PARAMETRIC EXPLORATION OF WING-BODY JUNCTION FLOW USING
COMPUTATIONAL FLUID DYNAMICS

A Thesis by

Bryan C. Hinson

Bachelor of Science, Wichita State University, 2006

Submitted to the Department of Aerospace Engineering
and the faculty of the Graduate School of
Wichita State University
in partial fulfillment of
the requirements for the degree of
Master of Science

December 2012

© Copyright 2012 by Bryan C. Hinson
All Rights Reserved

PARAMETRIC EXPLORATION OF WING-BODY JUNCTION FLOW USING COMPUTATIONAL FLUID DYNAMICS

The following faculty members have examined the final copy of this thesis for form and content, and recommend that it be accepted in partial fulfillment of the requirement for the degree of Master of Science with a major in Aerospace Engineering.

Klaus A. Hoffmann, Committee Chair

Kamran Rokhsaz, Committee Member

Ikram Ahmed, Committee Member

ACKNOWLEDGMENTS

I am fortunate to have been mentored by Dr. Neal Pfeiffer and Dr. Chittur “Venkat” Venkatasubban early in my professional career. Neal and Venkat have had tremendous influence on me, both professionally and academically, and I am grateful for this. I especially want to thank Venkat for giving me the opportunity to pursue computational fluid dynamics (CFD) as a young engineer at Hawker Beechcraft Corporation—that opportunity made this thesis possible. Thanks is also due to Dr. Ray Maple, who has guided my studies of CFD and helped to steer my thesis towards completion. I would like to thank Dr. Klaus Hoffmann for advising my thesis, and Dr. Kamran Rokhsaz and Dr. Ikram Ahmed for serving on my thesis committee. I thank my parents for always holding academics in high regard. To my wife, thank you for always believing in my abilities.

This work was supported in part by Hawker Beechcraft Corporation through the use of computer systems and software.

ABSTRACT

In this thesis, wing-body junction flow is studied parametrically using computational fluid dynamics (CFD) in an attempt to understand the effects of junction flow on aircraft drag, with a focus on application to large business jet or commercial transport aircraft. A CFD methodology is validated against detailed experimental data for a junction flow. The same methodology is validated against a high Reynolds number, transonic wind tunnel test of a wing. CFD results for a wing with a leading-edge strake (an aerodynamic surface designed to reduce flow separation, thereby reducing aircraft drag) are presented and compared to experimental data, and the effects of scaling this strake are explored using CFD. The effectiveness of the strake on a swept wing is compared to the same for a straight wing. Finally, the results from this parametric study are successfully applied to sizing a leading-edge strake for a commercial transport aircraft. It is demonstrated that a systematic approach, starting with a simple validated model and building up to a realistic aircraft application, can build confidence in CFD results.

TABLE OF CONTENTS

Chapter		Page
1.	INTRODUCTION	1
1.1	Background: Junction Flow	1
1.1.1	Physics	1
1.1.2	Parametric Effects	6
1.1.3	Modeling	9
1.1.4	Controlling Junction Flow	13
1.2	Synopsis	19
2.	METHODOLOGY	20
2.1	USM3D Flow Solver	20
2.2	VGrid Unstructured Grid Generator	21
3.	VALIDATION	24
3.1	Low-Speed Junction Flow	24
3.1.1	Grids and Boundary Conditions	25
3.1.2	Results	31
3.2	High Reynolds Number, Transonic Flow	45
3.2.1	Geometry	46
3.2.2	Grids and Boundary Conditions	47
3.2.3	Results	50
4.	RESULTS AND DISCUSSION	56
4.1	Grid Convergence Analysis	56
4.1.1	Grid Generation Parameters	56
4.1.2	Devenport Strake Definition	58
4.1.3	Grid Convergence Results	58
4.2	Parametric Study	59
4.2.1	Geometries	60
4.2.2	Grids and Flight Conditions	62
4.2.3	Results	62
4.3	Aircraft Application	67
4.3.1	Strake Sizing	67
4.3.2	Geometry	71
4.3.3	Grids	72
4.3.4	Results	74

TABLE OF CONTENTS (continued)

Chapter	Page
5. CONCLUSIONS	81
5.1 Summary	81
5.2 Future Research	83
REFERENCES	84
APPENDICES	89
A. Non-Dimensional Wall Distances for Tetrahedral Cell Centers	90
B. Field Velocity Plots Comparing CFD to Rood Wing Experiment	92
C. Pressure Coefficient Plots Comparing CFD to NASA TM-4227 Data	104
C.1 CFD vs. Published TM-4227 Wing Data	104
C.2 CFD vs. Corrected TM-4227 Wing Data	115
C.3 CFD vs. Published TM-4227 Tunnel Wall Data	126

LIST OF TABLES

Table		Page
3.1	Reference quantities for Rood wing model.....	25
3.2	General grid information for Rood cases.....	30
3.3	Grid viscous layers growth parameters.....	31
3.4	Definition of analysis planes on Rood wing geometry	38
3.5	CFD error for all Fleming et al. data planes	44
3.6	General grid information for NASA TM-4227 cases	49
3.7	Boundary layer parameters for NASA TM-4227 grids	49
3.8	Pressure coefficient correction to NASA TM-4227 data	52
4.1	Target grid point spacings for inviscid, isotropic grid in junction region	57
4.2	Grid point spacing parameters for viscous layers growth on Rood wing models	57
4.3	General grid information for Rood wing grid convergence analysis	58
4.4	Lift and drag deltas for Rood wing grid refinement study	59
4.5	Complete list of parameter combinations used in parametric study	60
4.6	Reference quantities for NASA Common Research Model	67
4.7	Bluntness factors for several airfoils.....	70
4.8	Grid point spacing parameters for growth of viscous layers on CRM models	74
4.9	General grid information for CRM wing-body models	74
A.1	Centroid calculation for cells adjacent to viscous wall	91

LIST OF FIGURES

Figure	Page
1.1 Sketch of wing-body junction and resulting horseshoe vortex.	2
1.2 Rood airfoil.	2
1.3 Oil flow patterns made by turbulent flow around a Rood wing mounted to a flat plate.....	3
1.4 Oil flow visualization of DLR-F6 wing-body intersection.....	4
1.5 Illustrated time sequence of flow events in leading-edge region of Rood wing-body junction.	5
1.6 Barber's proposed model of the relationship between onset boundary layer thickness and junction flow.	9
1.7 Conceptual sketch of junction fillet.	14
1.8 Leading-edge strake geometry.	15
1.9 Contours of U from experiment in wake of Rood wing with and without leading-edge strake.....	16
1.10 Schematic of test article: square cylinder with thin, swept circular cylinder installed.	17
1.11 Smoke visualization of wall-mounted square cylinder.....	17
2.1 Example of VGrid sourcing on Rood wing model.	22
3.1 Schematic of CFD domain for low-speed validation.	26
3.2 Boundary layer profile from solution on grid vgrid.6.	27
3.3 Trailing-edge detail on Rood wing grid.	27
3.4 Surface grid, vgrid.3.	28
3.5 Surface grid, vgrid.4.	28
3.6 Surface grid, vgrid.5.	29
3.7 Surface grid, vgrid.6.	29
3.8 Typical USM3D time history for Rood junction flow.....	31

LIST OF FIGURES (continued)

Figure	Page
3.9 Isometric view of CFD model of Rood wing showing locations of surface pressure taps and sample measurement planes.	32
3.10 Rood wing-section pressures at $y/b_{ref} = 0.042$; CFD vs. experiment [18].	33
3.11 Rood wing-section pressures at $y/b_{ref} = 0.125$; CFD vs. experiment [18].	34
3.12 Rood wing-section pressures at $y/b_{ref} = 0.457$; CFD vs. experiment [18].	34
3.13 Rood wing-section pressures at $y/b_{ref} = 0.541$; CFD vs. experiment [18].	35
3.14 Pressure coefficient on wind tunnel floor from experiment.	36
3.15 Difference in pressure coefficient on wind tunnel floor: CFD (vgrid.3) vs. experiment.	36
3.16 Difference in pressure coefficient on wind tunnel floor: CFD (vgrid.7) vs. experiment.	36
3.17 Difference in pressure coefficient on wind tunnel floor: CFD (vgrid.8) vs. experiment.	37
3.18 Pressure coefficients at far row of pressure taps on tunnel floor, from CFD and experiment.	37
3.19 Contours of U in Plane 6 ($x/c = 0.42$) of Rood wing junction from experiment [2] and USM3D solutions.....	39
3.20 Contours of U in Plane 11 ($x/c = 1.5$) of Rood wing junction from experiment [2] and USM3D solutions.....	40
3.21 Contours of W in Plane 6 ($x/c = 0.42$) of Rood wing junction from experiment [2] and USM3D solutions.	41
3.22 Contours of W in Plane 11 ($x/c = 1.5$) of Rood wing junction from experiment [2] and USM3D solutions.	42
3.23 Root mean square of U error.	43
3.24 Root mean square of W error.	44
3.25 Plan view of NASA TM-4227 wind tunnel model.	45

LIST OF FIGURES (continued)

Figure	Page
3.26 Conceptual sketch of flowfield in adaptive wall test section.	46
3.27 High-speed wing grid with shaded faces.....	46
3.28 Schematic of CFD domain for high-speed validation.	47
3.29 Top view of high-speed wing grid.	48
3.30 Leading-edge detail on high-speed wing grid; tip on left, root on right.	48
3.31 Trailing-edge detail on high-speed wing grid; root on left, tip on right.	49
3.32 Wing-surface pressures from TM-4227 experiment and CFD solution.	51
3.33 Wind tunnel wall pressures from TM-4227 experiment and corresponding CFD flowfield pressures.	51
3.34 Wing-surface pressures from TM-4227 experiment (corrected) and CFD solution.	53
4.1 Effect of grid refinement on computed drag coefficient.	59
4.2 Shaded surface grids for strake geometries on Rood wing.	61
4.3 Shaded surface grids for strake geometries on swept Rood wing.	61
4.4 Effects of strake length on Rood wing drag polar.	63
4.5 Effects of strake length on 30° swept Rood wing drag polar.	63
4.6 Drag deltas from parametric changes to Rood wing.	64
4.7 Contours of U from CFD in wake of Rood wing with varying sweep angles and strake sizes.	65
4.8 Contours of U from CFD in wake of Rood wing with varying sweep angles and strake sizes.	66
4.9 CRM wing and horizontal stabilizer planforms (taken from [37]).	68
4.10 Drag deltas from parametric changes to Rood wing, scaled according to equation (4.3).	70
4.11 Isometric views of CRM geometry with and without strake.....	71

LIST OF FIGURES (continued)

Figure	Page
4.12 Top, side, and bottom views of CRM geometry with strake.	72
4.13 Isometric views of junction region of CRM grids with and without strake.	73
4.14 Lift and drag characteristics of CRM wing-body from AOA sweeps at wind tunnel Reynolds number.	75
4.15 Lift and drag characteristics of CRM wing-body from AOA sweeps at flight Reynolds number.	76
4.16 Contours of U in plane at $x/c = 0.64$ from USM3D for CRM at $M_\infty = 0.85$, $C_L = 0.50$, Re 40 million	77
4.17 Contours of U in wake plane at $x/c = 1.5$ from USM3D for CRM at $M_\infty = 0.85$, $C_L = 0.50$, Re 40 million	78
4.18 Contours of U in wake plane at $x/c = 3.0$ from USM3D for CRM at $M_\infty = 0.85$, $C_L = 0.50$, Re 40 million	78
4.19 Pressure coefficient contours on upper surface of CRM wing.	79
A.1 Representation of three tetrahedra that constitute one prismatic layer of boundary layer cells.	90
B.1 U contours in Plane 5: CFD (top), Experiment [2] (bottom).	92
B.2 U contours in Plane 6: CFD (top), Experiment [2] (bottom).	93
B.3 U contours in Plane 7: CFD (top), Experiment [2] (bottom).	94
B.4 U contours in Plane 9: CFD (top), Experiment [2] (bottom).	95
B.5 U contours in Plane 10: CFD (top), Experiment [2] (bottom).	96
B.6 U contours in Plane 11: CFD (top), Experiment [2] (bottom).	97
B.7 W contours in Plane 5: CFD (top), Experiment [2] (bottom).	98
B.8 W contours in Plane 6: CFD (top), Experiment [2] (bottom).	99
B.9 W contours in Plane 7: CFD (top), Experiment [2] (bottom).	100
B.10 W contours in Plane 9: CFD (top), Experiment [2] (bottom).	101

LIST OF FIGURES (continued)

Figure		Page
B.11	W contours in Plane 10: CFD (top), Experiment [2] (bottom).	102
B.12	W contours in Plane 11: CFD (top), Experiment [2] (bottom).	103

LIST OF ABBREVIATIONS

2D	Two-Dimensional
3D	Three-Dimensional
AIAA	American Institute of Aeronautics and Astronautics
AOA	Angle of Attack
AUSM	Advection Upstream Splitting Method
AWTS	Adaptive Wall Test Section
BL	Boundary Layer
CATIA	Computer Aided Three-Dimensional Interactive Application
CFD	Computational Fluid Dynamics
CFL	Courant-Friedrichs-Lewy
CRM	Common Research Model (NASA)
DDES	Delayed DES
DES	Detached Eddy Simulation
DLR	Deutsches Zentrum für Luft- und Raumfahrt (German Aerospace Center)
DPW-4	Fourth AIAA Drag Prediction Workshop
ERCOTAC	European Research Community on Flow, Turbulence and Combustion
GCI	Grid Convergence Index
GMRES	Generalized Minimal Residual
GUI	Graphical User Interface
IGES	Initial Graphics Exchange Specification
LaRC	Langley Research Center (NASA)
LDV	Laser Doppler Velocimetry

LIST OF ABBREVIATIONS (continued)

LES	Large Eddy Simulation
LOLS	Line of Low Shear
NACA	National Advisory Committee for Aeronautics (now NASA)
NASA	National Aeronautics and Space Administration
ONERA	Office National d'Etudes et de Recherches Aerospatiales (National Institute for Aerospace Studies and Research), France
RANS	Reynolds-Averaged Navier-Stokes
RMS	Root Mean Square
RSM	Reynolds Stress Model
SST	Shear Stress Transport
TCT	Transonic Cryogenic Tunnel (NASA)
TE	Trailing Edge

LIST OF SYMBOLS

$^{\circ}$	Degrees (angular unit of measure)
AR	Wing aspect ratio
a	Viscous grid layer geometric growth rate; also VGrid source strength
BF	Bluntness factor
b	Viscous grid layer growth acceleration
b_{ref}	Wing reference span length
c	Airfoil chord length
c_{ref}	Wing reference chord length
C_D	Drag coefficient
C_L	Lift coefficient
C_p	Pressure coefficient
k	Turbulent kinetic energy
M_{∞}	Freestream Mach number
MDF	Momentum deficit factor
$Re_{\odot}$	Reynolds number based on length $\odot$; if $\odot$ is omitted, then it is assumed to be equal to wing reference chord
r_0	Airfoil leading-edge radius
S_{ref}	Wing reference area
s_T	Distance along airfoil surface to maximum thickness point
T	Airfoil maximum thickness
U	Mean x-component of velocity
W	Mean z-component of velocity
x_T	Chordwise position of airfoil maximum thickness
x_{ref}	X-location of moment reference

LIST OF SYMBOLS (continued)

y^+	Non-dimensional wall-normal distance
y_{ref}	Y-location of moment reference
z_{ref}	Z-location of moment reference
Δ	Change in value
Δs	Local grid-point spacing
δ	Boundary layer thickness
δ_1	Distance from surface of first grid point
ϵ	Rate of turbulent energy dissipation
θ	Boundary layer momentum thickness
$\Lambda_{c/4}$	Wing quarter-chord sweep angle
λ	Wing taper ratio
π	Mathematical constant, $\equiv \cos^{-1}(-1)$ in radians
ω	Rate of turbulent energy dissipation per unit turbulent kinetic energy

CHAPTER 1

INTRODUCTION

The desire for more efficient air vehicles is ever-present. Reducing fuel burn of a large business jet or commercial transport aircraft reduces operating costs and engine emissions, or can increase aircraft range. Of the many ways to increase aircraft efficiency, including using more efficient engines and designing lighter vehicle structures, the one that primarily concerns the aerodynamicist is aircraft drag reduction.

