2.10 Ice Shape 2.14 Hinge Moments 2.18 Wake	2.02 Brackets 2.09 Multi-Element Flaps

Table 7 Priorities of different issue of the HL-CRM at UW

Group A listed the issues that need a large amount of investigation.

- Cruise, Takeoff, and Landing are the three flight stages that must be studied for each topic. Therefore, deciding the optimum positioning for slats and flaps during landing and takeoff is required. Since, the data in this area is rare and even the NASA HL-CRM did not provide the data for deciding their leading and trailing positions. The investigation in how different placements of slats and flaps influence is important.
- Current CFD workflows still contain uncertainties and the actual flow properties cannot be predicted accurately. The main reason for inaccurate prediction is mainly due to the lack of a theoretically based turbulence simulation capability. In order to develop a generalized transition prediction capability to fill the missing models, more specific transition/separation/turbulence testing data from a high-fidelity geometry is needed.
- One of the main targets for designing future aircrafts is low noise. For airframe acoustics, the sources that generate noise from high lift devices and landing gears have been confirmed from previous studies. However, airframe noise reduction techniques

are either ineffective or still in research. Therefore, the author believes that the UW HL-CRM should have the ability to collect acoustic data and test different noise reduction methods.

- AFCs is one of the latest topics in aerodynamics. NASA aims to increase the efficiency of the aircraft by replacing heavy double-slotted flaps by simple hinged flaps with AFC devices, and to reduce the size of the vertical tail also with the installation of AFCs. Although the mechanism is understood, this idea is still in the early stages, more experiments need to be done in order to transfer laboratory work into real world applications.

Group B listed the issues which research interests are not as broad as those listed in Group A. The mechanisms of nacelle-wake separations and wingtip vortices have been understood roughly. The main research interests in these issues are applying AFCs on nacelle/pylon, and the realization of using folded winglets to increase aircraft performance. The reason for listing ice shape, hinge moments, and wake studies in group B is ambiguous. They're all important issues that should be the first priorities for consideration. However, the measurement capabilities in the KWT only allows our model to study specific topics. For instance, KWT can't investigate ice accretion since it requires an icing wind tunnel to simulate icing environments, but the aerodynamics caused by different ice shapes can be studied by installing an artificial ice model from 3D printing or wax on the wing. For the remaining, KWT has the ability to measure nearfield wake vortices, but don't have the space for vortices to evolve into mid/far field wakes and KWT doesn't have the ability to measure hinge moments. It is highly desirable to add such a capability to the wind tunnel.

In group C, previous studies show that the effect of supporting brackets on aerodynamics is low, and their effect on aeroacoustics is due to the small holes and gaps on the brackets, which is the same as landing gears. Therefore, focusing on the small decorations on landing gears might be a better choice. For multi-element flaps, the reason why aircraft manufacturers are giving up multi-element flaps has been explained in the previous chapter. However, our understanding of the bracket effects as well as multi-element flaps might be broadened if more studies related to these topics have been done.

3.2 FUTURE STUDIES

The next step is to provide a detailed investigation for the availability, the unsteadiness, and the capacities of different measurement methods required for each experiment. For instance, if we want to measure surface pressure using PSP, what chemical composition in the paint should be used, what is the error between different paints. After that, the modularity, the mechanism, unsteady measurement capabilities, the fidelity, the fabrication, and the scale of each component needed to be incorporated in the design of this model. Building a HL-CRM that could use 10+ years is not easy, there's still a long road to achieve.

BIBLIOGRAPHY

- [1] Lacy, D. S., Sclafani, A. J., “Development of the High Lift Common Research Model (HL-CRM): A Representative High Lift Configuration for Transonic Transports,” AIAA SciTech Forum, AIAA 2016-0308, 2016 [N1]
- [2] Hansen H., “Overview About the European High Lift Research Programme EUROLIFT,” AIAA Aerospace Sciences Meeting and Exhibit, AIAA 2004-0767, 2004
- [3] Rudnik, R., “CFD Assessment For High Lift Flows in the European Project EUROLIFT,” Applied Aerodynamics Conference, AIAA 2003-3794, 2003
- [4] Séraudie, A., Perraud, J., Moens, F., “Transition Measurement and Analysis on a Swept Wing in High Lift Configuration,” Aerospace Science and Technology, Vol. 7, Issue 8, 2003
- [5] Rudnik, R., Geyr, H. F. v., “The European High Lift Project EUROLIFT II - Objectives, Approach, and Structure,” AIAA Applied Aerodynamics Conference, AIAA 2007-4296, 2007
- [6] Wild, J., Brezillon, J., Amoignon, O., Quest, J., Moens, F., Quagliarella, D., “Advanced High-Lift Design by Numerical Methods and Wind Tunnel Verification within the European Project EUROLIFT II,” AIAA Applied Aerodynamics Conference, AIAA 2007-4300, 2007
- [7] van der Burg, J. W., von Geyr, H. F., Heinrich, R., Eliasson, P., Delille, T., Krier, J., “Geometrical Model Installation and Deformation Effects in the European Project EUROLIFT II,” AIAA Applied Aerodynamics Conference, AIAA 2007-4297, 2007
- [8] Quix, H., Schulz, M., Quest, J., Rudnik, R., Schröder, A., “Low Speed High Lift Validation Tests within the European Project EUROLIFT II,” AIAA Applied Aerodynamics Conference, AIAA 2007-4298, 2007
- [9] Rudnik, R., Germain, E., “Re-No. Scaling Effects on the EUROLIFT High Lift Configurations,” AIAA Aerospace Sciences Meeting and Exhibit, AIAA 2007-0752, 2007
- [10] Rudnik, R., “Stall Behaviour of the EUROLIFT High Lift Configurations,” AIAA Aerospace Sciences Meeting and Exhibit, AIAA 2008-0836, 2008
- [11] Ito, T., Yokokawa, Y., Ura, H., Kato, H., Mitsuo, K., Yamamoto, K., “High-Lift Device Testing in JAXA 6.5m x 5.5m Low-speed Wind Tunnel,” AIAA paper 2006-3643, 2006

