The Pennsylvania State University
The Graduate School
Department of Aerospace Engineering

REYNOLDS STRESS MODELING OF SEPARATED TURBULENT FLOWS
OVER HELICOPTERS

A Thesis in
Aerospace Engineering

by
Emre Alpman

© 2006 Emre Alpman

Submitted in Partial Fulfillment
of the Requirements
for the Degree of

Doctor of Philosophy

May 2006
The thesis of Emre Alpman was reviewed and approved* by the following:

Lyle N. Long  
Professor of Aerospace Engineering  
Thesis Advisor  
Chair of Committee

Barnes W. McCormick  
Boeing Professor Emeritus of Aerospace Engineering

Joseph F. Horn  
Assistant Professor of Aerospace Engineering

Savas Yavuzkurt  
Professor of Mechanical Engineering

Yousry Y. Azmy  
Professor of Nuclear Engineering

George A. Lesieutre  
Professor of Aerospace Engineering  
Head of the Department of Aerospace Engineering

*Signatures are on file in the Graduate School
ABSTRACT

A numerical investigation of inviscid and viscous flows around three-dimensional complex bodies is made using unstructured meshes. Inviscid flow solutions around an RAH-66 Comanche helicopter fuselage are performed to analyze the aerodynamics of ducted tail rotors in low-power, near-edgewise flow conditions. A numerical solution of the Euler Equations is obtained for the flow over the Comanche fuselage with a uniform actuator disk and blade element models for the FANTAIL™; the main rotor is excluded in this study. The solutions are obtained by running the PUMA2 computational fluid dynamics code with an unstructured grid with 2.8 million tetrahedral cells. PUMA2 is an in-house computer code written in ANSI C++. Excellent correlation between the calculations and a variety of static test data are presented and discussed. The dynamic relationship between the antitorque thrust moment and applied collective pitch angle is studied by changing the pitch angle input by five degrees at a rate of 144 degrees per second. Dynamic fan thrust and moment response to applied collective pitch in hover and forward flight are presented and discussed.

In order to remove the deficiency of the Euler equations in predicting separated flows, which is mostly the case in helicopter fuselage aerodynamics, a concurrent study is performed to simulate turbulent flows around three-dimensional bodies. Most of the turbulence models in the literature contain simplified assumptions which make them computationally cheap but of limited accuracy. Dramatic improvements in the computer processing speed and parallel processing made it possible to use more complete models, such as Reynolds Stress Models, for turbulent flow simulations around complex
geometries, which is the focus of this work. The Reynolds Stress Model consists of coupling Reynolds transport equations with the Favre-Reynolds averaged Navier-Stokes equations, which results in a system of 12 coupled nonlinear partial differential equations. The solutions are obtained by running the PUMA_RSM computational fluid dynamics code on unstructured meshes. Results for high Reynolds number flow around a 6:1 prolate spheroid, a sphere and a Bell 214ST fuselage are presented.

For the prolate spheroid basic flow features such as cross-flow separation are simulated. Predictions of mean pressure and circumferential locations of cross flow separation points are in good agreement with experiment. Most of the separation location predictions are in less than five percent discrepancy with measurements. The effects of the freestream turbulence intensity and turbulent Reynolds number are analyzed and discussed. A grid refinement study is performed to improve the computations. The fine mesh solution predicted locations of primary and secondary separation points with errors of roughly two and zero degrees, respectively.

Mean pressure and skin friction predictions for the sphere solutions are also in good agreement with the measurements. The computed separation location is very close to the measured one, the error is less than one degree. The distribution of turbulent stresses shows that the turbulent flow around a sphere is highly anisotropic and supports the notion that using anisotropic turbulence models are necessary for three-dimensional separated flows.

Flow simulations around a Bell 214ST fuselage are performed for an isolated fuselage at three different flight conditions and helicopter with rotors modeled using
momentum theory in forward flight. Predicted pressure and drag force correlate well with the wind tunnel data. For a high angle of attack case drag was able to be predicted with a less than ten percent error. Drag predictions show not only the abilities of Reynolds Stress Model against eddy viscosity models but also its relative speed compared to Large Eddy Simulation.