Of particular interest in aircraft configuration aerodynamics is the drag associated with a wing-body junction. In addition to the interference drag between the wing and body along their mutual intersection, there is a complex flow structure that forms when the fuselage boundary layer approaches the wing leading edge. The boundary layer (BL) separates in the presence of the adverse pressure gradient created by the wing, and the resulting vortex wraps around the wing and continues downstream, resembling a horseshoe in shape. Thus, this flow structure is commonly known as a *horseshoe vortex* and is the source of additional aircraft drag.

This thesis explores modeling of the junction flow using computational fluid dynamics (CFD) and the ability of using CFD to study drag reduction of the wing-body junction.

1.1 Background: Junction Flow

Junction flow occurs when a wall-bounded flow interacts with a protrusion attached to that wall. Examples include bridge piers at a river bottom, the junction of a sail or conning tower with the hull of a submarine, and, the focus of this work, a wing attached to the fuselage of an airplane.

1.1.1 Physics

Time-Mean Characteristics

Junction flows have been studied with both laminar and turbulent approach boundary layers. In either case, the basic mechanism for the formation of the horseshoe vortex is the same. As the flow approaches the appendage, an adverse pressure gradient is felt by the

approaching boundary layer, due to the stagnation region on the appendage. This adverse pressure gradient causes boundary layer separation and the formation of multiple vortices in front of the appendage. These vortices then continue downstream, stretching as they wrap around the appendage, thus becoming the classic *horseshoe vortex* flow structure [1] (see Figure 1.1).

Figure 1.1: Sketch of wing-body junction and resulting horseshoe vortex. Measurement planes and coordinate axes were used in experiment (from Fleming et al. [2]).

By far, the most popular geometry studied pertaining to junction flow is the Rood airfoil. The Rood airfoil, named after it's inventor, Edwin P. Rood, is defined by a 3:2 semi-ellipse mated with the tail of a National Advisory Committee for Aeronautics (NACA) 0020 airfoil at its maximum thickness location (see Figure 1.2).

Figure 1.2: Rood airfoil.

A typical surface oil flow visualization for a cylindrical wing of Rood airfoil (Rood wing) mounted to a flat plate with turbulent approach boundary layer is shown in Figure 1.3. The upstream separation line and saddle point are clearly visible. According to researchers at Virginia Polytechnic Institute, the line closer to the appendage is a line of low shear (LOLS) because pigment can be seen to cross over this line, whereas this would not be the case with a true separation line [2, 3]. These two lines meet downstream at approximately the airfoil maximum thickness location. Researchers at Texas A&M argue that the LOLS is really a secondary separation line and continuously refer to it as such [4].

Figure 1.3: Oil flow patterns made by turbulent flow around a Rood wing mounted to a flat plate. Circles are laser Doppler velocimetry (LDV) measurement locations (from Devenport and Simpson [3]).

Another feature typical of wing-body junction flows is the aft corner separation. This feature was of particular interest at the Third AIAA Drag Prediction Workshop [5]. Figure 1.4 shows a clear example of an aft corner separation. Also note the hint of a saddle point and separation line, and an LOLS or secondary separation line at the leading edge of the wing-body intersection. These two lines appear to merge at about 60% root chord. These patterns are consistent with the low-speed experiments mentioned earlier.

Figure 1.4: Oil flow visualization of Deutsches Zentrum für Luft- und Raumfahrt (DLR)-F6 wing-body intersection, performed at Office National d’Etudes et de Recherches Aérospatiales (ONERA) S2MA wind tunnel. $M_\infty = 0.75$, $C_L = 0.5$. The aft corner separation is identified by red-colored oil, which originated on the lower surface of the wing and found its way onto the upper surface (adapted from Rudnik et al. [5]).

Time-Dependent Characteristics of Turbulent Junction Flow

Turbulent junction flow is always unsteady and quite complex [1]. While the large-scale behavior of junction flow is characterized by the horseshoe vortices, the behavior of turbulent junction flow is aperiodic and sometimes “bimodal,” meaning that the flow switches between two modes at irregular intervals; this was reported in detail by Devenport and Simpson [3] using LDV velocity measurements in the symmetry plane forward of a Rood wing/flat plate junction. These two modes are called the *backflow* mode and the *zero-flow* mode, referring to the behavior of velocity time-histories in the leading-edge region of the appendage [3]. Kim et al. [6] studied this region of flow in the Rood/plate junction

using a hydrogen bubble technique in a water tunnel and further defined its time-dependent behavior. A series of sketches that qualitatively illustrate the bimodal time sequence is shown in Figure 1.5.

Figure 1.5: Illustrated time sequence of flow events in leading-edge region of Rood wing-body junction. The percentage in parenthesis represents the approximate time proportion of the event (from Kim et al. [6]).

The existence of bimodal behavior is not limited to the Rood airfoil shape, as bimodal velocity histograms were also reported for plate-mounted cylinders with a parallel centerbody shape, a teardrop shape, and an NACA 0015 airfoil [7]. However, thinner airfoils with smaller leading-edge radii did not exhibit bimodal behavior. Bimodal behavior was also reported for the Rood wing with a 50% thinner approach boundary layer, indicating that airfoil shape is the primary driver of bimodality [3]. Turbulence energy production and turbulent stresses an order of magnitude higher than in the upstream boundary layer were found in the bimodal zone [1]. From their experimental measurements in the plane of symmetry at the leading edge of the Rood wing, Devenport and Simpson [3] estimated components of the turbulent kinetic energy equation. They concluded that the region of bimodal flow, which includes the time-mean junction vortex, is one of intense turbulent energy production due to the large-scale unsteadiness in that region, rather than conventional shear-layer mechanisms. The components of Reynolds stress production in the normal directions are equally important as those for shear stress production in accounting for turbulent kinetic energy production in the bimodal region [1, 3]. Furthermore, the variations in dissipation of the Reynolds normal stresses near the wall necessitate use of an anisotropic dissipation tensor to match experimental data [8].

1.1.2 Parametric Effects

Geometric Parameters

Several experiments have been conducted regarding the geometric parameters that affect junction flow. Wei et al. [9] investigated various cylinder cross-sections, including square, circular, and rhombuses with varying leading-edge angles. Based on their findings, they concluded that the cross-section bluntness had a strong effect on the location and strength of the primary horseshoe vortex. The sharper leading edge of the diamond shape created lower-strength primary vortices that formed closer to the appendage compared to the circle or square shape. This relationship between leading-edge bluntness and vortex location and strength is intuitive, given the underlying mechanism for the formation of the horseshoe

vortices explained earlier. The blunter leading-edge shapes create larger regions of high pressure, affecting more flow with the resulting adverse pressure gradient when compared to the shapes with sharper leading edges. Simpson [1] reports a *bluntness factor*,

$$BF = \frac{1}{2} \frac{r_0}{x_T} \left[\frac{T}{s_T} + \frac{s_T}{x_T} \right] \quad (1.1)$$

which was correlated with the existence and strength of the bimodal velocity fluctuations at the leading edge of several wing-body junctions with different airfoil shapes. The Sandia 1850 airfoil ($BF = 0.0133$) and the NACA 0012 airfoil ($BF = 0.0287$) did not exhibit any bimodal behavior. The NACA 0015 airfoil, the parallel centerbody airfoil, the Rood airfoil, and the teardrop airfoil, with BF values ranging from 0.0452 to 1.071, exhibited bimodal velocity fluctuations in the leading-edge region and higher values of pressure fluctuations.

Ahmed and Khan performed experiments to determine the effect of wing sweep on junction flow [4]. They tested wall-mounted Rood wings with several sweep angles, ranging $\pm 45^\circ$, in turbulent flow at Re_T of 47,500 to 190,000. Surface flow visualization showed the presence of a saddle point of separation, and primary and secondary separation lines for all sweep angles at all Reynolds numbers. As Reynolds number increased, the saddle point of separation moved downstream, while the secondary separation line moved upstream in the symmetry plane. With positive wing sweep (wing tip aft of wing root), both the primary and secondary separation lines moved downstream as a result of decreased streamwise pressure gradient and increased spanwise flow. Negative sweep did not have as strong of an effect on separation line location. All movements of separation lines were within $\pm 0.3T$ of the zero-sweep locations for all wing sweeps at any given Reynolds number. Changes in Reynolds number affected separation line locations by no more than $\pm 0.1T$. These researchers also looked at the spanwise location of the bifurcating streamline, which is the location along the wing stagnation line where the streamlines split into spanwise and recirculating flow. Through LDV measurements taken at $Re_T = 95,000$ ($Re_\theta = 4,950$), they concluded that the bifurcating streamline moved towards the body (wall) for increasing positive sweep angles,

and away from the body for negative sweep angles. The vortex location in the symmetry plane also moved aft for increasing positive sweep, and upstream for negative sweep angles. Surface pressure measurements showed that the local minimum in pressure in the symmetry plane is the location of the vortex. The local pressure maximum in the symmetry plane was not at the saddle point, but rather closer to the secondary separation point, indicating it to be a point of reattachment. Surface pressures outside of the plane of symmetry showed the vortex footprint, but further downstream the surface pressures indicated that the vortex had lifted off the body surface. It was unclear whether the swept wings were obtained through rotating or shearing the zero-sweep wing. This is an important question, because rotating the wing will change the root airfoil shape. However, given the consistent data trends with wing sweep angle and the locations of pressure ports on the body close to and all around the root airfoil, it is probably safe to assume that the wings were sheared, preserving the root airfoil shape.

Flow Parameters

For a wall-mounted wing at nonzero angle of attack (AOA), the approaching flow encounters a blunter shape compared to the zero AOA condition, which leads to increased flow distortion [1]. The low-pressure side vortex leg is pushed further away from the wing, and the high-pressure side vortex strength is increased [10, 11].

Boundary layer characteristics have a strong impact on junction flow. Barber [12] proposed a relationship between onset boundary layer height, and the size and location of the horseshoe vortex, as well as the size of the aft corner separation (see Figure 1.6). Barber's wake surveys showed that a thicker boundary layer ($\delta/c = 0.12$) produces more total pressure loss away from the wing surface due to the horseshoe vortex legs compared to a thinner boundary layer produced just in front of the wing. The thinner boundary layer produced appreciably more pressure loss in the wake of the wing trailing edge (TE) due to the larger corner separation.

Figure 1.6: Barber's proposed model of the relationship between onset boundary layer thickness and junction flow (adapted from Barber [12]).

Fleming et al. [2] experimentally investigated the effect of the approaching turbulent boundary layer on the junction flow around a Rood cylinder. In comparing experimental data for five approach boundary layers, differing in θ/T and Re_θ , they defined a *momentum deficit factor*,

$$MDF = (Re_T)(Re_\theta) \quad (1.2)$$

which correlates many of the differences observed between the flow data sets. Flows with larger values of MDF showed smaller flow distortions and stronger near-wall vortical structures. Flows with smaller values of MDF showed more flow distortion due to a more energetic separation, which is a result of comparatively more momentum closer to the wall. They also found that non-dimensionalized, peak core vorticity levels varied proportionally to MDF . Also, flows with higher MDF had larger trailing vortex leg separation in the wake, which seems to be evidence of stronger, longer-lasting vortex legs for these flows.

1.1.3 Modeling

This section provides some highlights of methods that have been used to model junction flow, focusing mainly on turbulence modeling. While purely analytical methods have been used to estimate properties of the junction flow [13], this section focuses on numerical approaches commonly used in modern CFD solvers.

Devenport and Simpson [14] evaluated the accuracy of six turbulence models based on LDV measurements taken around a Rood wing junction flow. Three were eddy viscosity models: a basic mixing-length model, Cebeci-Smith, and Johnson-King. The other three were the k - ϵ model, an algebraic stress model, and Bradshaw’s model. These turbulence models were evaluated analytically by supplying either the measured velocity data (for eddy-viscosity models) or the turbulent kinetic energy (for the other three models) and comparing the resulting shear stress magnitude with that measured from the experiment. The authors concluded that the Cebeci-Smith and algebraic stress models gave the most accurate results for this flow. The k - ϵ model performed the worst of all the models considered. Because their data shows no clear relationship between the angle of turbulent shear stress and the velocity gradient vector, the authors conclude that accurate modeling of the full shear stress transport equation is needed. This requires accurate prediction of the pressure-strain term, most likely based on experimental data.

Ölçmen and Simpson [15] evaluated the accuracy of four turbulent diffusion models compared to LDV measurements taken around a Rood wing in junction flow. The four triple-product relations investigated were Daly-Harlow, Hanjalic-Launder, Mellor-Herring, and Lumley. These models were compared to experimental data taken at four different locations in the junction flow: a two-dimensional (2D) location, and three locations in a plane forward of the airfoil maximum thickness location and perpendicular to the wing surface. These three locations correspond to the vortex core, the separation line, and a point outside of the separation where the flow behaves much like the 2D boundary layer. The authors noted that these turbulent diffusion models are sensitive to the distribution of viscous dissipation. The Mellor-Herring model did not perform well. All of the models failed to capture the magnitude of the fluctuating velocity triple products. The authors suggest this could mean that $2k/\epsilon$, which is found in all four equations for the triple product, may not be the correct scaling parameter.