- [12] Yokokawa, Y., Kanazaki, M., Murayama, M., Kato, H., Ito, T., Yamamoto, K., "Investigation of the Flow Over Nacelle/Pylon and Wing Controlled with a Vortex Generator in High Lift Configuration," Proceedings of 26th ICAS, 2008
- [13] Kato, H., Watanabe, S., Hashimoto, T., Fuji, K., Ito, T., Yamamoto, K., "PIV measurement of a High-Lift-Device Model in JAXA 6.5m $\times$ 5.5m Low-speed Wind Tunnel," AIAA Aerospace Sciences Meeting and Exhibit, AIAA 2007-1064, 2007
- [14] Mitsuo, K., Kurita, M., Kuchi-Ishi, S., Fujii, K., Ito, T., Watanabe, S., Yamamoto, K., "PSP Measurement of a High-Lift-Device Model in JAXA 6.5m $\times$ 5.5m Low-Speed Wind Tunnel," AIAA Aerospace Sciences Meeting and Exhibit, AIAA 2007-1065, 2007
- [15] Ura H., Yokokawa, Y., Ito, T., "Phased Array Measurement of High Lift Devices in Low Speed Wind Tunnel," AIAA/CEAS Aeroacoustics Conference, AIAA 2006-2565, 2006
- [16] Yokokawa, Y., Murayama, M., Kanazaki, M., Murota, K., Ito, T., Yamamoto, K., "Investigation and Improvement of High-lift Aerodynamic Performances in Lowspeed Wind Tunnel Testing," AIAA Aerospace Sciences Meeting and Exhibit, AIAA 2008-0350, 2008
- [17] [J7] Kanazaki, M., Yokokawa, Y., Murayama, M., Ito, T., Jeong, S., Yamamoto, K., "Efficient Design Exploration of Nacelle Chine Installation in Wind Tunnel Testing," AIAA Aerospace Sciences Meeting and Exhibit, AIAA 2008-0155, 2008
- [18] Kato, H., Watanabe, S., Murayama, M., "PIV Investigation of Nacelle Chine Effects on High-Lift System Performance," AIAA Aerospace Sciences Meeting and Exhibit, AIAA 2008-0240, 2008
- [19] Yokokawa, Y., Murayama, M., Uchida, H., Tanaka, K., Ito, T., Yamamoto, K., "Aerodynamic Influence of a Half-Span Model Installation for High-Lift Configuration Experiment," AIAA paper 2010-0684, 2010
- [20] Ito, Y., Murayama, M., Yokokawa, Y., Yamamoto, K., Tanaka, K., Hirai, T., "Wind Tunnel Interference Effects on Japan Aerospace Exploration Agency's Standard Model," AIAA SciTech Forum, AIAA 2019-2178, 2019
- [21] Yokokawa, Y., Murayama, M., Ito, T., Yamamoto, K., "Experiment and CFD of a High-lift Configuration Civil Transport Aircraft Model," AIAA paper 2006-3452, 2006
- [22] Murayama, M., Yokokawa, Y., Tanaka, K., Yamamoto, K., Ito, Y., "Numerical Simulation of Half-span Aircraft Model with High-Lift Devices in Wind Tunnel," AIAA Aerospace Sciences Meeting and Exhibit, AIAA 2008-0333, 2008
- [23] Fell, J. S., Webb, S. R., Laws, C. T., Snyder, M. L., Rhew, R. D., "Development of the 10% NASA High-Lift Common Research Model (CRM-HL)," AIAA SciTech Forum, AIAA 2021-0385, 2021

- [24] Taylor, N. J., Gammon, M., “A Wind Tunnel Model of the NASA High Lift Common Research Model: A Geometry-Handling Perspective,” AIAA AVIATION Forum, AIAA 2019-3460, 2019
- [25] Lacy, D., Clark, A. M., “Definition of Initial Landing and Takeoff Reference Configurations for the High Lift Common Research Model (CRM-HL),” AIAA AVIATION Forum, AIAA 2020-2771, 2020
- [26] Lin, J. C., Melton, L. P., Viken, S. A., Andino, M. Y., Koklu, M., Hannon, J. A., Vatsa, V. N., “High Lift Common Research Model for Wind Tunnel Testing: An Active Flow Control Perspective,” AIAA SciTech Forum, AIAA 2017-0319, 2017
- [27] Melton, L. P., Lin, J. C., Hannon, J. A., Koklu, M., Andino, M. Y., Paschal, K. B., “Sweeping Jet Flow Control on the Simplified High-Lift Version of the Common Research Model,” AIAA AVIATION Forum, AIAA 2019-3726, 2019
- [28] Lin, J. C., Melton, L. P., Hannon, J. A., Andino, M. Y., Koklu, M., Paschal, K. B., Vatsa, V. N., “Testing of High-Lift Common Research Model with Integrated Active Flow Control,” Journal of Aircraft, 2020
- [29] Paschal, K. B., Kegerise, M. A., Neuhart, D. H., Koklu, M., Lin, J. C., “Unsteady Pressure Measurements on a Simplified High-Lift Configuration of the Common Research Model using Active Flow Control,” AIAA SciTech Forum, AIAA 2021-1647, 2021
- [30] Lin, J. C., Melton, L. P., Hannon, J. A., Andino, M. Y., Koklu, M., Paschal, K. B., Vatsa, V. N., “Wind Tunnel Testing of Active Flow Control on High-Lift Common Research Model,” AIAA AVIATION Forum, AIAA 2019-3723, 2019
- [31] Koklu, M., Lin J. C., Hannon, J. A., Melton, L. P., Andino, M. Y., Paschal, K. B., Vatsa, V. N., “Mitigation of Nacelle/Pylon Wake on the High-Lift Common Research Model Using a Nacelle Chine,” AIAA SciTech Forum, AIAA 2020-0786, 2020
- [32] Koklu, M., Lin, J. C., Hannon, J. A., Melton, L. P., Andino, M. Y., Paschal, K. B., Vatsa, V. N., “Surface Flow Visualization of the High-Lift Common Research Model,” AIAA AVIATION Forum, AIAA 2019-3727, 2019
- [33] Rivers, M., Hunter, C., Vatsa, V., “Computational Fluid Dynamic Analyses for the High-Lift Common Research Model Using the USM3D and FUN3D Flow Solvers,” AIAA SciTech, AIAA 2017-032-, 2017
- [34] Clark, A. M., Slotnick, J. P., Taylor, N., Rumsey, C. L., “Requirements and Challenges for CFD Validation within the High-Lift Common Research Model Ecosystem,” AIAA AVIATION Forum, AIAA 2020-2772, 2020
- [35] Kallas, S., Geoghegan Quinn, M., “Flightpath 2050 Europe’s Vision for Aviation - Report of the High Level Group on Aviation Research,” Technical Report, European Commission, 2011