Three-dimensional flow solutions require a considerable amount of computational time and memory requirements. In order to compensate for this, parallel processing is applied with the MPI communication standard. The codes are run on Beowulf clusters. The parallel performance of the code PUMA_RSM is analyzed and presented.
# TABLE OF CONTENTS

LIST OF FIGURES ..................................................................................................... viii

LIST OF TABLES ....................................................................................................... xix

ACKNOWLEDGEMENTS ......................................................................................... xx

Chapter 1 Introduction ................................................................................................ 1

1.1 Bluff Body Aerodynamics .............................................................................. 1
1.2 Helicopter Aerodynamics ............................................................................... 11

Chapter 2 Governing Equations .................................................................................. 22

2.1 Navier-Stokes Equations ................................................................................ 22
2.2 Euler Equations ............................................................................................... 23
2.3 Favre-Reynolds Averaged Navier-Stokes Equations ..................................... 24
2.4 Reynolds Stress Transport Equations ............................................................. 26
  2.4.1 Diffusion Term ..................................................................................... 28
  2.4.2 Dissipation Term .................................................................................. 28
  2.4.3 Redistribution Term.............................................................................. 29
2.5 Turbulence Dissipation Rate Equation ........................................................... 33

Chapter 3 Flow Solvers ............................................................................................... 35

3.1 Flow Solver PUMA2 ...................................................................................... 35
  3.1.1 Finite Volume Formulation .................................................................. 35
    3.1.1.1 Spatial Discretization .................................................................. 36
      3.1.1.1.1 Spatial Discretization of Convective Fluxes .................... 37
      3.1.1.1.2 Spatial Discretization of Viscous Fluxes ......................... 39
  3.1.1.2 Time Integration............................................................................. 40
    3.1.1.2.1 Time Step Computation ................................................... 41
  3.1.2 Boundary Conditions............................................................................ 42
  3.1.3 FANTAIL™ Modeling ........................................................................ 44
    3.1.3.1 Momentum Theory with Uniform Loading ............................... 45
    3.1.3.2 Blade Element Theory............................................................. 45
3.2 Flow Solver PUMA_RSM .............................................................................. 48
  3.2.1 Finite Volume Formulation .................................................................. 48
    3.2.1.1 Spatial Discretization ................................................................. 51
      3.2.1.1.1 Spatial Discretization of Convective Fluxes .................... 52
    3.2.1.2 Time Integration........................................................................ 54
    3.2.1.2.1 Time Step Computation ................................................... 54
  3.2.2 Boundary Conditions............................................................................ 55
LIST OF FIGURES

Figure 1-1: Iso-surface of absolute velocity (Ref. [1]) ................................................2
Figure 1-2: Flow over a cricket ball. (Ref [2]).............................................................3
Figure 1-3: Flow over a clockwise spinning golf ball (Ref [2]) .................................4
Figure 1-4: Variation of golf ball and sphere drag (Ref. [2]) .......................................4
Figure 1-5: Surface flow pattern of an automobile viewed from the driver’s side (Ref [3]) ................................................................................................................5
Figure 1-6: AH-64 Apache ..........................................................................................6
Figure 1-7: ROBIN Geometry (Ref. [7]) .....................................................................8
Figure 1-8: EUROCOPTER DGV Fuselage Configurations (Ref [14]) .......................9
Figure 1-9: Drag Predictions for the simpler DGV configuration (Ref. [13]) .........9
Figure 1-10: Iso-vorticity surface over Bell 214ST fuselage (Ref. [15]) .................10
Figure 1-11: Vortex shedding from the main rotor hub of RAH-66 Comanche (Ref. [17]) .............................................................................................................11
Figure 1-12: Turbulence Models ...............................................................................15
Figure 3-1: RAH-66 Comanche FANTAIL™ ..........................................................44
Figure 3-3: Lumley Triangle .....................................................................................57
Figure 4-1: Sustained 1-Hz directional axis oscillation in shallow turning partial power descent at 80 Knots ....................................................................................62
Figure 4-2: Computational mesh for RAH-66 Comanche .......................................64
Figure 4-3: Pressure distribution over isolated fuselage ($M_{\infty}=0.13$, $V_{\infty}=85.9$ knots, $\alpha=0^\circ$) .................................................................................................................66
Figure 4-4: Pressure distribution over fuselage ($M_{\infty}=0.13$, $V_{\infty}=85.9$ knots, $\alpha=0^\circ$, $C_{T\text{fan}}=0.008$) .............................................................................................................66
Figure 4-5: Comparison of computed dorsal midline pressure distribution with experiment ($M_{\infty}=0.13$, $V_{\infty}=85.9$ knots, $\alpha=0^\circ$, $C_{T\text{fan}}=0.008$) ..................................................67
Figure 4-6: Cp contours in the vicinity of FANTAIL™ (Hover, starboard and port views, $\theta_{.75} = -10^\circ$) ................................................................. 68

Figure 4-7: Velocity vectors in the vicinity of FANTAIL™, hover, $\theta_{.75} = -10^\circ$ (Net thrust is towards port side) ................................................................. 69