Kubendran et al. [16] performed an experiment of a wing junction flow with and without a leading-edge strake and modeled this flow with an incompressible Reynolds-averaged Navier-Stokes (RANS) solver. Viscous terms were treated with a thin-layer approximation, and the Baldwin-Lomax turbulence model was used. The CFD calculations were able to qualitatively predict the effect of adding the leading-edge strake to the wing, in that the boundary layer profiles in planes normal to the freestream were more uniform with the strake installed. The CFD predicted the boundary layer in these planes to be thicker than the experiment showed, and the vortex location was predicted to be further away from the wing than seen in the experiment; however, the authors admit that the freestream Reynolds number was the only constraint used in the CFD, and no comparison was presented regarding the upstream boundary layer thickness.

Paciorri et al. [17] compared RANS calculations on the Rood geometry with European Research Community on Flow, Turbulence and Combustion (ERCOTAC) data [18] using both a structured and an unstructured solver. Both solvers give steady-state solutions to the incompressible, pressure-based RANS equations. The structured solver used cell-centered, finite-volume discretization and multi-stage Runge-Kutta with local time stepping and multi-grid. The unstructured solver used a vertex-centered, finite-volume formulation with residual distribution. Implicit time integration was solved using a preconditioned generalized minimal residual (GMRES) algorithm. Solutions were obtained for both solvers using the one-equation Spalart-Allmaras turbulence model and the Lam and Bremhorst $k-\epsilon$ turbulence model. A grid convergence study was performed on the unstructured model using the standard grid convergence index (GCI) by Roache. It showed small values of uncertainty ($< 10\%$) in pressure and velocities in most areas of the flowfield; however, higher values of uncertainty in pressure were seen near the vortex core in the symmetry plane. Both structured and unstructured solutions showed large but similar changes in the near-wall boundary layer profile in the symmetry plane upstream of the wing leading edge with increasing grid refinement. However, that same boundary layer profile compared remarkably well between

the finest grids of the structured and unstructured models. Both turbulence models predicted the vortex core in the plane at the wing trailing edge, perpendicular to the freestream, to be nearer the wing than the experimental data showed, with the Spalart-Allmaras model being closer to the experiment. Both turbulence models also over-predicted the viscous dissipation in this plane, with the Spalart-Allmaras model again being closer to the experiment. The authors attributed this last discrepancy to over-estimation of the eddy viscosity by both turbulence models.

Fu et al. [19] explored implementations of RANS, detached eddy simulation (DES), and delayed DES (DDES) by comparing computed results to experimental data. The ERCOFTAC Rood case and the NASA TN D-712, a transonic delta-wing-body, were both used in comparisons. The advantage of the DES methods is that large eddy simulation (LES)-type calculations are performed to capture large separations, while RANS is used in the boundary layer where it is most suited. The authors concluded that the RANS modeling provided satisfactory results at low AOA. In results from both test cases, the DES method predicted flow separation to be far upstream of the experiment. This behavior is attributed to grid-dependent switching between LES and RANS modes. The DDES methodology corrected this behavior and was the only methodology that gave satisfactory results for the delta wing at high AOA. DDES also computed the turbulent kinetic energy in the horseshoe vortex at the symmetry plane more accurately than the other two methods.

Gand et al. [10] performed both steady RANS calculations with several turbulence models and LES calculations and compared the results to experimental data. The geometry was an NACA 0012 wing, specifically chosen because it was expected to exhibit an aft corner separation based on *MDF* and *BF*. For the RANS calculations, a grid convergence study was performed, and models with and without wind tunnel walls were analyzed. Turbulence models used with the RANS calculations were Spalart-Allmaras (with and without a rotation correction by Dacles-Mariani et al.), Wilcox’s k - ω model, Menter’s shear stress transport (SST) k - ω model, and the Speziale-Sarkar-Gratski Reynolds stress model in its low-Reynolds

number implementation. While the experimental data did not indicate the presence of an aft corner separation, every RANS calculation predicted the existence of one. The RANS calculations slightly over-predicted the total pressure in the measurement planes, while the LES calculations under-predicted total pressure and were overly dissipative in the convective region. The error in the LES result could be due to the time-averaging approach taken with post-processing the LES data. Unlike the RANS calculations, the LES computation does match the experiment in the aft corner region.

1.1.4 Controlling Junction Flow

This section presents methods for controlling junction flow with the goal of reducing or eliminating the horseshoe vortex and/or its inherent unsteadiness and thereby reducing drag, vibration, and/or noise. Only passive means of controlling the flow are discussed here. Active means of flow control such as boundary layer suction rarely “buy their way on board” any practical aircraft due to the added cost, complexity, weight, and energy consumption of such a system.

Devenport et al. [11] experimentally studied the effect of a fillet on a plate-mounted Rood wing with turbulent onset boundary layer. The fillet was a quarter-circle section of radius $0.53T$ that wrapped around the entire wing/body intersection (see Figure 1.7). This fillet did not reduce the unsteadiness of the separation at the leading edge of the wing and did not reduce the strength of the horseshoe vortices. Hot-wire velocity measurements in the wake revealed that the fillet increased the area of distorted time-averaged boundary layer flow by 25%; this indicates that the fillet actually increased the size and/or strength of the time-mean horseshoe vortex, probably by increasing the root section effective leading-edge radius. At 6° AOA, the fillet pushed the low pressure side vortex further away from the wing compared to the wing without fillet. Surface-pressure fluctuations in the wake beneath the vortex legs were also increased with the addition of the fillet. Therefore, while this type of fillet might reduce the wing-body interference drag, it does not favorably alter the leading-edge separation that forms the horseshoe vortex. The reported increase in wake distortion indicates that this fillet may have actually increased the drag due to the horseshoe vortex.

Figure 1.7: Conceptual sketch of junction fillet (from Devenport et al. [11]).

Ölçmen and Simpson [20] investigated leading-edge fences as a passive means of reducing the large pressure fluctuations in the leading-edge region of several wall-mounted wings. This was most likely motivated by the desire to decrease flow noise on submarines by making the flow more steady. These fences were attached to the leading edge of the wing, normal to the spanwise direction. Various fence configurations were tested, with a double fence being the most effective at reducing surface pressure fluctuations. Overall, the fences did eliminate the bimodal flow behavior and shifted the velocity time-histories closer to a Gaussian distribution, indicating that the leading-edge separation was more chaotic and breaking down into smaller vortices.

Devenport et al. [21] experimentally studied the effects of a leading-edge strake on a Rood wing in turbulent junction flow. These experiments were very similar to those performed for the junction fillet mentioned previously [11]. The strake was designed to smooth the adverse pressure gradient at the leading edge of the wing (see Figure 1.8). At zero AOA, the strake eliminates leading-edge separation and, thus, the horseshoe vortex and bimodal unsteadiness. At non-zero AOA, a separation forms on the strake on the high-pressure side, although the resulting distortion in the wake is still less than that for the wing without the

strake. These characteristics are evident in the surface oil flows and also in the hot-wire wake measurements (Figure 1.9). Pressure fluctuations at the wall beneath the downstream vortices were reduced for cases with the thin boundary layer at all angles of attack, with the low-pressure side vortex leg being affected more at non-zero AOA. This experiment showed that the leading-edge strake is an effective means of treating the adverse effects of junction flow.

Figure 1.8: Leading-edge strake geometry: (a) isometric view, (b) side view (from Devenport et al. [21]).

Figure 1.9: Contours of U from experiment in wake of Rood wing at $x/c = 3.0$, with and without leading-edge strake: (a) baseline wing, 0° AOA, thin BL; (b) wing with strake, 0° AOA, thin BL; (c) baseline wing, 0° AOA, thick BL; (d) wing with strake, 0° AOA, thick BL; (e) baseline wing, 6° AOA, thin BL; (f) wing with strake, 6° AOA, thick BL. Contour values shown in part (a). Arrows on Z axis mark locations of microphone measurements. Taken from Devenport et al. [21].

As mentioned earlier, Kubendran et al. [16] also experimentally investigated a leading-edge strake on straight wings of NACA airfoils with similar results. Their strake had a linear planform and was formed by stretching the airfoil forward of its maximum thickness point by $c/3$ at the wing root, tapering to zero at a distance of $c/6$ spanwise. Flow visualization indicated that this strake eliminated the leading-edge separation, and calculations based on wake surveys revealed that the strake reduced the drag.

Wei et al. [9] proposed a unique solution for modifying the horseshoe vortex. Based on the knowledge that small leading-edge radii and swept leading edges lead to weaker horseshoe vortices, they placed a thin, swept, circular cylinder in front of a square cylinder such that one end of the thin cylinder was attached to the wall, and the other end was attached to the square cylinder at some distance from the wall. A schematic of this arrangement is shown in Figure 1.10. Smoke wire flow visualizations (Figure 1.11) and surface pressure measurements confirmed that the thin, swept cylinder had the desired effects of reducing the size of the horseshoe vortices, reducing distortion in the wake, and eliminating the adverse pressure gradient in front of the blunt cylinder.

Figure 1.10: Schematic of test article: square cylinder with thin, swept circular cylinder installed (from Wei et al. [9]).

(a) Baseline square cylinder

(b) With 40° swept cylinder, $d/D=0.2$

Figure 1.11: Smoke visualization of wall-mounted square cylinder at $Re=8,000$ (from Wei et al. [9]).

In support of the Third AIAA Drag Prediction Workshop, Vassberg et al. [22] designed a fairing to eliminate the aft corner separation on the DLR-F6 configuration (see Figure 1.4). The DLR-F6 wing makes an angle of about 60° with the fuselage in the region of the separation. Thus, the fairing was designed in two pieces. The first piece was a large fairing that made the wing-body intersection closer to 90° ; CFD calculations showed this fairing succeeded in reducing the size of the aft corner separation. The second fairing was a smaller, local blister to fill in the remaining separated zone. The addition of this second fairing onto the first completely eliminated the aft corner separation in the CFD solution.

Leading-Edge Strake Design Methods

The leading-edge strake is by far the most common (and maybe the only) means by which commercial aircraft designers reduce horseshoe vortex drag. Photographs from Milstein [23] show that the Airbus A319, Airbus A320, and Boeing 747-400 employ strakes on their wings and horizontal stabilizers, presumably to reduce drag. The Airbus A340-300, McDonnell Douglas MD-82, and Boeing 737-700 have strakes on their wings. Interestingly, the Boeing 747-200 does not have a leading-edge strake on its wing, while the 747-400 does have a sizable strake on its wing.

Van Oudheusden et al. [13] presented a method for optimizing a leading-edge strake, designed to eliminate flow separation and minimize contamination on a laminar wing. A panel method along with an integral boundary layer was used to calculate the boundary layer properties along the attachment line along the body and strake.

Green and Whitesides [24] presented a similar approach, except OVERFLOW was employed to calculate the flow properties. They developed a relationship between skin-friction coefficient and turning angle to iteratively define the optimum leading-edge strake shape.

In both of these cases, control of the approaching boundary layer was the goal of the design methodology.

1.2 Synopsis

In this work, wing-body junction flow is studied parametrically using CFD in an attempt to further understand the effects of junction flow on aircraft drag, with a focus on application to large business jet or commercial transport aircraft. A CFD methodology is validated against detailed experimental data for a junction flow. The same methodology is validated against a high Reynolds number, transonic wind tunnel test of a wing. CFD results for a wing with a leading-edge strake are presented and compared to experimental data, and the effects of scaling this strake are explored using CFD. The effectiveness of the strake on a swept wing is compared to the same for a straight wing. Finally, the results from this parametric study are applied to sizing a leading-edge strake for a commercial transport aircraft.

CHAPTER 2

METHODOLOGY

The CFD methodology employed in this work is one that is common to the contemporary, commercial aircraft industrial environment [25]. Steady-state RANS calculations on unstructured grids are used exclusively. The Spalart-Allmaras turbulence model is also used exclusively because it is the most prevalent model used in civil aeronautical applications.

2.1 USM3D Flow Solver

The CFD solver chosen for this work is USM3D, an unstructured, three-dimensional Navier-Stokes solver available from NASA Langley Research Center (LaRC) [26]. USM3D uses finite volume discretization of the integral form of the Navier-Stokes equations on tetrahedral cells. Inviscid fluxes are computed across cell faces using Roe’s flux-difference splitting; the advection upstream splitting method (AUSM) is also available but only recommended for high-supersonic cases [27]. Second-order spacial accuracy is achieved through a Laplacian-weighted averaging scheme for reconstructing the cell-centered values to the nodes, and finally to the faces, of the tetrahedral cells. Viscous stresses are approximated at the cell faces using a linear reconstruction involving the solution quantities (conservation variables) at the three nodes of the face and the two neighboring cell centroids. Derivatives of flow quantities used in the viscous stress terms are then computed from these values.

For steady-state calculations, implicit Gauss-Seidel time-stepping is used. Convergence is accelerated by using local time-stepping with local Courant-Friedrichs-Lewy (CFL) number dependent on cell aspect ratio. This is useful because an isotropic tetrahedral cell can remain stable with a much larger CFL value than a highly stretched one.

The one-equation Spalart-Allmaras turbulence model (solving to the wall) was used in this work. This model can also be run with wall functions. Other turbulence models available in USM3D include DES with Spalart-Allmaras, linear k - ϵ model, Girimaji algebraic Reynolds stress model (RSM), Shih/Zhu and Lumley algebraic RSM, and Menter’s SST model.

2.2 VGrid Unstructured Grid Generator

Unstructured grids were generated using GridTool and VGrid, also available from NASA LaRC. GridTool is a graphical user interface (GUI)-based preprocessor in which the user constructs surface patches on the geometry to be gridded with the goal of obtaining a water-tight model that is then sent to VGrid for the grid generation. VGrid first generates the surface mesh patch by patch using the advancing front method. It then uses the advancing layers method to “grow” boundary layer cells off the surface. The boundary layer cells resemble prisms that have been divided into tetrahedra, although they are constructed one cell at a time. VGrid then generates the rest of the volume grid using the advancing front method. Grid-point spacings are controlled by “sources,” which are defined in GridTool. The sources allow for generating high-quality grids that contain both isotropic and highly anisotropic cells anywhere in the flow domain.