- [36] Cameron, M. A., Jacobson, M. Z., Naiman, A. D., Lele, S. K., “Effects of Plume-Scale Versus Grid-Scale Treatment of Aircraft Exhaust Photochemistry,” *Geophysical Research Letters*, Vol. 40, Issue 21, 2013
- [37] Lacy, D., Clark, A. M., “Definition of Initial Landing and Takeoff Reference Configurations for the High Lift Common Research Model (CRM-HL),” *AIAA AVIATION Forum*, AIAA 2020-2771, 2020
- [38] Lin, J. C., Melton, L. P., Viken, S. A., Andino, M. Y., Koklu, M., Hannon, J. A., Vatsa, V. N., “High Lift Common Research Model for Wind Tunnel Testing: An Active Flow Control Perspective,” *AIAA SciTech Forum*, AIAA 2017-0319, 2017
- [39] Radespie, R., Heinze, W., “SFB 880: Fundamentals of High Lift for Future Commercial Aircraft,” *CEAS Aeronautical Journal*, Vol. 5, Issue 3, 2014
- [40] Shmilovich, A., Yadlin, Y., Dickey, E. D., Hartwich, P. M., Khodadoust, A., “Development of an Active Flow Control Technique for an Airplane High-Lift Configuration,” *AIAA SciTech Forum*, AIAA 2017-0322, 2017
- [41] Souza, P. R. C., Proença, A. R., de Almeida, O., Self, R. H., “Aerodynamics and Aeroacoustics Investigation of a Low Speed Subsonic Jet,” *International Journal of Acoustics and Vibration*, Vol. 22, No. 1, pp. 121-130, 2017
- [42] Guo, Y., Thomas, R. H., Clark, I. A., June, J. C., “Far Term Noise Reduction Roadmap for the Mid-Fuselage Nacelle Subsonic Transport,” *AIAA Aviation Forum*, AIAA 2018-3126, 2018
- [43] Murayama, M., Yokokawa, Y., Yamamoto, K., Ueda, Y., “CFD Validation Study for a High-Lift Configuration of a Civil Aircraft Model,” *AIAA Applied Aerodynamics Conference*, AIAA 2007-3924, 2007
- [44] Fares, E., Duda, B., Khorrami, M. R., “Airframe Noise Prediction of a Full Aircraft in Model and Full Scale Using a Lattice Boltzmann Approach,” *AIAA/CEAS Aeroacoustics Conference*, AIAA 2016-2707, 2016
- [45] Wang, X., Hu, Z., Zhang, X., “Aeroacoustic Effects of High-Lift Wing Slat Track and Cut-Out System,” *International Journal of Aeroacoustics*, Vol. 12, No. 3, pp.283-308, 2013
- [46] Fricke, S., Ciobaca, V., Wild, J., Norman, D., “Numerical Studies of Active Flow Control Applied at the Engine-Wing Junction,” *Advances in Simulation of Wing and Nacelle Stall, Notes on Numerical Fluid Mechanics and Multidisciplinary Design*, Vol. 131, pp. 397–411, 2016
- [47] Schloesser, P., Soudakov, V., Bauer, M., Wild, J., “Active Separation Control at the Pylon-Wing Junction of a Real-Scale Model,” *AIAA Journal*, Vol. 57, No. 1, 2019
- [48] Smith, S. C., Nemec, M., Krist, S. E., “Integrated Nacelle-Wing Shape Optimization for an Ultra-High Bypass Fanjet Installation on a Single-Aisle Transport Configuration,” *AIAA*

Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, AIAA 2013-0543, 2013

- [49] Übelacker, S., Hain, R., Kähler, C. J., “Flow Investigations in a Stalling Nacelle Inlet Under Disturbed Inflow,” *Advances in Simulation of Wing and Nacelle Stall*. FOR 1066 2014. Notes on Numerical Fluid Mechanics and Multidisciplinary Design, vol 131. Springer, Cham., 2016
- [50] Xiao, Z., Liu, J., Luo, K., Huang, J., Fu, S., “Investigation of Flows Around a Rudimentary Landing Gear with Advanced Detached-Eddy-Simulation Approaches,” *AIAA Journal*, Vol. 51, Issue 1, 2013
- [51] Merino-Martinez, R., Neri, E., Snellen, M., Kennedy, J., Simons, D. G., Bennett, G. J., “Analysis of Nose Landing Gear Noise Comparing Numerical Computations, Prediction Models and Flyover and Wind-Tunnel Measurements,” *AIAA AVIATION Forum*, AIAA, 2018-3299, 2018
- [52] Neuhart, D. H., Khorrami, M. R., Choudhari, M. M., “Aerodynamics of a Gulfstream G550 Nose Landing Gear Model,” *AIAA/CEAS Aeroacoustics Conference*, AIAA 2009-3152, 2009
- [53] Khorrami, M.R., Lockard, D.P., Humphreys, Jr., W.M., Choudhari, M.M., and Van de Ven, T., “Preliminary Analysis of Acoustic Measurements from the NASA-Gulfstream Airframe Noise Flight Test,” *AIAA/CEAS Aeroacoustics Conference*, AIAA 2008-2814, 2008
- [54] Lazos, B. S., “Mean Flow Features Around the Inline Wheels of Four-Wheel Landing Gear,” *AIAA Journal*, Vol. 40, No. 2, 2002
- [55] Guo, Y. P., Yamamoto, K. J., Stoker, R. W., “Experimental Study on Aircraft Landing Gear Noise,” *Journal of Aircraft*, Vol. 43, Issue 2, 2006
- [56] Jaegar, S. M., Soderman, P. T., Horne, W. C., James, K. D., “Microphone Array Assessment of an Isolated 26%-Scale High Fidelity Landing Gear,” *AIAA/CEAS Aeroacoustics Conference*, AIAA, 2002-2410, 2002
- [57] Riccidari, T. R., Wolf, W. R., Kreitzman, J., Moffitt, N. J., Bent, P., “An Assessment of High-Fidelity Flow Simulation Methodologies for Noise Prediction of Realistic Landing Gear Configurations,” *AIAA SciTech Forum*, AIAA, 2019-0003
- [58] Oza, U., Hu, Z., Zhang, X., “Effect of Cavity Flow on Landing Gear Aerodynamic Loads,” *AIAA AVIATION Forum*, AIAA, 2015-2288, 2015
- [59] Zhao, K., Okolo, P., Neri, E., Chen, P., Kennedy, J., Bennett, G. J., “Noise Reduction Technologies for Aircraft Landing Gear-A Bibliographic Review,” *Progress in Aerospace Sciences*, 2019
- [60] Rising, J., “Development of a Reduced Area Horizontal Tail for a Wide Body Jet Aircraft,” *NASA CR-172278*, 1984

- [61] Hoang, N. T. B., Bui, B. V., “Investigation of Wind Tunnel Wall Effect and Wing-Fuselage Interference Regarding the Prediction of Wing Aerodynamics and its Influence on the Horizontal Tail,” *Journal of Mechanical Science and Technology*, Vol. 33, Issue 6, 2019
- [62] Hoang, N. T. B., Bui, B. V., “Experimental and Numerical Studies of Wingtip and Downwash Effects on Horizontal Tail,” *Journal of Mechanical Science and Technology*, Vol. 33, Issue 2, 2019
- [63] Nicolosi, F., Vecchia, P. D., Ciliberti, D., “An investigation on Vertical Tailplane Contribution to Aircraft Sideforce,” *Aerospace Science and Technology*, Vol. 28, Issue 1, pp. 401-416, 2013
- [64] Schroiijen, M. J. T., Veldhuis, L. .L. M., Slingerland, R., “Propeller Empennage Interaction Effects on Vertical Tail Design of Multiengine Aircraft,” *Journal of Aircraft*, Vol. 47, No. 4, 2010
- [65] Lin, J. C., Andino, M. Y., Alexander, M. G., Whalen, E. A., Spoor, M. A., Tran, J. T., “An Overview of Active Flow Control Enhanced Vertical Tail Technology Development,” *AIAA Aerospace Sciences Meeting*, AIAA 2016-0056, 2016
- [66] Sawant, S. G., Deem, E. A., Agentis, D. J., Cattafesta, L. N., Arnold, D. P., “Chip-Scale Electrodynamic Active Flow Control for Practical High-Lift Systems,” *Journal of Aircraft Actuators*,” *Proc. IEEE MEMS*, pp. 936–939, 2015
- [67] Rathay, N. W., Boucher, M. J., Amitay, M., Whalen, E., “Performance Enhancement of a Vertical Tail Using Synthetic Jet Actuators,” *AIAA Journal*, Vol. 52, No. 4, 2014
- [68] Jansen, K. E., Rasquin, M., Farnsworth, J. A., Rathay, N., Monastero, M. C., Amitay, M., “Interaction of a Synthetic Jet with Separated Flow over a Vertical Tail,” *AIAA Journal*, Vol. 56, No. 7, 2018
- [69] Andino, M. Y., Lin, J. C., Washburn, A. E., Whalen, E. A., Graff, E. C., Wygnanski, I. J., “Flow Separation Control on a Full-Scale Vertical Tail Model using Sweeping Jet Actuators,” *AIAA Aerospace Sciences Meeting*, AIAA 2015-0785, 2015
- [70] Sccholz, P., Singh, V. M., Gebhardt, A., Kirz, J., Löffler, S., Weiss, J., “The Efficiency of Different Flow Control Methods on a Vertical Tail,” *AIAA SciTech Forum*, AIAA 2020-1537, 2020
- [71] Sachs, G., “Why Birds and Miniscale Airplanes Need No Vertical Tail,” *Journal of Aircraft*, Vol. 44, No. 4, 2007
- [72] Smits, A. J., and Lim, T. T. (Eds), "Flow Visualization: Techniques and Examples, 2nd ed.," Imperial College Press, London, 2012
- [73] Klein, C., Yorita, D., Henne, U., Ondrus, V., Hensch, A. K., Longo, R., Gimbel, AK., van Deetzen, S., “Application of Temperature Sensitive Paint to Investigate Laminar-to-Turbulent Transition on Nacelles,” *AIAA Scitech Forum*, 2020-1608, 2020

- [74] Kirk, T. M., Yarusevych, S., “Vortex Shedding within Laminar Separation Bubbles Forming Over an Airfoil,” *Experiment in Fluids*, Vol 58, Issue 5, 2017
- [75] Iijima, H., Uchiyama, T., Kato, H., "Skin-friction measurement on NASA Common Research Model using oil film in JAXA," *AIAA SciTech Forum*, AIAA 2020-0295, 2020
- [76] Ziliac, G. G., Pulliam, T. H., Rivers, M. B., Zerr, J., Delgado, M., Halcomb, N., Lee, H., “A Comparison of the Measured and Computed Skin Friction Distribution on the Common Research Model,” *AIAA*, AIAA 2011-1129, 2011
- [77] Iyer, P. S., Malik, M. R., “Wall-modeled LES of the NASA Juncture Flow Experiment,” *AIAA SciTech Forum*, AIAA 2020-1307, 2020
- [78] Kegerise, M. A. and Neuhart, D. H., “An Experimental Investigation of a Wing Fuselage Junction Model in the NASA Langley 14- by 22-Foot Subsonic Tunnel,” *NASA/TM-2019-220286*, 2019
- [79] Rumsey, C., “Overview of the NASA Juncture Flow Project,” *AIAA SciTech*, Oral presentation, 2020
- [80] Kegerise, M. A., Neuhart, D. H., Hannon, J. A., Rumsey, C. L., "An Experimental Investigation of a Wing-Fuselage Junction Model in the NASA Langley 14- by 22-Foot Subsonic Wind Tunnel," *AIAA SciTech Forum*, AIAA-2019-0077, 2019
- [81] Kamliya Jawahar, H., Azarpeyvand, M., Il'ario da Silva, C. R., “Experimental Investigation of Flow Around Three-Element High-Lift Airfoil with Morphing Fillers,” *AIAA AVIATION Forum*, AIAA 2017-3364, 2017
- [82] Reese, D. T., Danehy, P. M., Walker, E. L., Rivers, S. M., Goad, W. K., “FLEET Velocimetry in the Common Research Model’s Wing Wake,” *AIAA SciTech Forum*, AIAA 2020-1276, 2020
- [83] Gkiolas, D., Mathioulakis, D., “PIV and Surface Pressure Measurements on a NACA64-418 Airfoil Undergoing Stall Flutter,” *AIAA SciTech Forum*, AIAA 2020-0090, 2020
- [84] Zhu, W., McCrink, H., M., Bons, J. P., Gregory, J. W., “Aerodynamic Performance and Trailing Edge Flow Physics on an Airfoil in an Oscillating Freestream,” *AIAA SciTech Forum*, AIAA 2020-1758, 2020
- [85] Brendel, M., Mueller, T. J., “Boundary-Layer Measurements on an Airfoil at Low Reynolds Numbers,” *Journal of Aircraft*, Vol. 25, No. 7, 1988
- [86] Kirk, T. M., Yarusevych, S., “Vortex Shedding within Laminar Separation Bubbles Forming Over an Airfoil,” *Experiment in Fluids*, Vol 58, Issue 5, 2017
- [87] Gul, M., Uzol, O., Akmandor, I. S., “An Experimental Study on Active Flow Control Using Synthetic Jet Actuators over S809 Airfoil,” *Journal of Physics Conference Series*, Vol. 524, 012101, 2014