Two versions of VGrid were used in this work: VGrid 3.9 and VGrid 4.1. In VGrid 3.9, a smooth distribution of grid-point spacing is achieved by solving an elliptic partial differential equation (Poisson equation) on a uniform Cartesian background grid with user-specified sources and Dirichlet boundary conditions [28]. In VGrid 3.9, point and line sources are used to set spacings. VGrid 4.1 also contains volume sources (sphere, cylinder, and cone) to increase control of grid spacings in regions of the flow volume. Figure 2.1 shows examples of line and volume sources on the Rood wing model. In this example, the line sources are placed spanwise inside the wing to control the spacing on the wing in general, and the volume sources are placed for more control of the point spacings in the junction.

Figure 2.1: Example of VGrid sourcing on Rood wing model, showing line sources, and cone and sphere volume sources.

The advancing front method generates the actual grid points and tetrahedra simultaneously based on the spacings set in the field. Cells are constructed adjacent to existing cells and the grid front moves through the flow domain until the entire volume is filled with cells. The initial advancing front step usually results in some holes in the mesh (but no negative volume cells). An additional step is usually required to fill these holes. This additional step (called Postgrid) can usually be batched but sometimes needs to be done manually via the program's GUI. Postgrid is also used to improve the grid quality by remeshing cells with included angles below a user-defined threshold.

VGrid 4.1 does not use the structured background grid for determining cell spacings. Instead, the spacing Δs at any given point in the field is simply equal to the minimum of the values of a decay function evaluated at that point for each source in the domain [29]. The decay function is given by

$$\Delta s_{|\mathbf{r}|} = \Delta s_0 (1 + (\Gamma/a))^\eta \quad (2.1)$$

where

$$\eta = \left(1 + \frac{|\mathbf{r}|}{\Delta \bar{s}} \Omega\right) \frac{\ln \left(1 + \frac{|\mathbf{r}| \Gamma}{\Delta s_0 a}\right)}{\ln \left(1 + \frac{\Gamma}{a}\right)} \quad (2.2)$$

and $|\mathbf{r}|$ is the distance between the source and the point of interest in the field. The global geometric growth rate Γ is set to a single value for the entire grid. It controls the rate of decay for point spacings from all sources. A typical value for Γ is 0.1. The source strength a is unique to each source and nominally set to unity; a value greater than unity makes the source stronger and propogates its spacing farther into the field; a value less than unity does the opposite.

In both versions of VGrid, the boundary layer grid spacing is controlled by

$$\delta_i = \delta_1[1 + a(1 + b)^{(i-1)}]^{(i-1)} \quad (2.3)$$

where δ_1 is the height of the first grid point off the viscous surface, a is the geometric growth rate, and b is the growth acceleration. These three parameters are user-defined, usually based on a target y^+ and the number of boundary layer points desired. Because δ_1 is a global distance, the target y^+ can only occur at a certain location of interest, usually the midpoint of the wing mean aerodynamic chord, if applicable.

CHAPTER 3

VALIDATION

A comprehensive, publicly available experimental dataset of a transonic junction flow could not be found. Thus, validation of the flow solver and gridding methodologies used in this work was performed in two separate steps: one for a low Reynolds number, low-speed junction flow, and the other for a high Reynolds number, transonic flow. The most extensive, publicly available set of experimental data for a junction flow is the low-speed ERCOFTAC Case 8 dataset [18]. This was used for the low-speed junction flow validation. Transonic validation was performed using experimental data from NASA for a low-aspect ratio, high-speed wing [30].

3.1 Low-Speed Junction Flow

Validation of a low-speed junction flow was performed against publicly available data from ERCOFTAC Case 8 [18]. This data, taken from Fleming et al. [2], is for a Rood wing at zero AOA, mounted to a flat plate (wind tunnel floor). The data contains velocity components obtained experimentally using a hot wire anemometer and laser Doppler velocimetry. Surface pressures at points on the wing and the tunnel floor are also available.

The Fleming et al. experiment [2] was completed at the Virginia Tech Department of Atmospheric and Ocean Engineering low-speed boundary layer tunnel at a nominal freestream velocity of 27 m/s and $Re_\theta = 6,300$ ($M_\infty \approx 0.077$, $Re \approx 484,000$). The Rood wing model had a chord length of 305 mm and a span of 229 mm; other reference parameters are given in Table 3.1.

TABLE 3.1
REFERENCE QUANTITIES FOR ROOD WING MODEL

Rood Wing		
S_{ref}	69,845.0	mm ²
c_{ref}	305.00	mm
b_{ref}	229.00	mm
x_{ref}	76.250	mm
y_{ref}	0.00	mm
z_{ref}	0.00	mm
λ	1.00	
AR	0.751	
$\Lambda_{c/4}$	0°	

USM3D is a density-based, Navier-Stokes solver, which does not employ low-Mach preconditioning. Blazek [31] suggests $M_\infty \approx 0.2$ as a lower bound for such solvers, and in this author’s experience, $M_\infty = 0.2$ is a good lower bound for obtaining robust solutions in USM3D. Thus, these validation calculations were all performed at $M_\infty = 0.2$, $Re = 484,000$, with the assumption that compressibility effects would be small enough in the CFD solutions to be negligible.

The goal of this validation effort was to demonstrate that the chosen combination of grid generator and flow solver can capture the physics relevant to the junction flow. First, CFD boundary conditions are verified through surface pressure comparisons with experiment. Then, it is confirmed that the formation of a horseshoe vortex is predicted by CFD, and comparisons between CFD and experimental streamwise and cross-flow velocity components are presented.

3.1.1 Grids and Boundary Conditions

Several grids were built during this validation effort, all of which utilize the coordinate system defined by Figure 1.1. A schematic of the CFD domain is presented in Figure 3.1. The upstream boundary for all grids was located 2104 mm upstream of the wing leading

edge, which is the distance required for a turbulent flat plate boundary layer to match the published boundary layer height of 36.8 mm at $x = -0.5c$ for the Fleming et al. dataset. A representative boundary layer profile for one of the CFD solutions presented here is plotted in Figure 3.2. All grids were built for only the positive z half of the wing, utilizing the z -symmetry of the wing airfoil for these zero AOA runs. The $z = 0$ plane boundary was set to a symmetry plane boundary condition, and the wind tunnel floor ($y = 0$ plane boundary) was set to a no-slip boundary condition for all cases. The trailing-edge surface discretization was defined by eight grid faces across the thickness of the half-trailing edge, akin to 16 faces across the full-wing trailing edge (see Figure 3.3).

Figure 3.1: Schematic of CFD domain for low-speed validation. Symmetry plane (orange) divides wing in half, and no-slip boundary condition was applied to wing (blue) and wind tunnel floor (green).

Figure 3.2: Boundary layer profile from solution on grid vgrid.6; the boundary layer height reported by Fleming et al. [2] is marked by the blue line.

Figure 3.3: Trailing-edge detail on Rood wing grid. Only the positive z half of the wing is shown.

Three grids with increasing refinement in the forward region of the junction were generated using VGrid 3.9. These are referred to as vgrid.3, vgrid.4, and vgrid.5; surface grids are shown in Figures 3.4, 3.5, and 3.6, respectively. Grid vgrid.6 was generated using

VGrid 4.1 and employed volume sources to refine the entire junction region including its near wake. Surface point spacing in the forward junction of vgrid.6 is approximately equal to that of vgrid.4, but refinement was carried aft about four chord lengths from the wing leading edge (Figure 3.7). Solutions on these grids used characteristics-based farfield boundary conditions on all boundaries other than the no-slip and the symmetry plane boundary conditions presented in Figure 3.1. Based on results from Paciorri et al. [17], the farfield boundary condition planes in these grids were located farther away from the wing model than the actual wind tunnel walls (see Figure 3.1).

Figure 3.4: Surface grid, vgrid.3. Image on right is close-up of leading-edge region.

Figure 3.5: Surface grid, vgrid.4. Image on right is close-up of leading-edge region.

Figure 3.6: Surface grid, vgrid.5. Image on right is close-up of leading-edge region.

Figure 3.7: Surface grid, vgrid.6. Lower right image is close-up of leading-edge region.

Grids vgrid.7 and vgrid.8 were designed to explore different methods of specifying the wind tunnel boundaries. Grid vgrid.7 is the same as vgrid.4, except that the maximum y -boundary was relocated to be coincident with the wing tip and set to a symmetry plane boundary condition to test a potential wind tunnel wall endplate effect. Grid vgrid.8 has similar spacing parameters to those of vgrid.3, but its outer boundaries were located to match the wind tunnel dimensions and all set to viscous walls (except for the wing symmetry plane) to investigate potential tunnel blockage effects.

Characteristics of all grids are summarized in Tables 3.2 and 3.3. Regarding the y^+ value of the first grid point off of the surface, it is important to note that, because USM3D is a cell-centered solver, the y^+ value at the solution quantities will be, on average, half of the value listed in the table. This is clarified in Appendix A. Some grids were built with very low first-point y^+ spacings and geometric growth rates a to guarantee capture of as much flow detail as was practically possible. An independent grid convergence analysis is presented in Section 4.1.

TABLE 3.2
GENERAL GRID INFORMATION FOR ROOD CASES

Grid ID	VGrid Version	Boundary Faces	Total Cells
vgrid.3	3.9	128,438	7,319,068
vgrid.4	3.9	150,828	10,455,147
vgrid.5	3.9	226,366	21,915,596
vgrid.6	4.1	206,572	13,816,221
vgrid.7	3.9	142,502	8,953,722
vgrid.8	3.9	123,574	7,834,978

TABLE 3.3

GRID VISCOUS LAYERS GROWTH PARAMETERS

Grid ID	Target $Re \times 10^{-6}$	First Point y^+	δ_1	a	b	Target No. BL Points
vgrid.3	0.484	2.0	0.0273	0.045	0.02	40
vgrid.4	0.484	0.5	0.0068	0.068	0.02	40
vgrid.5	0.484	0.5	0.0068	0.068	0.02	40
vgrid.6	0.484	0.5	0.0068	0.068	0.02	40
vgrid.7	0.484	0.5	0.0068	0.068	0.02	40
vgrid.8	0.484	2.0	0.0273	0.045	0.02	40

3.1.2 Results

Solver Convergence Criteria

A typical time history for a single run in USM3D is plotted in Figure 3.8. Residual drop is a commonly used parameter in assessing convergence of a CFD solution; however, because the time history of the force coefficients is monotonic, the rate of change of force coefficients is also used as a convergence criterion.

Figure 3.8: Typical USM3D time history for Rood junction flow.

Surface Pressures

Surface pressures from the experiment were available on the wing and wind tunnel floor. Pressure tap locations are shown in Figure 3.9 in three sets: one set for four stations on the wing, another set on the tunnel floor near the wing, and a third set makes a single row of taps, further away from the wing. Comparisons will be made for all three sets of pressure taps.

Figure 3.9: Isometric view of CFD model of Rood wing showing locations of surface pressure taps and LDV measurement Planes 6 ($x/c = 0.42$) and 11 ($x/c = 1.5$). Pressure taps were located on the wing (magenta), tunnel floor near the wing (white), and further away from the wing (yellow). Contours of U from CFD are plotted on the planes for reference.

Wing pressures from CFD runs are compared to the experimental data in Figures 3.10 through 3.13. An inviscid, two-dimensional result from XFOIL [32] (an interactive computer program for the analysis and design of subsonic isolated airfoils) is shown on those plots as a reference. The pressure coefficients plotted for vgrid.3 are representative of those for vgrid.3 through vgrid.6. They match the experimental data everywhere on the wing sections except at the maximum thickness location, where the experimental data shows significantly lower pressures at the outboard sections.

The pressures from the solution on vgrid.7 give the expected result of agreeing with the two-dimensional, inviscid solution from XFOIL, due to the addition of the outboard symmetry plane. The inboard-most section over-predicts the pressure minimum compared to XFOIL, probably due to three-dimensional effects there. The outboard-most sections show slightly lower pressure levels than XFOIL, probably due to the presence of the boundary layer. This agreement further validates the methods employed here.

Grid vgrid.8 was designed to determine if tunnel blockage could be the reason for the lower-than-predicted experimental pressure levels at the wing's maximum thickness. While the pressures from vgrid.8 approach the reported experimental pressures at the maximum thickness location for all but the inboard-most station, the vgrid.8 solution under-predicts the pressure at all other locations along the chord, especially the pressure recovery on the tail of the airfoil. If the published pressures at maximum thickness are accurate, then there is an unknown phenomenon, more complex than simply the presence of solid wind tunnel walls, responsible for said pressure levels.

Figure 3.10: Rood wing-section pressures at $y/b_{ref} = 0.042$; CFD vs. experiment [18].

Figure 3.11: Rood wing-section pressures at $y/b_{ref} = 0.125$; CFD vs. experiment [18].

Figure 3.12: Rood wing-section pressures at $y/b_{ref} = 0.457$; CFD vs. experiment [18].

Figure 3.13: Rood wing-section pressures at $y/b_{ref} = 0.541$; CFD vs. experiment [18].

Wind tunnel floor pressures from the experiment are plotted in Figure 3.14. Figures 3.15 through 3.17 show differences between CFD pressures and those in Figure 3.14. Figure 3.15 is representative of results on grids vgrid.3 through vgrid.6. Reasonable agreement was achieved, with the most discrepancy occurring around the wing maximum thickness, as was the case with the wing-section results. Figures 3.16 and 3.17 show results for grids vgrid.7 and vgrid.8, respectively. These results are also similar to those seen in the wing-section pressure coefficients. The solution on grid vgrid.7 shows altogether lower pressures on the tunnel floor, while pressures for grid vgrid.8 are significantly lower.

Figure 3.18 shows pressure coefficients for CFD and the experiment for the row of pressure taps that were located on the tunnel floor, farther from the wing (yellow points in Figure 3.9). Again, data for grid vgrid.3 is representative of that for vgrid.4, vgrid.5, and vgrid.6. And again, the same trend is evident: vgrid.7 yields slightly lower pressures, while vgrid.8 yields significantly lower pressure than the published experimental data. Overall, these pressure comparisons lead to the conclusion that the boundary conditions used in the

initial grids (vgrid.3 through vgrid.6) yield accurate surface pressures and are acceptable for use in further validation.

Figure 3.14: Pressure coefficient on wind tunnel floor from experiment.

Figure 3.15: Difference in pressure coefficient on wind tunnel floor: CFD (vgrid.3) vs. experiment (solutions on grids vgrid.4, vgrid.5, and vgrid.6 are similar to this).

Figure 3.16: Difference in pressure coefficient on wind tunnel floor: CFD (vgrid.7) vs. experiment.

Figure 3.17: Difference in pressure coefficient on wind tunnel floor:
CFD (vgrid.8) vs. experiment.