- [88] Yen, S. C., Fei, Y. F., “Winglet Dihedral Effect on Flow Behavior and Aerodynamic Performance of NACA0012 Wings,” *Journal of Fluids Engineering*, Vol. 33, Issue 7, 2011
- [89] Alonso, A. M., “Analysis of Drag Reduction in Wingtip Devices,” Bachelor Thesis, University Carlos III de Madrid, 2017
- [90] McLean, D., “Wingtip Devices: What They Do and How They Do It,” *Proceedings of the Boeing Performance and Flight Operations Engineering Conference*, Seattle, WA, 2005
- [91] Siddiqui, N. A., Asrar, W., Sulaeman, E., “Literature Review: Biomimetic and Conventional Aircraft Wing Tips,” *International Journal of Aviation, Aeronautics, and Aerospace*, Vol. 4, Issue 2, 2017
- [92] Healy, F., Cheung, R., Neofet, T., Lowenberg, M., Rezgui, D., Cooper, J., “Folding Wingtips for Improved Roll Performance,” *AIAA SciTech*, AIAA 2021-1153
- [93] Dacles-Mariani, J., Zilliac, G. G., Chow, J. S., Bradshaw, P., “Numerical/Experimental Study of a Wingtip Vortex in the Near Field,” *AIAA Journal*, Vol. 33, No. 9, 1995
- [94] Lombard, J. E. W., Moxey, D., Sherwin, S. J., Hoessler, J. F. A., Dhandapani, S., Taylor, M. J., “Implicit Large-Eddy Simulation of a Wingtip Vortex,” *AIAA Journal*, Vol. 54, No. 2, 2016
- [95] Kubrynski, K., “Wing-Winglet Design Methodology for Low Speed Applications,” *AIAA* 2003-215, 2003
- [96] Gerz, T., Holzäpfel, F., Darracq, D., “Commercial Aircraft Wake Vortices,” *Progress in Aerospace Sciences*, Vol 38, Issue. 3, 2002, pp. 181, 208
- [97] Hallock, J. N., Holzäpfel, F., “A Review of Recent Wake Vortex Research for Increasing Airport Capacity,” *Progress in Aerospace Sciences*, Vol 9, 2018, pp. 27, 36
- [98] Mills, J., Ajaj, R., “Flight Dynamics and Control Using Folding Wingtips: An Experimental Study,” *Aerospace*, Vol. 4, Issue 2, 2017
- [99] Hantrais-Gervois, J. L., Rapin, M., “Aerodynamic and Structural Behaviour of a Wing Equipped with a Winglet at Cruise,” *AIAA* 2006-1489, 2006
- [100] Cui, P., Han, J., “Prediction of Flutter Characteristics for a Transport Wing with Wingtip Devices,” *Aerospace Science and Technology*. Vol. 23, Issue 1, 2012
- [101] Rudolph, P. K., “High Lift Systems on Commercial Subsonic Airliners,” *NASA-CR-4767*, 1994
- [102] Pecora, R., Magnifico, M., Amoroso, F., Monaco, E., “Multi-Parametric Flutter Analysis of a Morphing Wing Trailing Edge,” *The Aeronautical Journal*, Vol. 118, No. 1207, 2014

- [103] Keller, D., “High-Lift Design for a Forward Swept Natural Laminar Flow Wing,” CEAS Aeronautical Journal, pp. 81, 92, 2019
- [104] Pott-Pollenske, M., Friedel, H., “Aeroacoustic Characteristics of Externally Blown Flap Systems,” AIAA 2011-2729
- [105] Eliasson, P., Grundestam, O., Pen, S. H., Yao, H., Davidson, L., Eriksson, L. E., “Assessment of High-Lift Concepts for a Regional Aircraft in the ALONOCO Project,” AIAA Paper 2012-0277, 2012
- [106] Smith, A. M. O., “High-Lift Aerodynamics,” Journal of Aircraft, Vol. 12, No. 6, 1975, pp. 501–530
- [107] Pomeroy, B. W., Diebold, J. M., Ansell, P. J., Selig, M. S., “Study of Burst Wakes in a Multi-Element Airfoil Flowfield,” AIAA Journal, Vol. 52, No. 4, 2014
- [108] Matpadi Raghavendra, A. K., “Reducing the Effects of Wakes on a Flapped-Wing,” AIAA 2018-3959
- [109] Ortiz Melendez, H. D., Coder, J. D., Shmilovich, A., “High-Lift Simulations of Slotted, Natural-Laminar-Flow Airfoils,” AIAA 2019-0290
- [110] Petrov, A. V., Potapchik, A. V., Stepanov, J. G., Gubskiy, V. V., “Adaptive Wing High-Lift System,” Congress of the International Council of the Aeronautical Sciences, 2018
- [111] Fujiwara, G. E. C., Bragg, M., Camello, S., Lum, C., “Computational and Experimental Ice Accretions of Large Swept Wings in the Icing Research Tunnel,” AIAA Aviation, AIAA 2016-3734, 2016
- [112] Bragg, M. B., Broeren, A. P., Blumenthal, L. A., “Iced Airfoil Aerodynamics,” Progress in Aerospace Science, Vol. 41, Issue 5, 2005, pp. 323, 362
- [113] Broeren, A. P., Diebold, J. M., Bragg, M. B., “Aerodynamic Classification of Swept-Wing Ice Accretion,” NASA/TM—2013-216381, 2013
- [114] Broeren, A. P., Potapczuk, M. G., Riley, J. T., Villedieu, P., Moëns, F., Bragg, M. B., “Swept-Wing Ice Accretion Characterization and Aerodynamics,” NASA/TM—2013-216555, 2013
- [115] Zhang, C., Wang, F., Kong, W., Liu, H., “The Characteristics of SLD Icing Accretions and Aerodynamic Effects on High-Lift Configurations,” AIAA 2015-3385, 2015
- [116] Prince Raj, L., Lee, J. W., Myong, R. S., “Ice Accretion and Aerodynamic Effects on a Multi-Element Airfoil under SLD Icing Conditions,” Aerospace Science and Technology, Vol. 85, 2019, pp. 320, 333
- [117] Camello, S. C., “Effect of Ice Accretion Simulation Fidelity on Low-Reynolds Number Swept-Wing Aerodynamic Performance,” Master Thesis, University of Washington, Seattle, WA, 2017