Figure 3.18: Pressure coefficients at far row of pressure taps on tunnel
floor, from CFD and experiment.

Field Velocities

Comparisons between CFD and data from the Fleming et al. [2] experiment for velocity components in streamwise planes are presented here. The streamwise measurement planes are defined in Table 3.4 and they correspond to those shown in Figure 1.1. Results in Planes 6 and 11 are presented here because they are representative of the complete data set.

They are shown in the context of the CFD model in Figure 3.9. Results for the remaining planes are included in Appendix B.

TABLE 3.4

DEFINITION OF ANALYSIS PLANES ON ROOD WING GEOMETRY

Plane	x/c
5	0.18
6	0.42
7	0.64
9	0.93
10	1.05
11	1.50
12	3.00

Figures 3.19 and 3.20 show U contours in Planes 6 and 11, respectively, for the Fleming et al. experiment [2] and the CFD calculations. Comparing the CFD solutions to the experiment, the general flow structure is predicted quite well. The low-momentum “tongue” of fluid that is pushed away from the wall by the horseshoe vortex and the adjacent introduction of higher-momentum fluid into the boundary layer closer to the wing surface can clearly be seen in both the experiment and the CFD solutions. The most noticeable difference between CFD and the experiment in Figures 3.19 and 3.20 is the underprediction of U by CFD in the low-momentum tongue. This discrepancy is exacerbated by grid refinement, as seen in Figure 3.20, where vgrid.6 is the only grid with refinement in the region of Plane 11. In Plane 6 (Figure 3.19), the CFD is predicting $\frac{\partial U}{\partial y}$ in the region between the low-momentum tongue and the wing to be monotonic or zero, while the experimental data shows a sharp peak where the high-momentum fluid is injected into the boundary layer by the horseshoe vortex. Both of these discrepancies are most likely due to inadequacies in the turbulence modeling. The RANS approach used here does not capture any anisotropy of the Reynolds stress tensor. However, the location of the low-momentum tongue and the velocity magnitudes outside of the tongue are predicted quite well.

Figure 3.19: Contours of U in Plane 6 ($x/c = 0.42$) of Rood wing junction from experiment [2] and USM3D solutions.

Figure 3.20: Contours of U in Plane 11 ($x/c = 1.5$) of Rood wing junction from experiment [2] and USM3D solutions.

Figures 3.21 and 3.22 show W contours in Planes 6 and 11, respectively, for CFD and the experiment. The vorticity of the fluid can be seen there, with fluid moving away from the tunnel centerline along the tunnel floor and toward the tunnel centerline further away from the tunnel floor. In this case, grid refinement appears to bring the CFD solutions closer to matching experiment, as the peak negative W value is matched more closely with

the finest grids. As with the U contours, the locations of primary flow features and velocity magnitudes in the CFD solutions are overall closely matched with experiment in these plots.

Figure 3.21: Contours of W in Plane 6 ($x/c = 0.42$) of Rood wing junction from experiment [2] and USM3D solutions.

Figure 3.22: Contours of W in Plane 11 ($x/c = 1.5$) of Rood wing junction from experiment [2] and USM3D solutions.

Root mean square (RMS) error for each measurement plane for U and W for all grids is plotted in Figures 3.23 and 3.24, respectively. Table 3.5 lists the RMS error for all data planes combined.

In Figure 3.23, grids vgrid.3, vgrid.4, and vgrid.5 trend as expected in Planes 6, 7, and 9, with error reducing with increasing grid refinement. As expected, grid vgrid.6 matches closely with grid vgrid.4 at those planes, because they have equal surface point spacings in that region. However, these trends do not continue in planes downstream of Plane 9, due to the effects of grid refinement on the low-momentum tongue discussed above. Grid vgrid.6 performs worst of all grids in Planes 11 and 13, except for grid vgrid.8. Again, this most likely highlights the shortcomings of applying RANS calculations to this type of flow problem. The use of a coarser grid for the purpose of matching experimental data is not justifiable; the modeling approach must show grid convergence to be deemed acceptable.

In Figure 3.24, the W error distributions for all grids except vgrid.8 match closely. W is mostly unaffected by grid resolution within the limits of this study.

Figure 3.23: Root mean square of U error.

Figure 3.24: Root mean square of W error.

TABLE 3.5

CFD ERROR FOR ALL FLEMING ET AL. DATA PLANES

Grid ID	RMS	
	$\Delta \left(\frac{U}{U_{ref}} \right)$	$\Delta \left(\frac{W}{U_{ref}} \right)$
vgrid.3	0.0563	0.0223
vgrid.4	0.0531	0.0213
vgrid.5	0.0533	0.0214
vgrid.6	0.0580	0.0218
vgrid.7	0.0551	0.0218
vgrid.8	0.0716	0.0248

Overall, the methodology presented here is deemed suitable for this work. The main flow features are captured in the CFD solutions, including surface pressures, velocity magnitudes, and the momentum deficit in the wake due to the presence of the horseshoe vortex. A grid convergence study on drag will be presented in Section 4.1.

3.2 High Reynolds Number, Transonic Flow

Validation of high Reynolds number, transonic flow was performed against experimental data published by NASA on a low-aspect ratio, thin wing (NASA TM-4227, Chu and Lawing [30]). The wind tunnel model was a trapezoidal wing with NACA 64A-105 airfoil section and was tested in the NASA 0.3-meter Transonic Cryogenic Tunnel (TCT). A plan view of the wind tunnel model is presented in Figure 3.25.

Figure 3.25: Plan view of NASA TM-4227 wind tunnel model (taken from [30]).

The 0.3-meter TCT is a closed-circuit wind tunnel that uses nitrogen gas as the test medium. The test section used in this case was a 13-inch square adaptive wall test section (AWTS), which has rigid side walls with boundary layer removal ducts. The upper and lower walls are flexible and are driven by computer-controlled pushrods to change their shape. The objective when operating the AWTS is to approximate an upper and lower streamline, achieved through iterating the shape of the walls at each test condition, thus eliminating the effect of their presence (see Figure 3.26). The original intention for the AWTS was for two-dimensional airfoil testing; however, the model tested here is a three-dimensional wing, mounted to a turntable on one of the test section side walls.

Figure 3.26: Conceptual sketch of flowfield in adaptive wall test section (taken from [33]).

3.2.1 Geometry

The geometry for the wing model for use in the CFD validation was generated in Computer Aided Three-Dimensional Interactive Application (CATIA) v5. The airfoil was constructed using polynomial fits to data points from Abbott and Doenhoff [34] for thickness and camber profiles, and checked against data points provided in Chu and Lawing [30]. The wing is a simple ruled surfaced using this airfoil shape at the root and tip. The wing trailing-edge thickness and the wing root fairing were defined by point distributions from Chu and Lawing [30]. A rounded wing tip shape was added to ensure robust flow solution convergence. A representation of the wing surfaces is shown in Figure 3.27.

Figure 3.27: High-speed wing grid with shaded faces.

3.2.2 Grids and Boundary Conditions

A schematic of the CFD domain is presented in Figure 3.28. The wing fairing was placed on a symmetry plane to approximate the wind tunnel wall with boundary layer removed. Because the AWTs is supposed to eliminate wall interference by shaping the upper and lower walls to approximate free-air streamlines, the farfield boundaries were set to model flight of the wing in free air using a characteristics-based inflow/outflow boundary condition. These boundaries form a rectangular box around the wing model, extending approximately 23 times the maximum length of the model forward, aft, up, down, and spanwise from the wing model origin.

Figure 3.28: Schematic of CFD domain for high-speed validation (not to scale—wing is enlarged for clarity). Symmetry plane boundary condition (orange) was used to approximate the wind tunnel wall with boundary layer removed. The wing (blue) had no-slip boundary condition applied.

A full-viscous unstructured grid was generated with VGrid 3.9. The grid contained an approximate total of 8.8 million cells, with eight cells across the thickness of the wing trailing edge along the entire span (see Figures 3.29, 3.30, and 3.31). Regarding viscous grid generation, δ_1 was set to a distance of $y^+ = 2$ at the midpoint of the wing mean aerodynamic chord at a freestream Reynolds number of 14.1 million. The growth rate a was set to generate 30 cells in the boundary layer at this condition with $b = 0.02$.

Figure 3.29: Top view of high-speed wing grid.

Figure 3.30: Leading-edge detail on high-speed wing grid; tip on left, root on right.

Figure 3.31: Trailing-edge detail on high-speed wing grid; root on left, tip on right.

An additional grid was built with refined cells for assessing discretization error. This grid was generated with VGrid 4.1 and employed volume sources. Boundary layer grid generation parameters were the same as the those for the first grid. Tables 3.6 and 3.7 list information for the two grids used in this validation. The surface mesh for the refined grid contained over three times the number of faces of the initial grid, while the total number of cells increased by a factor of five.

TABLE 3.6

GENERAL GRID INFORMATION FOR NASA TM-4227 CASES

Grid ID	VGrid Version	Boundary Faces	Total Cells
vgrid.1	3.9	101,184	8,815,658
vgrid.2	4.1	353,430	43,773,977

TABLE 3.7

BOUNDARY LAYER PARAMETERS FOR NASA TM-4227 GRIDS

Grid ID	Target $Re \times 10^{-6}$	First Point y^+	δ_1	a	b	Target No. BL Points
vgrid.1	14.1	2.0	$1.51e-5$	0.134	0.02	30
vgrid.2	14.1	2.0	$1.51e-5$	0.134	0.02	30

Pressure coefficient plots from results on the refined grid for selected angles of attack are included in Appendix C. There is no appreciable change in the pressure coefficients between the first grid and the refined grid at any of the conditions analyzed for any of the three spanwise wing stations. Thus, barring boundary layer grid parameter effects, the CFD results presented here are “grid-converged” for pressure coefficient.

3.2.3 Results

Flow solutions for comparison to experimental data were obtained in USM3D at a nominal Mach number of 0.90 and a Reynolds number of 14.1 million, based on the mean aerodynamic chord of 3.755 in., at angles of attack of approximately -4° to 10° . A direct comparison of pressure coefficient between the USM3D solution and reported experimental data at one sample spanwise station and one sample AOA is shown in Figure 3.32. While the slopes of the CFD and experimental curves seem to match well, there is an apparent offset in reference pressure. This offset is typical of the lowest AOAs and is reduced as AOA increases. Looking at the reported wind tunnel wall pressures, there is significant scatter and a general positive shift compared to the CFD solution (Figure 3.33). The CFD pressure coefficients in Figure 3.33 are those in the flowfield at the spatial coordinates of the tunnel wall pressure taps at zero wall deflection. The deflected wall positions were not published, so exact comparisons at those points cannot be made. The effect of the wing on the pressure coefficient at those points in the CFD solution can be seen around tunnel x -location of zero, but as expected, the pressure returns to freestream at the fore- and aft-most extents of the sample range. The most significant consequence of the data shown in Figure 3.33 is that at the locations where the CFD solution is showing the pressure to be close to freestream, the TM-4227 pressure coefficients are still largely positive. This observation and the apparent shift in wing-surface pressures presented earlier lead to the conclusion that the TM-4227 wing pressure data requires a reference pressure correction in order to make useful comparisons with CFD.

Figure 3.32: Wing-surface pressures from TM-4227 experiment and CFD solution.

Figure 3.33: Wind tunnel wall pressures from TM-4227 experiment and corresponding CFD flowfield pressures.

Appendix C.3 contains plots of TM-4227 wind tunnel wall centerline pressures along with CFD pressures taken at zero wall deflection. Comparing the wind tunnel wall pressures with the wing pressure coefficient comparison plots in Appendix C.1, there is an unquestionable correlation between published tunnel centerline pressure magnitude and wing pressure coefficient offset between CFD and published TM-4227 data.

Thus, a delta pressure coefficient was calculated for each onset condition by subtracting the average of all the TM-4227 wall pressure coefficients from the average of all the corresponding non-deflected wall CFD pressure coefficients at that condition. This delta pressure coefficient was then added to all TM-4227 pressure coefficients at the corresponding onset condition to obtain corrected pressure coefficients on the wing. Table 3.8 lists the delta pressure coefficient applied to the TM-4227 data at each onset condition. Figure 3.34 shows the sample wing pressure comparison with the corrected wind tunnel pressure coefficients. Appendix C.2 contains corrected pressure coefficient comparisons for each onset condition.

TABLE 3.8
PRESSURE COEFFICIENT CORRECTION TO NASA TM-4227 DATA

Condition	ΔC_p
Mach 0.877, Re 14.1, AOA -3.92	-0.0525
Mach 0.874, Re 14.1, AOA -1.80	-0.0589
Mach 0.876, Re 14.1, AOA +0.28	-0.0622
Mach 0.890, Re 14.1, AOA +2.24	-0.0229
Mach 0.886, Re 14.1, AOA +5.46	-0.0283
Mach 0.892, Re 14.1, AOA +8.55	-0.0085
Mach 0.907, Re 14.1, AOA +10.31	+0.0010

Figure 3.34: Wing-surface pressures from TM-4227 experiment (corrected) and CFD solution.

Referring to the pressure coefficient comparison plots in Appendix C.2, the CFD solutions compare moderately well to the corrected TM-4227 data. Given that a correction was applied to the NASA data, exact pressure level comparisons are somewhat in question, but several of the conditions were matched closely. In general, this flow exhibited complex shock structures, involving leading-edge shocks and/or multiple shock patterns. Pressures aft of shocks and many shock locations were matched quite well. The conditions with negative angles of attack exhibited sharp lower surface shocks at the wing leading edge. Pressure levels at these conditions match well overall, although the CFD is predicting subtly different pressure distribution shapes. Minimum pressures tend to be underpredicted at the mid-range angles of attack, but the shapes of the pressure distributions are similar.

Leading-edge pressure minima were closely matched at the higher angles of attack. The 10° AOA condition is especially interesting. The inboard section exhibits two distinct shockwaves that become less organized at the midboard section. While the CFD solution tends to over-estimate the sharpness of these shocks, the pressure levels match quite well.

The lack of pressure recovery at the outboard section due to the strong tip vortex is also predicted by the CFD calculations.

It appears that although the CFD and published tunnel pressure levels do not compare well at the low angles of attack, overall circulation at the three wing sections across the angle of attack range could be quite close. Model forces were not available, so lift and drag comparisons cannot be accomplished.

There are several potential sources of error in these comparisons, many of which are impossible to quantify without information beyond what is published. The RANS simulation performed here was fully turbulent, whereas the experiment was performed with free transition. Given the high Reynolds number, this may not be very significant: assuming transition will occur at $Re_x = 500,000$, only 3.5 percent of the chord will be laminar. For the most part, pressures toward the wing leading edge match fairly well.

This experiment involved a three-dimensional wing model in a wind tunnel test section designed for “two-dimensional” airfoil testing. The adaptive wall strategy employed is documented in Rebstock and Lee [35]. The conclusions drawn there are that the two-dimensional approach “improves correctability” when testing three-dimensional models. This means that the chordwise wall interferences are reduced, thus reducing induced wing camber, as would be expected in a two-dimensional test, and allowing for more accurate angle of attack and Mach number corrections. However, the spanwise variation in wall interference remains because there is no mechanism to treat such interference in this type of test section.

Due to the extreme temperature and pressure ranges that the TCT operates within, pressure coefficients were corrected for real gas effects of nitrogen before being published in Chu and Lawing [30]. This should be a simple procedure and is not expected to be a significant source of error.

Overall, this validation is considered adequate for the purposes of this work. Pressure levels and shock locations are both predicted quite well. There is concern regarding the pressure shifts seen at low angles of attack. However, a simple, justifiable correction based

on published wall centerline pressures was applied, which aligns the CFD data with data from the experiment. Any other plausible sources of experimental error are not readily quantifiable.

CHAPTER 4

RESULTS AND DISCUSSION

The CFD methodology presented in Chapter 2 and validated in Chapter 3 was used to explore how a leading-edge strake might be used to reduce aircraft drag. First, a grid convergence analysis was conducted on the Rood wing geometry at a flight condition more applicable to modern business jet or civil transport aircraft. Next, a parametric study was conducted to understand how strake length and wing sweep affect computed drag. Finally, results from the parametric study were used to size a strake for a modern civil transport aircraft.

4.1 Grid Convergence Analysis

Grid convergence was explored using the same Rood wing configuration that was used in the validation. Because obtaining “absolute” or *continuum* drag is still elusive for modern-day RANS CFD methodology [25], a drag delta was used as the convergence metric. The Rood wing with the addition of the Devenport strake served as the second geometry for calculating the drag delta.

The solutions generated for this grid convergence analysis were obtained for a moderate Mach number of 0.50 and a Reynolds number of 20 million. Mach 0.50 was chosen to avoid the addition of wave drag that would obfuscate the results of the parametric study, which includes sweep effects. The high Reynolds number of 20 million reflects a more realistic flight condition for a large business jet or civil transport aircraft.

4.1.1 Grid Generation Parameters

VGrid 4.1 was used to generate all grids presented here. Grids of three levels of refinement were used to assess grid convergence. Table 4.1 lists the target isotropic grid-point spacings applied to the junction region. The volume source point spacings were set to the values in the “Junction” column in the range $-0.69 < \frac{x}{c} < 1.97$, then varied linearly to the value in the “Final Wake” column at $\frac{x}{c} = 3.61$. This is the volume source layout that was used for grid vgrid.6 in Chapter 3, except the forward limit was moved to include

the Devenport strake (see Figure 2.1). The “TE Cells” column refers to the number of boundary faces placed across the thickness of the wing trailing edge. Table 4.2 lists the growth parameters of the viscous layers for each of the three grids.

TABLE 4.1

TARGET GRID POINT SPACINGS FOR INVISCID, ISOTROPIC GRID IN JUNCTION REGION

Refinement Level	Grid Spacing ($\%c_{root}$)		TE Cells
	Junction	Final Wake	
Coarse	1.48	1.97	8
Medium	0.98	2.62	8
Fine	0.66	3.93	16

TABLE 4.2

GRID POINT SPACING PARAMETERS FOR VISCOUS LAYERS GROWTH ON ROOD WING MODELS

Refinement Level	Target $Re \times 10^{-6}$	First Point y^+	δ_1	a	b	Target No. BL Points
Coarse	20.0	2.0	0.000896	0.15	0.02	29
Medium	20.0	1.5	0.000672	0.15	0.02	30
Fine	20.0	1.0	0.000448	0.09	0.02	40

These three sets of grid generation parameters were used to generate grids on both the basic Rood wing geometry and the Rood wing with the Devenport strake. The final grid sizes are listed in Table 4.3.

TABLE 4.3

GENERAL GRID INFORMATION FOR ROOD WING GRID CONVERGENCE ANALYSIS

Refinement Level	Baseline		With Strake	
	Boundary Faces	Total Cells	Boundary Faces	Total Cells
Coarse	105,776	9,412,506	105,646	9,387,024
Medium	147,754	15,678,458	147,348	15,628,352
Fine	226,258	33,415,118	226,826	33,437,314

4.1.2 Devenport Strake Definition

The Devenport strake is defined graphically with cross-sections in Devenport et al. [21]; a side view is shown in Figure 1.8. The surface geometry for the strake was generated by fitting conic sections to the cross-sections and then using those curves to generate a surface in CATIA v5.

4.1.3 Grid Convergence Results

Figure 4.1 shows the resulting drag coefficient for the two configurations for each of the three refinement levels. The boundary conditions used in this study were set such that the viscous wall representing the wind tunnel floor is adjacent to the farfield characteristics-based inflow/outflow boundary condition, resulting in spurious skin friction values at those boundary faces. Therefore, the drag coefficients presented were calculated on a subset of the simulated viscous wall, defined in the title of the plot. The expected trend of reduced overall C_D with increasing cell count is observed [25]. Table 4.4 lists the lift and drag deltas between the two configurations for the three refinement levels; C_L does not change any appreciable amount, and C_D is constant within one count. These results indicate that, while the overall CFD computed drag for a given configuration does change with grid refinement, the drag difference between two similar configurations can be reliably predicted to a constant value.

Figure 4.1: Effect of grid refinement on computed drag coefficient (N is number of cells in grid).

TABLE 4.4

LIFT AND DRAG DELTAS FOR ROOD WING GRID REFINEMENT STUDY

Refinement Level	$\Delta = \text{Baseline} - \text{w/Strake}$	
	ΔC_L	ΔC_D
Coarse	0.00039	0.00203
Medium	0.00030	0.00196
Fine	0.00026	0.00196

4.2 Parametric Study

At the wing root, the published Devenport strake extends 0.51 wing chords forward of the baseline Rood wing leading edge. This much extension may prove prohibitive on a conventional aircraft configuration due to weight or interference with other aircraft systems or components. Also, the Rood wing is not tapered or swept, so the intersection of the wing

leading edge with the wall is 90° in the absence of a strake. A transonic aircraft is certain to have a wing with a swept leading edge. Due to the existence of spanwise flow, the adverse pressure gradient at the junction of a swept wing and wall is not as strong as that for a straight wing. Therefore, the drag reduction due to a strake on a swept wing may be less than that for a straight wing. For these reasons, a parametric study was conducted to learn the effect of strake length and wing sweep on the drag reduction due to a leading-edge strake.

4.2.1 Geometries

The scaled strake geometries were obtained by scaling the Devenport strake chordwise based on the distance from the leading edge of the wing root section to the leading edge of the strake root section. The wing and strake were swept by shearing the defining sections chordwise the appropriate distance. The intersection of the strake with the wing remained the same in all cases. Table 4.5 summarizes the combinations of geometric changes explored. Figure 4.2 shows the strake shapes on the baseline Rood wing. The swept geometries are shown in Figure 4.3.

TABLE 4.5
COMPLETE LIST OF PARAMETER COMBINATIONS
USED IN PARAMETRIC STUDY

Wing Sweep (deg)	Strake Length	
	vs. Devenport	vs. Wing Chord
0	none	none
0	100%	0.51 <i>c</i>
0	50%	0.26 <i>c</i>
0	25%	0.13 <i>c</i>
30	none	none
30	100%	0.51 <i>c</i>
30	50%	0.26 <i>c</i>

Figure 4.2: Shaded surface grids for strake geometries on Rood wing: baseline Devenport strake (blue), 50% Devenport strake (white), 25% Devenport strake (red).

Figure 4.3: Shaded surface grids for strake geometries on 30° swept Rood wing: 100% Devenport strake (blue), 50% Devenport strake (white).

Figure 4.3: Shaded surface grids for strake geometries on 30° swept Rood wing: 100% Devenport strake (blue), 50% Devenport strake (white) (continued).

4.2.2 Grids and Flight Conditions

Grids were generated for each geometry with the fine grid parameters described in Section 4.1.1. The USM3D solver was used to generate drag polars on each grid at Mach 0.50 and a Reynolds number of 20 million.

4.2.3 Results

The resulting polars for the zero-sweep and 30°-sweep geometries are plotted in Figures 4.4 and 4.5, respectively. To be clear, the drag is plotted as $C_D - \frac{C_L^2}{\pi AR}$ simply to reduce the range of the data and thus increase resolution for comparison purposes; the difference in drag at any C_L is not changed by this quadratic shift. Solutions for the 25% strake were obtained for only two AOAs because a clear trend is established with the data presented.

Figure 4.6 shows ΔC_D versus AOA for all geometries. The deltas for the zero-sweep geometries are relative to the zero-sweep Rood wing without strake. Likewise, the deltas for the 30° swept geometries are relative to the 30° swept Rood wing without strake. Changes in C_L for all cases are on the order of the symbol size.

Figure 4.4: Effects of strake length on Rood wing drag polar.

Figure 4.5: Effects of strake length on 30° swept Rood wing drag polar.

Figure 4.6: Drag deltas from parametric changes to Rood wing.

For both the zero-sweep and swept wings, the drag reduction increases monotonically with strake length. However, the zero-sweep wing exhibits a fairly constant (± 1 count) ΔC_D for a given strake size at any AOA, while ΔC_D for the swept wing with a given strake size increases with AOA. This is most likely due to the changing angle between the swept wing and the wall as AOA increases from zero. The angle between the swept wing and wall is maximum for the stagnation streamline at zero AOA and decreases as AOA increases, while the same angle is 90° at any AOA on a straight wing.

Figures 4.7 and 4.8 show U contours in Plane 12 for all geometries at zero and 6° AOA, respectively. The boundary layer distortion seen in these plots compares well with the wake data from experiment in Figure 1.9 for Rood wings with and without the Devenport strake. Coupled with Figure 4.6, the correlation of decreasing wake distortion with decreasing drag is clearly seen. Comparing Figure 4.7 parts *a* and *e*, the 30° swept wing without strake shows less momentum loss than the straight wing without strake, as expected.

Figure 4.7: Contours of U in wake plane $x/c = 3.0$ from USM3D at Mach 0.50, Re 20 million, zero AOA: (a) straight Rood wing, (b) straight Rood wing with 25% Devenport strake, (c) straight Rood wing with 50% Devenport strake, (d) straight Rood wing with full size Devenport strake, (e) 30° swept Rood wing, (f) 30° swept Rood wing with 50% Devenport strake, (g) 30° swept Rood wing with full size Devenport strake.

Figure 4.8: Contours of U in wake plane $x/c = 3.0$ from USM3D at Mach 0.50, Re 20 million, 6° AOA: (a) straight Rood wing, (b) straight Rood wing with 25% Devenport strake, (c) straight Rood wing with 50% Devenport strake, (d) straight Rood wing with full size Devenport strake, (e) 30° swept Rood wing, (f) 30° swept Rood wing with 50% Devenport strake, (g) 30° swept Rood wing with full size Devenport strake.

4.3 Aircraft Application

4.3.1 Strake Sizing

To achieve the goal of reducing horseshoe vortex drag on a realistic aircraft configuration, the results from the parametric CFD study of the Rood wing were applied to sizing a strake for the NASA Common Research Model (CRM) wing-body configuration. The CRM is a publicly available geometry of a modern wide-body commercial transport aircraft by Vassberg et al. [36]. It was chosen for this work due to its relevance in both its modern, supercritical wing design and its inclusion in recent AIAA Drag Prediction Workshops [25]. Reference quantities for the CRM are provided in Table 4.6, and the wing planform is shown in Figure 4.9.

TABLE 4.6
REFERENCE QUANTITIES FOR NASA COMMON
RESEARCH MODEL

NASA CRM		
S_{ref}	594,720.0	in^2
c_{ref}	275.80	in
b_{ref}	2,313.50	in
x_{ref}	1,325.90	in
y_{ref}	468.75	in
z_{ref}	177.95	in
λ	0.275	
AR	9.0	
$\Lambda_{c/4}$	35°	

CRM Planforms: Wing & Horizontal

Figure 4.9: CRM wing and horizontal stabilizer planforms (taken from [37]).

Referring back to Figures 4.2 and 4.3, it is evident that the aspect ratio of the Rood wing model is not representative of a modern transport aircraft wing. Therefore, to understand how the drag deltas from the Rood wing study might apply to the CRM, the disparity in the ratio of strake size to total wing size must be accounted for. The wing aspect ratio AR is a candidate for a scaling parameter, but AR does not take into account the taper of the wing. Given that the junction drag is confined to the region around the wing root, a non-dimensional parameter χ that is similar to AR but includes the length of the wing root chord is suggested:

$$\chi = \frac{b_{ref}}{c_{root}} \quad (4.1)$$

For a wing with $\lambda = 1$, $\chi \equiv AR$.

Let Φ be a factor to be applied to ΔC_D from the Rood wing study:

$$\Phi = \frac{\chi^{Rood}}{\chi^{CRM}} \quad (4.2)$$

Then the scaled drag coefficient that estimates the effect of the strake on the CRM configuration is

$$\Delta C_D^{scaled} = \Phi \times \Delta C_D^{Rood} \quad (4.3)$$

Applying equation (4.3) to the drag deltas plotted in Figure 4.6 results in scaled drag deltas, which are presented in Figure 4.10. Examination of the CRM planform in Figure 4.9 yields a leading-edge sweep angle of approximately 37° . Data from DPW-4 [25] suggests that the design C_L occurs around 2.3° AOA for the CRM wing-body. Referring to Figure 4.10, the two strakes on the 30° swept wing yield $\Delta C_D^{scaled} \sim -4.5$ counts at that AOA, with less than one count distinguishing the $0.51c_{root}$ strake from the $0.26c_{root}$ strake.

Figure 4.10: Drag deltas from parametric changes to Rood wing, scaled according to equation (4.3).

Table 4.7 lists bluntness factors for the Rood airfoil, the CRM root section, and two NACA airfoils. Recalling from Section 1.1.2 that higher BF values have been correlated with stronger leading-edge vortices [1], the drag due to the horseshoe vortex on the CRM is probably lower than that for the Rood wing. Thus, a strake of length $0.26c_{root}$ was chosen to be incorporated into the CRM wing-body configuration, with the expectation that not all of the ΔC_D^{scaled} from the Rood wing will be realized.