- [118] Li, L., Liu, Y., Hu, H., “An Experimental Study of Dynamic Ice Accretion Process on Aero-Engine Spinners,” AIAA 2017-0551
- [119] Koivisto, P., “A Literature Review on Cold Soaked Fuel Frost Formation and Sublimation,” Trafo Research Report, 2015
- [120] Soinnie, E., Rosnell, T., “Aerodynamic, Laser Scanning and Photogrammetric Measurements of Cold Soaked Fuel Frost,” AIAA 2018-3830
- [121] Joshi, S., Gujarathi, Y. S., “A Review on Active and Passive Flow Control Techniques,” Int. Journal of Recent Tech. in Mech. and Electrical Eng., Vol. 3, Issue 4, 2016
- [122] Kral, L. D., “Active Flow Control Technology,” ASME Fluids Eng. Tech. Brief, 2000
- [123] Tang, H., Salunkhe, P., Zheng, Y., Du, J., Wu, Y., “On the Use of Synthetic Jet Actuator Arrays for Active Flow Separation Control,” Experimental Thermal and Fluid Science, Vol. 57, 2014
- [124] Little, J., Samimy, M., “High-Lift Airfoil Separation with Dielectric Barrier Discharge Plasma Actuation,” AIAA Journal, Vol. 48, No. 12, 2010
- [125] Melton, L. P., Koklu, M., “Active Flow Control using Sweeping Jet Actuators on a Semi-span Wing Model,” AIAA SciTech, AIAA 2016-1817, 2016
- [126] Shmilovich, A., Yadlin, Y., “Active Flow Control for Practical High-Lift Systems,” Journal of Aircraft, Vol. 46, No. 4, 2009
- [127] Yu, X., Liu, B., Jiang, H., “Characteristics of the Tip Leakage Vortex in a Low Speed Axial Compressor,” AIAA, Vol. 45, Issue. 4, 2007
- [128] Kim, S. H., Kim, C., “Separation Control on NACA 23012 using Synthetic Jet,” Aerospace Science and Technology, Vol. 13, 2009
- [129] Wang, L., Luo Z., Xia, Z., Liu, B., Deng, X., “Review of Actuators for High Speed Active Flow Control,” Science China Technological Sciences, Vol. 55, Issue, 8, 2012
- [130] Radespiel, R., Burnazzi, M., Casper, M., Scholz, P., “Active Flow Control for High Lift with Steady Blowing,” The Aeronautical Journal, Vol. 120, No. 1223, 2016
- [131] Abbas, A., de Vicente, J., Valero, E., “Aerodynamic Technologies to Improve Aircraft Performance,” Aerospace Science and Technology, Vol. 28, Issue 1, 2013, pp. 100, 132
- [132] Slotnick, J., Khodadoust, A., Alonso, J., Darmofal, D., Gropp, W., Lurie, E., Mavriplis, D., “CFD Vision 2030 Study: A Path to Revolutionary Computational Aerosciences,”
- [133] Spalart, P. R., Venkatakrishnan, V., “On the Role and Challenges of CFD in the Aerospace Industry,” The Aeronautical Journal, Vol. 120, No. 1223, 2016

- [134] Rumsey, C. L., Slotnick, J. P., Sclafani, A. J., "Overview and Summary of the Third AIAA High Lift Prediction Workshop," *Journal of Aircraft*, Vol. 56, No. 2, 2019
- [135] Ashton, N., Denison, M., Zastawny, M., "3rd High-Lift Workshop Summary Paper - OpenFOAM, STAR-CCM+ & LAVA Simulations on Unstructured grids," *AIAA SciTech Forum*, AIAA 2018-1253
- [136] Rudnik, R., Wilkending, M., "TAU-SOLAR Contribution to the 3rd High Lift Prediction Workshop," *AIAA SciTech Forum*, AIAA 2018-1035
- [137] Langlois, M., Yang, H., Sermeus, K., "Bombardier Contribution to the 3rd AIAA High-Lift Workshop," *AIAA SciTech Forum*, AIAA 2018-1038
- [138] Trapani, G., Brionnaud, R., Holmann, D. M., "XFlow Contribution to the Third High-Lift Prediction Workshop," *AIAA SciTech Forum*, AIAA 2018-2847
- [139] Hambrick E. M., Thomason, N. M., "Conceptual Aircraft Hinge Moment Measurement System," Senior Project, California Polytechnic State University at San Luis, 2010
- [140] Nurohman, C., Arifianto, O., Barecasco, A., "Hinge Moment Coefficient Prediction Tool and Control Force Analysis of Extra-300 Aerobatic Aircraft," *Journal of Physics Conference Series*, Vol. 1005, 2018
- [141] Grismer, D., Grismer, M., Simon, J., Tinapple, J., "An Experimental Investigation of Factors Influencing Hinge Moments," *AIAA paper*, AIAA 2000-4016, 2000
- [142] Baumann, P. H., Bütetfisch, K. A., "Measurement of Hinge Moments and Model Deformations in Wind Tunnels by Means of Moiré Interferometry," *SPIE Optical Techniques in Fluid, Thermal, and Combustion Flow*, Vol. 2546, pp.16-32, 1995
- [143] Ren, Z., Gao, S., Jia, Z., Shang, Y., Gao, Y., "Piezoelectric Sensor of Control Surface Hinge Moment," *Sensors & Transducers*, Vol. 152, Issue 4, 2013
- [144] Ansell, P. J., Bragg, M. B., Kerho, M. F., "Envelope Protection System Using Flap Hinge Moment Measurements," *AIAA Applied Aerodynamics Conference*, AIAA 2010-4225
- [145] Ansell, P. J., Bragg, M. B., Kerho, M. F., "Envelope Protection System for Iced Airfoils Using Flap Hinge Moment," *SAE Technical Paper* 2011-38-0066, 2011
- [146] Ansell, P. J., Kerho, M. F., Bragg, M. B., "Envelop Protection for Contaminant-Induced Adverse Aerodynamics on a Wing Using Flap Hinge Moment Measurement," *AIAA Applied Aerodynamics Conference*, AIAA 2013-2654, 2013
- [147] Ansell, P. J., Kerho, M. F., Bragg, M. B., "Application of a Hinge-Moment-Base Envelope Protection System on a Wing," *Journal of Aircraft*, Vol 51, No. 6, 2014
- [148] Dobrzynski, W., "Almost 40 Years of Airframe Noise Research: What Did We Achieve," *Journal of Aircraft*, Vol. 7, Issue 2, 2010