TABLE 4.7

BLUNTNESS FACTORS FOR SEVERAL AIRFOILS

Airfoil	BF
Rood airfoil	0.5179
NACA 0015	0.0452
NACA 0012	0.0287
CRM root section	0.0281

4.3.2 Geometry

The CRM geometry from Morrison [37] was downloaded in initial graphics exchange specification (IGES) format and then read into CATIA v5 for adding the strake. The Devenport strake was incorporated into the CRM wing-body geometry by first scaling the baseline, non-swept Devenport strake by the ratio of root chord lengths $c_{root}^{CRM}/c_{root}^{Rood}$. Then, the strake length was scaled identically as it was for the parametric study to a length of $0.26c_{root}$. Finally, the strake sections were sheared to match the CRM wing leading-edge sweep angle, as was done in the parametric study. The strake side-of-body section was generated by scaling chordwise the section of the side-of-body airfoil from its maximum ordinate forward on both the upper and lower surfaces. The other defining sections of the strake were conic sections, set tangent to the upper and lower surfaces of the wing. The leading edge radii of the conic strake sections were set to match those of the wing as closely as possible. Once the strake surface was defined, it was extrapolated into the fuselage because the fuselage/wing fairing on the CRM tapers into the fuselage forward of the wing. Figures 4.11 and 4.12 show the $0.26c_{root}$ strake installed on the CRM wing-body geometry.

Figure 4.11: Isometric views of CRM geometry with and without strake (cyan color).

Figure 4.12: Top, side, and bottom views of CRM geometry with strake (cyan color).

4.3.3 Grids

Grids were generated on the CRM wing-body configuration with and without the strake using VGrid 4.1 and the same sourcing strategy that was employed for the parametric Rood wing cases and the medium grid-point spacing parameters from Table 4.1. The NASA LaRC GridTool file for the CRM was downloaded from Morrison [37] and used as a starting point. Grids for the wind tunnel Reynolds number of five million and design flight Reynolds number of 40 million were generated for both configurations. Both grids were a half wing-body model on a symmetry plane. Figure 4.13 shows the final surface grid in the junction region for the configuration with and without the strake. Table 4.8 lists the growth parameters for the viscous layers for both Reynolds numbers, and Table 4.9 lists the sizes of the final grids.

Figure 4.13: Isometric views of junction region of CRM grids with and without strake.

TABLE 4.8

GRID POINT SPACING PARAMETERS FOR GROWTH OF
VISCOUS LAYERS ON CRM MODELS

Refinement Level	Target $Re \times 10^{-6}$	First Point y^+	δ_1	a	b	Target No. BL Points
Medium	5.0	1.5	0.002115	0.122	0.02	30
Medium	40.0	1.5	0.0003255	0.145	0.02	32

TABLE 4.9

GENERAL GRID INFORMATION FOR CRM WING-BODY MODELS

Reynolds Number	Baseline		With Strake	
	Boundary Faces	Total Cells	Boundary Faces	Total Cells
5.0×10^6	183,304	16,142,818	183,296	16,126,311
40.0×10^6	185,184	16,954,478	185,182	16,932,437

4.3.4 Results

Angle of attack sweeps were completed with USM3D at Mach 0.50 and Mach 0.85 at the wind tunnel Reynolds number of five million. An AOA sweep was also made at Mach 0.85 and the flight Reynolds number of 40 million. The resulting lift curves and drag polars are plotted in Figures 4.14 and 4.15.

The Mach 0.50 results show a constant decrease in drag of about two counts with the strake. At Mach 0.85 at both Reynolds numbers, the drag decrease due to the strake is again two counts at low lift coefficients but decreases to about one count at the design C_L of 0.50.

Figure 4.14: Lift and drag characteristics of CRM wing-body from AOA sweeps at wind tunnel Reynolds number.

Figure 4.15: Lift and drag characteristics of CRM wing-body from AOA sweeps at flight Reynolds number.

Figures 4.16, 4.17, and 4.18 show U contours in streamwise planes at $x/c = 0.64$, $x/c = 1.5$, and $x/c = 3.0$, respectively, for solutions at the Re 40 million design point, with and without the strake. Evidence of the horseshoe vortex structure is seen in the solution without strake in planes at $x/c = 0.64$ and $x/c = 1.5$, with the tongue of low-momentum fluid clearly visible on the lower fuselage. The upper horseshoe vortex leg does not create as much boundary layer distortion. The solution with the strake shows that the strake was effective at removing the horseshoe vortex and its associated boundary layer distortion. In the wake plane at $x/c = 3.0$, the horseshoe vortex structure is no longer discernable on the solution without the strake, but the U isobars are still more distorted compared to the solution with the strake.

Figure 4.16: Contours of U in plane at $x/c = 0.64$ from USM3D for CRM at $M_\infty = 0.85$, $C_L = 0.50$, Re 40 million, without (left) and with (right) $0.26c_{root}$ strake.

Figure 4.17: Contours of U in wake plane at $x/c = 1.5$ from USM3D for CRM at $M_\infty = 0.85$, $C_L = 0.50$, Re 40 million, without (left) and with (right) $0.26c_{root}$ strake.

Figure 4.18: Contours of U in wake plane at $x/c = 3.0$ from USM3D for CRM at $M_\infty = 0.85$, $C_L = 0.50$, Re 40 million, without (left) and with (right) $0.26c_{root}$ strake.

Contours of upper wing surface C_p from USM3D at the design point are plotted in Figure 4.19 for both configurations, along with an OVERFLOW result from Vassberg et al. [36]. Agreement in the shock locations and general structure of the pressure isobars can be seen for the baseline CRM. The strake disrupts the pressure contours in its immediate vicinity, locally increasing the pressure compared to the baseline wing, but the rest of the wing pressures are mostly unchanged.

Figure 4.19: Pressure coefficient contours on upper surface of CRM wing at $M_\infty = 0.85$, $C_L = 0.50$, Re 40 million: (a) OVERFLOW solution, taken from Vassberg et al. [36], and (b) USM3D solutions with and without the $0.26c_{root}$ strake.

Referring back to Figure 4.10, if the CRM application of the strake were to perform like the Rood wing with a strake, the estimated drag decrease due to a $0.26c_{root}$ strake would be approximately four counts. The CRM results indicate a two count drag reduction at subsonic conditions and a one count drag reduction at the transonic design point with a strake of length $0.26c_{root}$. This difference in drag reduction between the estimated value and the computed value could be due to the large discrepancy in BF on the CRM compared to the Rood wing, although optimization of the strake shape for integration with the CRM may improve its performance. Considering that no aerodynamic optimization was performed on the strake, these results are encouraging. They suggest that the results from the Rood wing study are applicable to a real aircraft problem and might be useful for other applications.

CHAPTER 5

CONCLUSIONS

5.1 Summary

An application of steady RANS CFD to the problem of wing-body junction drag has been presented. A systematic process was followed to increase confidence in the final application of a CFD methodology to horseshoe vortex drag reduction on a transonic civil transport aircraft. The process began with validating the CFD methodology for low-speed junction flow on a simple model: the Rood wing on a flat plate. Then the same CFD methodology was validated against data published by NASA for a transonic wing. A parametric study on the Rood wing model was completed, resulting in drag deltas due to a leading-edge strake of various sizes on wings of two different sweep angles. Finally, the results from the parametric study were applied to the sizing of a leading-edge strake for the NASA Common Research Model, a transonic commercial transport aircraft.

The validation of CFD against experimental data for the low-speed Rood wing junction flow yielded excellent qualitative agreement on field velocities and horseshoe vortex location. Various methods for defining wind tunnel boundary conditions were explored; the characteristics-based inflow/outflow boundary condition proved to be the most useful for matching experimental data. The CFD underpredicted core U values in the low-momentum tongue, and this disagreement actually worsened with grid refinement. Field values for W agreed well and did not exhibit grid size dependency. Peak minimum surface pressures at the airfoil maximum thickness were overpredicted but matched at other locations.

Validation against experimental data for the high Reynolds number, transonic flow yielded acceptable surface pressure comparisons. Shock locations and pressure recovery were predicted satisfactorily. A correction to the published experimental data was necessary to achieve this agreement. The CFD solutions were shown to be grid-converged for surface pressures.

In support of the parametric study, grid convergence was demonstrated to within less than one drag count for ΔC_D between the Rood wing with and without a leading-edge strake, across three levels of grid size. The expected trend of reduced drag with increasing refinement was observed for both configurations.

The computed drag reduction due to the leading-edge strake on the CRM was on the order of what was expected based on the parametric study. It was expected that the CRM would not benefit as much from the addition of the strake as the Rood wing did due to the airfoil shape, and the results appear to confirm this. This conclusion is based on an estimated scaled drag delta using wing planform parameters. The CFD results also showed that the drag reduction due to the strake was independent of Reynolds number at $M_\infty = 0.85$.

The low-momentum tongue of fluid that exists in conjunction with the horseshoe vortex around the Rood wing model was also seen in CFD results on the CRM model, and the strake was effective at removing this and other boundary layer distortion. This result is consistent with data from the CFD calculations on the Rood wing and the published experimental data for the Rood wing with and without strake.

Admittedly, RANS CFD does not capture the exact physics known to exist in the junction horseshoe vortex. The Reynolds stress tensor is known from experimental measurements to be anisotropic [8], and RANS CFD employs a single eddy viscosity for the entire tensor. However, qualitative and quantitative agreement has been presented here between RANS CFD and experimental data for a low-speed junction flow, and the methodology has proven to give consistent results throughout the research presented in this work. Given the underprediction of U by CFD in the low-momentum tongue for the Rood validation case, the calculated drag reduction due to the strake may be overpredicted by this CFD methodology. Nevertheless, the trends shown in the parametric study are reasonable, and the methodology presented in this work was proven useful for leading-edge strake design studies.

5.2 Future Research

Future work could include expanding the parametric study to wings of different airfoil sections and sweep angles to explore effects on computed drag. In particular, it is suspected that the Rood airfoil's high BF contributes to abnormally high horseshoe vortex drag; repeating the parametric strake study with a different airfoil might prove useful.

Other methods for turbulence modeling, such as DES, DDES, or LES could be employed and compared with results presented here. Obviously, more experimental data is always useful for validating CFD, so experiments might be carried out with the goal of obtaining drag data for junction flow treatments such as strakes. Also, a strake such as the one presented here could be a starting point for an aerodynamic optimization for an aircraft application.

A full aerodynamics project that would hope to incorporate a leading-edge strake on a wing would also need to show the strake's compatibility with areas other than cruise drag, such as high lift performance and ice protection.

REFERENCES

REFERENCES

- [1] Simpson, R. L., “Junction Flows,” *Annual Review of Fluid Mechanics*, Vol. 33, 2001, pp. 415–443.
- [2] Fleming, J. L., Simpson, R. L., Cowling, J. E., and Devenport, W. J., “An Experimental Study of a Turbulent Wing-Body Junction and Wake Flow,” *Experiments in Fluids*, Vol. 14, 1993, pp. 366–378.
- [3] Devenport, W. J., and Simpson, R. L., “Time-Dependent and Time-Averaged Turbulence Structure Near the Nose of a Wing-Body Junction,” *Journal of Fluid Mechanics*, Vol. 210, 1990, pp. 23–55.
- [4] Ahmed, A., and Khan, M. J., “Effect of Sweep on Wing-Body Junction Flow,” AIAA Paper 95-0868, 33rd Aerospace Sciences Meeting and Exhibit, Reno, NV, Jan. 9–12, 1995.
- [5] Rudnik, R., Sitzmann, M., Godard, J., and Lebrun, F., “Experimental Investigation of the Wing-Body Junction Flow on the DLR-F6 Configuration in the ONERA S2MA Facility,” AIAA Paper 2009-4113, 27th Applied Aerodynamics Conference, San Antonio, TX, June 22–25, 2009.
- [6] Kim, S., Walker, D. A., and Simpson, R. L., “Observation and Measurements of Flow Structures in the Stagnation Region of a Wing-Body Junction,” Report# VPI-E-91-20, Virginia Tech, Engineering Science and Mechanics, Blacksburg, VA, Sept. 1, 1991.
- [7] Ölçmen, M. S., and Simpson, R. L., “Influence of Wing Shapes on Surface Pressure Fluctuations at Wing-Body Junctions,” *AIAA Journal*, Vol. 32, No. 1, Jan. 1994, pp. 6–15.
- [8] Ölçmen, M. S., and Simpson, R. L., “Experimental Transport-Rate Budgets in Complex 3-D Turbulent Flow Near a Wing/Body Junction,” AIAA Paper 96-2035, 27th AIAA Fluid Dynamics Conference, New Orleans, LA, June 17–20, 1996.
- [9] Wei, Q. D., Wang, J. M., Chen, G., Lu, Z. B., and Bi, W. T., “Modification of Junction Flows by Altering the Section Shapes of the Cylinders,” *Journal of Visualization*, Vol. 11, No. 2, 2008, pp. 115–124.
- [10] Gand, F., Brunet, V., and Deck, S., “A Combined Experimental, RANS and LES Investigation of a Wing Body Junction Flow,” AIAA Paper 2010-4753, 40th Fluid Dynamics Conference and Exhibit, Chicago, IL, June 28–July 1, 2010.

REFERENCES (continued)

- [11] Devenport, W. J., Agarwal, N. K., Dewitz, M. B., Simpson, R. L., and Poddar, K., “Effects of a Fillet on the Flow Past a Wing-Body Junction,” *AIAA Journal*, Vol. 28, No. 12, 1990, pp. 2017–2024.
- [12] Barber, T. J., “An Investigation of Strut-Wall Intersection Losses,” *Journal of Aircraft*, Vol. 15, No. 10, 1978, pp. 676–681.
- [13] Oudheusden, B. W. van, Steenaert, C. B., and Boermans, L. M. M., “Attachment-Line Approach for Design of a Wing-Body Leading-Edge Fairing,” *Journal of Aircraft*, Vol. 41, No. 2, 2004, pp. 238–246.
- [14] Devenport, W. J., and Simpson, R. L., “Flow Past a Wing-Body Junction—Experimental Evaluation of Turbulence Models,” *AIAA Journal*, Vol. 30, No. 4, 1992, pp. 873–881.
- [15] Ölçmen, M. S., and Simpson, R. L., “Experimental Evaluation of Turbulent Diffusion Models in Complex 3-D Flow Near a Wing/Body Junction,” AIAA Paper 97-0650, 35th Aerospace Sciences Meeting and Exhibit, Reno, NV, Jan. 6–9, 1997.
- [16] Kubendran, L. R., Sung, C.-H., and Yang, C.-I., “Experiments and Code Validation for Juncture Flows,” *AGARD, Validation of Computational Fluid Dynamics*, Vol. 2, 1988,
- [17] Paciorri, R., Bonfiglioli, A., Di Mascio, A., and Favini, B., “RANS Simulations of a Junction Flow,” *International Journal of Computational Fluid Dynamics*, Vol. 19, No. 2, 2005, pp. 179–189.
- [18] *Case 8: Wing/Body Junction with Separation; Experiments by Devenport, Simpson and Fleming*, European Research Community on Flow, Turbulence and Combustion (ERCOFTAC) Classic Database, URL: <http://cfd.mace.manchester.ac.uk/ercoftac> [cited 10/2009].
- [19] Fu, S., Xiao, Z., Chen, H., Zhang, Y., and Huang, J., “Simulation of Wing-Body Junction Flows with Hybrid RANS/LES Methods,” *International Journal of Heat and Fluid Flow*, Vol. 28, No. 6, 2007, pp. 1379–1390.
- [20] Ölçmen, M. S., and Simpson, R. L., “Influence of Passive Flow-Control Devices on the Pressure Fluctuations at Wing-Body Junction Flows,” *Journal of Fluids Engineering*, Vol. 129, 2007, pp. 1030–1037.
- [21] Devenport, W. J., Simpson, R. L., Dewitz, M. B., and Agarwal, N. K., “Effects of a Leading-Edge Fillet on the Flow Past an Appendage-Body Junction,” *AIAA Journal*, Vol. 30, No. 9, 1992, pp. 2177–2183.