- [149] Bertsch, L., Snellen, M., Enghardt, L., Hillenherms, C., “Aircraft Noise Generation and Assessment: Executive Summary,” CEAS Aeronaut J 10, 3-9, 2019
- [150] Zhang, Y., Richardson, R., Cattafesta III, L. N., Pascioni, K. A., Choudhari, M. M., Lockard, D. P., Khorrami, M. R., Turner, T. L., “Slat Noise Control Using a Slat Gap Filler,” AIAA AVIATION Forum, AIAA 2020-2553, 2020
- [151] Choudhari, M. M., Khorrami, M. R., “Slat Cove Unsteadiness: Effect of 3D Flow Structures,” AIAA Aerospace Meeting and Exhibit, AIAA 2006-0211, 2006
- [152] Housman, J. A., Stitch, G. D., Kiris, C. C., “Predictions of Slat Noise from the 30P30N at High Angles of Attack using Zonal Hybrid RANS-LES,” Aeroacoustics Conferences, AIAA 2019-2438, 2019
- [153] Silva, G. P. G., Eguea, J. P., Croce, J. A. G., Catalano, F. M., “Aerodynamic Noise Reduction from Wing High-Lift Devices using Dielectric Barrier Discharge Plasma Actuators – Part II (Slat),” Aerospace Science and Technology, Vol. 97, 2019
- [154] Zhang, Y., O’Neill, A., Cattafesta III, L. N., Pascioni, K. A., Choudhari, M. M., Khorrami, M. R., Lockard, D. P., Turner, T., “Assessment of Noise Reduction Concepts for Leading Edge Slat Noise,” AIAA AVIATION Forum, AIAA 2018-3461, 2018
- [155] Kolb, A., Mancini, S., Rossignol, K. S., Ewert, R., “Flap Side-Edge Noise Simulation Using RANS-based Source Modelling,” AIAA AVIATION Forum, AIAA 2020-2581, 2020
- [156] Yamamoto, K., Takaishi, T., Murayama, M., Yokokawa, Y., Ito, Y., Kohzai, M., Arizona, H., Sakai, R., Shoji, H., Ueno, Y., Tsuchimoto, Y., Isotani, K., Hayama, K., Kumada, T., Shimada, A., “FQUROH: A Flight Demonstration Project for Airframe Noise Reduction Technology – the 2nd Flight Demonstration,” AIAA AVIATION Forum, AIAA 2018-4087, 2018
- [157] Isotani, K., Hayama, K., Yokokawa, Y., Murayama, M., Ura, H., Yamamoto, K., “An Aerodynamic Noise Reduction Study for Airframe Noise from Flap Tips,” AIAA/CEAS Aeroacoustics Conference, AIAA 2013-2074, 2013
- [158] Belyaev, I. V., Zaytsev, M. Y., Kopiev, V. F., Ostrikov, N. N., Faranosov, G. A., “Studying the Effect of Flap Angle on the Noise of Interaction of a High-Bypass Jet with a Swept Wing in a Co-Flow,” Acoustical Physics, Vol. 63, No. 1, 2017
- [159] Bertsch, L., Simons, D. G., Snellen, M., “Aircraft Noise: The Major Sources, Modelling Capabilities, and Reduction Possibilities,” DLR-Interner Bericht, 224-2015 A 110 29 S, 2015
- [160] Chiba, K., Imamura, T., Amemiya, K., Yamamoto, K., “Design Optimization of Shielding Effect for Aircraft Engine Noise,” Journal of Environment and Engineering, Vol. 2, Issue 3, 2007
- [161] Agarwal, A., Dowling, A. P., “The Calculation of Acoustic Shielding of Engine Noise by the Silent Aircraft Airframe,” AIAA 2005-2996, 2005

- [162] Vieira, A., Snellen, M., Simons, D. G., "Assessing the Shielding of Engine Noise by the Wings for Current Aircraft using Model Predictions and Measurements," *The Journal of the Acoustical Society of America*, Vol. 146, Issue 2, 2019
- [163] Stoker, R., Guo, Y., Streett, C., Burnside, N., "Airframe Noise Source Locations of a 777 Aircraft in Flight and Comparisons with Past Model Scale Tests," AIAA 2003-3232, 2003
- [164] Werner-Spatz, C., Heinze, W., Horst, P., Radespiel, R., "Multidisciplinary Conceptual Design for Aircraft with Circulation Control High-Lift Systems," *CEAS Aeronautical Journal*, Vol. 3, Issue 2-4, 2012
- [165] Balaji, R., Bramkamp, F., Hesse, M., Ballmann, J., "Effect of Flap and Slat Riggings on 2-D High-Lift Aerodynamics," *Journal of Aircraft*, Vol. 43, No. 5, 2006
- [166] Pagani, C. C., Souza, D. S., Medeiros, M. A. F., "Experimental Investigation on the Effect of Slat Geometrical Configurations on Aerodynamic Noise," *Journal of Sound and Vibration*, Vol. 394, pp. 256-279, 2017
- [167] Lin, J. C., Dominik, C. J., "Parametric Investigation of a High-Lift Airfoil at High Reynolds Numbers," *Journal of Aircraft*, Vol. 34, No. 4, 1997
- [168] Wang, J., Wang, J., Kim, K. C., "Wake/Shear Layer Interaction for Low-Reynolds-Number Flow Over Multi-Element Airfoil." *Experiments in Fluids*, Vol. 60, Issue. 1, 2019
- [169] Qu, Q., Ju, B., Huang, L., Liu, P., "Flow Physics of a Multi-Element Airfoil in Ground Effect," AIAA 2016-0856
- [170] Tian, Y., Li, Z., Liu, P. Q., "Upper Trailing-Edge Flap for Transonic Buffet Control," *Journal of Aircraft*, Vol. 55, No. 1, 2018
- [171] Gerz, T., Holzäpfel, F., Darracq, D., "Commercial Aircraft Wake Vortices," *Progress in Aerospace Sciences*, Vol 38, Issue. 3, 2002, pp. 181, 2018
- [172] Breitsamter, C., "Wake Vortex Characteristics of Transport Aircraft," *Progress in Aerospace Sciences*, Vol. 47, Issue 2, 2011
- [173] Sarpkaya, T., "New Model for Vortex Decay in the Atmosphere," *Journal of Aircraft*, Vol. 37, No. 1, 2000
- [174] Hallock, J. N., Holzäpfel, F., "A Review of Recent Wake Vortex Research for Increasing Airport Capacity," *Progress in Aerospace Sciences*, Vol 9, 2018, pp. 27, 36
- [175] Jacquin, L., Fabre, D., Sipp, D., Theofilis, V., Vollmer, H., "Instability and Unsteadiness of Aircraft Wake Vortices," *Aerospace Science and Technology*, Vol. 7, Issue. 8, 2003, pp. 577, 593
- [176] Holzäpfel, F., "Effects of Environmental and Aircraft Parameters on Wake Vortex Behavior," *Journal of Aircraft*, Vol. 51, No. 5, 2014