REFERENCES (continued)

- [22] Vassberg, J. C., Sclafani, A. J., and DeHaan, M. A., “A Wing-Body Fairing Design for the DLR-F6 Model: A DPW-III Case Study,” AIAA Paper 2005-4730, 23rd Applied Aerodynamics Conference, Toronto, Ontario Canada, June 6–9, 2005.
- [23] Milstein, J., *AirCRAFT, The Jet as Art*, New York: Harry N Abrams, Inc., May 1, 2007.
- [24] Green, B. E., and Whitesides, J. L., “Method for Designing Leading-Edge Fillets to Eliminate Flow Separation,” *Journal of Aircraft*, Vol. 40, No. 2, 2003, pp. 282–289.
- [25] Vassberg, J. C., Tinoco, E. N., Mani, M., Rider, B., Zickuhr, T., Levy, D. W., Brodersen, O. P., Eisfeld, B., Crippa, S., Wahls, R. A., Morrison, J. H., Mavriplis, D. J., and Murayama, M., “Summary of the Fourth AIAA CFD Drag Prediction Workshop,” AIAA Paper 2010-4547, 28th Applied Aerodynamics Conference, Chicago, IL, June 28–July 1, 2010.
- [26] Frink, N. T., “Tetrahedral Unstructured Navier-Stokes Method for Turbulent Flows,” *AIAA Journal*, Vol. 36, No. 11, 1998, pp. 1975–1982.
- [27] Frink, N. T., *USM3Dns Online Manual*, Oct. 17, 2006, URL: <http://tetruss.larc.nasa.gov/usm3d/> [cited 02/2010].
- [28] Pirzadeh, S. Z., “Structured Background Grids for Generation of Unstructured Grids by Advancing-Front Method,” *AIAA Journal*, Vol. 31, No. 2, Feb. 1993.
- [29] Pirzadeh, S. Z., “Advanced Unstructured Grid Generation for Complex Aerodynamic Applications,” *AIAA Journal*, Vol. 48, No. 5, May 2010.
- [30] Chu, J., and Lawing, P. L., “High-Reynolds-Number Test of a 5-Percent-Thick Low-Aspect-Ratio Semispan Wing in the Langley 0.3-Meter Transonic Cryogenic Tunnel: Wing Pressure Distributions,” NASA TM-4227, NASA Langley Research Center, Hampton, VA, Dec. 1990.
- [31] Blazek, J., *Computational Fluid Dynamics: Principles and Applications*, 2nd ed., Elsevier, 2005.
- [32] Drela, M., *XFOIL Subsonic Airfoil Development System*, Apr. 7, 2008, URL: <http://web.mit.edu/drela/Public/web/xfoil/> [cited 05/2010].
- [33] Mineck, R. E., “Hardware and Operating Features of the Adaptive Wall Test Section for the Langley 0.3-Meter Transonic Cryogenic Tunnel,” NASA TM-4114, NASA Langley Research Center, Hampton, VA, June 1989.

REFERENCES (continued)

- [34] Abbott, I. H., and Doenhoff, A. E. von, *Theory of Wing Sections*, New York: Dover Publications, Inc., 1959.
- [35] Rebstock, R., and Lee Jr., E. E., “Capabilities of Wind Tunnels with Two Adaptive Walls to Minimize Boundary Interference in 3-D Model Testing,” *NASA CP-3020*, Vol. 1, Part 2, Mar. 1989, pp. 891–910.
- [36] Vassberg, J. C., DeHaan, M. A., Rivers, S. M., and Wahls, R. A., “Development of a Common Research Model for Applied CFD Validation Studies,” AIAA Paper 2008-6919, 2008.
- [37] Morrison, J. H., *4th AIAA CFD Drag Prediction Workshop*, Dec. 3, 2011, URL: <http://aaac.larc.nasa.gov/tsab/cfdlarc/aiaa-dpw/Workshop4/workshop4.html> [cited 06/2012].
- [38] Bali, N. P., and Narayana Iyengar, N. Ch., *Text Book of Engineering Mathematics*, Laxmi Publications, 2004, pp. 233–234.

APPENDICES

APPENDIX A

Non-Dimensional Wall Distances for Tetrahedral Cell Centers

A brief explanation is given here regarding the y^+ values for the centers of cells comprising the prismatic-shaped viscous layers of a fully tetrahedral, unstructured grid.

Figure A.1 presents a diagram for a typical, single layer of boundary layer cells composed of three tetrahedra defined by the sets of points $ABCE$, $ACDE$, and $CDEF$. Let A , B , and C be boundary points. It is assumed that the grid is generated such that vectors $\overrightarrow{AD}$, $\overrightarrow{BE}$, and $\overrightarrow{CF}$ are normal to the local surface and that their lengths are equal:

$$\delta_1 = |\overrightarrow{AD}| = |\overrightarrow{BE}| = |\overrightarrow{CF}|$$

Figure A.1: Representation of three tetrahedra that constitute one prismatic layer of boundary layer cells.

The centroid $\mathbf{c}$ of a tetrahedron is given by the arithmetic mean of the coordinates of its vertices [38]:

$$\mathbf{c} = \frac{1}{4} \sum_{i=1}^4 \mathbf{x}_i$$

where i is the index of the vertex located at point $\mathbf{x}$.

APPENDIX A (continued)

If the local surface curvature is small enough such that

$$\overrightarrow{AD} \parallel \overrightarrow{BE} \parallel \overrightarrow{CF}$$

then the distance from the wall of the centroid for each tetrahedron is as given in Table A.1.

TABLE A.1

CENTROID CALCULATION FOR CELLS ADJACENT TO VISCOUS WALL

Tetrahedron	Wall Distance (δ_1^{-1})				
	Vertex 1	Vertex 2	Vertex 3	Vertex 4	Centroid
<i>ABCE</i>	0	0	0	1	0.25
<i>ACDE</i>	0	0	1	1	0.50
<i>CDEF</i>	0	1	1	1	0.75

The average wall distance for the first prismatic layer of tetrahedron is $0.50\delta_1$ if all tetrahedron are weighted equally. However, the tetrahedral cell on the boundary has its centroid at only $0.25\delta_1$.

APPENDIX B

Field Velocity Plots Comparing CFD to Rood Wing Experiment

Figure B.1: U contours in Plane 5: CFD (top), Experiment [2] (bottom).

APPENDIX B (continued)

Figure B.2: U contours in Plane 6: CFD (top), Experiment [2] (bottom).

APPENDIX B (continued)

USM3D, plane 7, $X/C=0.64$, $X(\text{mm})=195.20$

JLF data, plane 7, $X/C=0.64$, $X(\text{mm})=195.20$

Figure B.3: U contours in Plane 7: CFD (top), Experiment [2] (bottom).

APPENDIX B (continued)

Figure B.4: U contours in Plane 9: CFD (top), Experiment [2] (bottom).

APPENDIX B (continued)

USM3D, plane 10, $X/C=1.05$, $X(\text{mm})=320.25$

JLF data, plane 10, $X/C=1.05$, $X(\text{mm})=320.25$

Figure B.5: U contours in Plane 10: CFD (top), Experiment [2] (bottom).

APPENDIX B (continued)

USM3D, plane 11, $X/C=1.50$, $X(\text{mm})=457.50$

JLF data, plane 11, $X/C=1.50$, $X(\text{mm})=457.50$

Figure B.6: U contours in Plane 11: CFD (top), Experiment [2] (bottom).

APPENDIX B (continued)

JLF data, plane 5, $X/C=0.18$, $X(\text{mm})=54.90$

Figure B.7: W contours in Plane 5: CFD (top), Experiment [2] (bottom).

APPENDIX B (continued)

Figure B.8: W contours in Plane 6: CFD (top), Experiment [2] (bottom).

APPENDIX B (continued)

Figure B.9: W contours in Plane 7: CFD (top), Experiment [2] (bottom).

APPENDIX B (continued)

Figure B.10: W contours in Plane 9: CFD (top), Experiment [2] (bottom).

APPENDIX B (continued)

USM3D, plane 10, $X/C=1.05$, $X(\text{mm})=320.25$

JLF data, plane 10, $X/C=1.05$, $X(\text{mm})=320.25$

Figure B.11: W contours in Plane 10: CFD (top), Experiment [2] (bottom).

APPENDIX B (continued)

USM3D, plane 11, $X/C=1.50$, $X(\text{mm})=457.50$

JLF data, plane 11, $X/C=1.50$, $X(\text{mm})=457.50$

Figure B.12: W contours in Plane 11: CFD (top), Experiment [2] (bottom).

APPENDIX C

Pressure Coefficient Plots Comparing CFD to NASA TM-4227 Data

C.1 CFD vs. Published TM-4227 Wing Data

APPENDIX C (continued)

Mach 0.877, Re 14.1 million, AOA -3.92
Eta = 0.91, Uncorrected

Mach 0.874, Re 14.1 million, AOA -1.80
Eta = 0.28, Uncorrected

APPENDIX C (continued)

Mach 0.874, Re 14.1 million, AOA -1.80
Eta = 0.62, Uncorrected

Mach 0.874, Re 14.1 million, AOA -1.80
Eta = 0.91, Uncorrected

APPENDIX C (continued)

Mach 0.876, Re 14.1 million, AOA +0.28
Eta = 0.28, Uncorrected

Mach 0.876, Re 14.1 million, AOA +0.28
Eta = 0.62, Uncorrected

APPENDIX C (continued)

Mach 0.876, Re 14.1 million, AOA +0.28
Eta = 0.91, Uncorrected

Mach 0.89, Re 14.1 million, AOA +2.24
Eta = 0.28, Uncorrected

APPENDIX C (continued)

Mach 0.89, Re 14.1 million, AOA +2.24
Eta = 0.62, Uncorrected

Mach 0.89, Re 14.1 million, AOA +2.24
Eta = 0.91, Uncorrected

APPENDIX C (continued)

Mach 0.886, Re 14.1 million, AOA +5.46
Eta = 0.28, Uncorrected

Mach 0.886, Re 14.1 million, AOA +5.46
Eta = 0.62, Uncorrected

APPENDIX C (continued)

Mach 0.886, Re 14.1 million, AOA +5.46
Eta = 0.91, Uncorrected

Mach 0.892, Re 14.1 million, AOA +8.55
Eta = 0.28, Uncorrected

APPENDIX C (continued)

Mach 0.892, Re 14.1 million, AOA +8.55
Eta = 0.62, Uncorrected

Mach 0.892, Re 14.1 million, AOA +8.55
Eta = 0.91, Uncorrected

APPENDIX C (continued)

Mach 0.907, Re 14.1 million, AOA +10.31
Eta = 0.28, Uncorrected

Mach 0.907, Re 14.1 million, AOA +10.31
Eta = 0.62, Uncorrected

APPENDIX C (continued)

Mach 0.907, Re 14.1 million, AOA +10.31
Eta = 0.91, Uncorrected

APPENDIX C (continued)

C.2 CFD vs. Corrected TM-4227 Wing Data

APPENDIX C (continued)

Mach 0.877, Re 14.1 million, AOA -3.92
Eta = 0.91, Corrected

Mach 0.874, Re 14.1 million, AOA -1.80
Eta = 0.28, Corrected

APPENDIX C (continued)

Mach 0.874, Re 14.1 million, AOA -1.80
Eta = 0.62, Corrected

Mach 0.874, Re 14.1 million, AOA -1.80
Eta = 0.91, Corrected

APPENDIX C (continued)

Mach 0.876, Re 14.1 million, AOA +0.28
Eta = 0.28, Corrected

Mach 0.876, Re 14.1 million, AOA +0.28
Eta = 0.62, Corrected

APPENDIX C (continued)

Mach 0.876, Re 14.1 million, AOA +0.28
Eta = 0.91, Corrected

Mach 0.89, Re 14.1 million, AOA +2.24
Eta = 0.28, Corrected

APPENDIX C (continued)

Mach 0.89, Re 14.1 million, AOA +2.24
Eta = 0.62, Corrected

Mach 0.89, Re 14.1 million, AOA +2.24
Eta = 0.91, Corrected

APPENDIX C (continued)

Mach 0.886, Re 14.1 million, AOA +5.46
Eta = 0.28, Corrected

Mach 0.886, Re 14.1 million, AOA +5.46
Eta = 0.62, Corrected

APPENDIX C (continued)

Mach 0.886, Re 14.1 million, AOA +5.46
Eta = 0.91, Corrected

Mach 0.892, Re 14.1 million, AOA +8.55
Eta = 0.28, Corrected

APPENDIX C (continued)

Mach 0.892, Re 14.1 million, AOA +8.55
Eta = 0.62, Corrected

Mach 0.892, Re 14.1 million, AOA +8.55
Eta = 0.91, Corrected

APPENDIX C (continued)

Mach 0.907, Re 14.1 million, AOA +10.31
Eta = 0.28, Corrected

Mach 0.907, Re 14.1 million, AOA +10.31
Eta = 0.62, Corrected

APPENDIX C (continued)

Mach 0.907, Re 14.1 million, AOA +10.31
Eta = 0.91, Corrected

APPENDIX C (continued)

C.3 CFD vs. Published TM-4227 Tunnel Wall Data

APPENDIX C (continued)

APPENDIX C (continued)

APPENDIX C (continued)