- [177] Stephan, A., Holzäpfel, F., Misaka, T., “Aircraft Wake-Vortex Decay in Ground Proximity - Physical Mechanisms and Artificial Enhancement,” *Journal of Aircraft*, Vol. 50, No. 4, 2013
- [178] Holzäpfel, F., Hofbauer, T., Darracq, D., Moet, H., Garnier, F., Gago, C. F., “Analysis of Wake Vortex Decay Mechanisms in the Atmosphere,” *Aerospace Science and Technology*, Vol. 7, Issue 4, 2003
- [179] [9] Thobois, L. Cariou, J-P, “Next Generation Scanning LIDAR Systems for Optimizing Wake Turbulence Separation Minima,” *Journal of Radars*, Vol. 6, No. 6, 2017
- [180] Lain, Y., Shyy, W., “Laminar-Turbulent Transition of a Low Reynolds Number Rigid of Flexible Airfoil,” *AIAA Journal*, Vol. 45, No. 7, 2007
- [181] Coder, J. G., Ortiz-Melendez, H. D., “Transitional Delayed Detached-Eddy Simulation of Multielement High-Lift Airfoils,” *Journal of Aircraft*, Vol. 56, No. 4, 2019
- [182] “Transition on a High-Lift Swept Wing in the European Project EUROLIFT, 2004” *Journal of Aircraft*, Vol. 41, No. 5, 2004
- [183] Boiko, A., Westin, K., Klingmann, B., Kozlov, V., & Alfredsson, P., “Experiments in a Boundary Layer Subjected to Free Stream Turbulence. Part II: The Role of TS-waves in the Transition Process,” *Journal of Fluid Mechanics*, 281, 219-245, 1994
- [184] Eliasson P., Hanifi A., Peng S. H., “Influence of Transition on High-Lift Prediction for the NASA Trap Wing Model,” *AIAA* 2011-3009, 2011
- [185] Bokhorst, E., “Forcing of the Primary and Secondary Crossflow Instability,” PhD Thesis, City, University of London
- [186] Fiore, M., Vermeersch, O., Forte, M., Casalis, G., François, C., “Characterization of a highly efficient chevron-shaped anti-contamination device,” *Experiments in Fluids*, Vol. 57, Issue 4, 2016
- [187] Somashekar, V., Immanuel Selwyn Raj, V., “Experimental Investigation on Laminar Separation Bubble Over an Airfoil – A Review,” *Indian J. Sci. Technol.*, Vo. 11, Issue, 11, 2018
- [188] Olson, D. A., Katz, A. W., Naguib, A. M., Koochesfahani, M. M., Rizzetta, D. P., Visbal, M. R., “On the Challenges in Experimental Characterization of Flow Separation Over Airfoils at Low Reynolds Number,” *Experiment in Fluids*, 54, 1470 (2013)
- [189] Aholt, J., Finaish, F., “Active Flow Control Strategy of Laminar Separation Bubbles Developed over Subsonic Airfoils at Low Reynolds Number,” *AIAA paper* 2011-0733, 2011
- [190] Saric, W. S., Reed, H. L., “Crossflow Instabilities – Theory & Technology,” *AIAA paper*, 2003-0771, 2003

- [191] Serpieri, J., Kotsonis, M., “Three-Dimensional Organisation of Primary and Secondary Crossflow Instability,” *Journal of Fluid Mechanics*, Vol. 799, pp. 200-245, 2016
- [192] Lemarechal, J., Costantini, M., Klein, C., Kloker, M. J., Würz, W., Kurz, H. B. E., Streit, T., Schaber, S., “Investigation of Stationary Crossflow Instability Induced Transition with the Temperature-Sensitive Paint Method,” *Experimental Thermal and Fluid Science*, Vol. 109, 2019
- [193] Radeztsky Jr., R. H., Reibert, M. S., Saric, W. S., “Effect of Isolated Micron-Sized Roughness on Transition in Swept-Wing Flows,” *AIAA Journal*, Vol. 37, No. 11, 1999
- [194] R. Mukund, Viswanath, P. R., Narasimha, R., Prabhu, A., Crouch, J. D., “Relaminarization in Highly Favourable Pressure Gradients on a Convex Surface,” *Journal of Fluid Mechanics*, Vol. 566, pp. 97-115, 2006
- [195] Viswanath, P. R., Mukund, R., Crouch, J. D., “Relaminarization on Swept Leading Edges Under High-Lift Conditions,” *AIAA Journal*, Vol. 44, No. 11, 2006

APPENDIX A

RESEARCH TOPICS PERFORMED WITH THESE MODELS			
	EUROLIFT	JAXA JSM	NASA HL-CRM
OVERVIEW	[2][5]	[11][12]	[1][23][26][34]
2.1 CRUISE, TAKEOFF, LANDING	ALL	ALL	ALL
2.2 BRACKETS		[21] (FTF)	
2.3 NACELLE/PYLON	[8][9][10]	[16][17][18]	[31] [32]
2.4 HORIZONTAL & VERTICAL TAIL			
2.5 LANDING GEAR NOISE			
2.6 FLOW VISUALIZATION	[2][8][9][10]	[13][14] [16][17][18]	[29] [31] [32]
2.7 BOUNDARY LAYER STUDIES	[2][8][9][10]	[13][14] [16][17][18]	[27] [28] [29][30] [31][32]
2.8 WINGTIPS			
2.9 MULTI-ELEMENT FLAPS			
2.10 ICE SHAPE TESTING			
2.11 ACTIVE FLOW CONTROL			[26][27][28][29][30]
2.12 LOW RE CFD	[3] [6]	[22]	[33]
2.13 HINGE MOMENTS			
2.14 AIRFRAME NOISE		[15]	[26] (listed in the test plan)
2.15 LE/TE POSITIONING	[6]		[25]
2.16 WAKE STUDIES			
2.17 TRANSITION STUDIES	[4]		