Figure 4-8: Cp contours in the vicinity of FANTAIL™, hover, starboard and port views, $\theta_{.75} = 20^\circ$ .................................................................................. 69

Figure 4-9: Velocity vectors in the vicinity of FANTAIL™, Hover, $\theta_{.75}=20^\circ$ (Net thrust is towards starboard side) ....................................................... 70

Figure 4-10: Cp contours in the vicinity of FANTAIL™ Hover, starboard and port views, $\theta_{.75} = 40^\circ$ ............................................................................ 70

Figure 4-11: Velocity vectors in the vicinity of FANTAIL™ (Hover, $\theta_{.75} = 40^\circ$) (Net thrust is towards starboard side) ....................................................... 71

Figure 4-12: Cp contours in the vicinity of FANTAIL™ (Forward Flight, starboard and port views, $V_\infty=150$ kts, $\theta_{.75} = 0^\circ$) .................................................... 72

Figure 4-13: Velocity vectors in the vicinity of FANTAIL™ (Forward Flight, $V_\infty=150$ kts, $\theta_{.75} = 0^\circ$) (Net thrust is towards port side) .................................................... 72

Figure 4-14: Cp contours in the vicinity of FANTAIL™ (Forward Flight, starboard and port views, $V_\infty=150$ kts, $\theta_{.75} = 20^\circ$) .................................................... 73

Figure 4-15: Velocity vectors in the vicinity of FANTAIL™ (Forward Flight, $V_\infty=150$ kts, $\theta_{.75} = 20^\circ$) (Net thrust is towards starboard side) .................................................... 73

Figure 4-16: Cp contours in the vicinity of FANTAIL™ (Forward Flight, starboard and port views, $V_\infty=150$ kts, $\theta_{.75} = 40^\circ$) .................................................... 74

Figure 4-17: Velocity vectors in the vicinity of FANTAIL™ (Forward Flight, $V_\infty=150$ kts, $\theta_{.75} = 40^\circ$) (Net thrust is towards starboard side) .................................................... 74

Figure 4-18: Cp contours in the vicinity of FANTAIL™ (Left Sideward Flight, starboard and port views, $V_\infty=45$ kts, $\theta_{.75} = 0^\circ$) .................................................... 75

Figure 4-19: Velocity vectors in the vicinity of FANTAIL™ (Left Sideward Flight, $V_\infty=45$ kts, $\theta_{.75} = 0^\circ$) (Net thrust is towards starboard side) .................................................... 76

Figure 4-20: Comparison of total thrust with experiment. (Hover) .................................................... 77

Figure 4-21: Comparison of fan thrust with experiment. (Hover) .................................................... 78
Figure 4-22: Comparison of fan thrust with experiment, right sideward flight, 
$V_\infty=45$ kts.............................................................................................................. 78

Figure 4-23: Fan thrust characteristics in different flight conditions. ...................... 79

Figure 4-24: Variation of yawing moment with sideslip angle, forward flight, 
$V_\infty=150$ kts, $\theta_{.75} = 0^\circ$ ................................................................. 81

Figure 4-25: Variation of yawing moment components with sideslip angle, 
forward flight, $V_\infty=150$ kts, $\theta_{.75} = 0^\circ$ ................................................................. 81

Figure 4-26: Variation of collective pitch angle with time (Hover, $\theta_{.75}=20^\circ\rightarrow15^\circ$) ... 83

Figure 4-27: Variation of fan thrust with time (Hover, $\theta_{.75}=20^\circ\rightarrow15^\circ$) .............. 83

Figure 4-28: Variation of total yawing moment with time. (Hover, $\theta_{.75}=20^\circ\rightarrow15^\circ$) .. 84

Figure 4-29: Variations of yawing moment components due to fan and fuselage 
with time. (Hover, $\theta_{.75}=20^\circ\rightarrow15^\circ$) ................................................................. 84

Figure 4-30: Variation of fan thrust with time. (Forward Flight, $V=150$ knots, 
$\theta_{.75}=20^\circ\rightarrow15^\circ$)................................................................................................. 85

Figure 4-31: Variation of total yawing moment with time. (Forward Flight, 
$V=150$ knots, $\theta_{.75}=20^\circ\rightarrow15^\circ$)................................................................................................. 85

Figure 4-32: Variations of yawing moment components due to fan and fuselage 
with time. (Forward Flight, $V=150$ knots, $\theta_{.75}=20^\circ\rightarrow15^\circ$) ................................................................. 86

Figure 4-33: Variation of collective pitch angle with time. (Forward Flight, 
$V=150$ knots, $\theta_{.75}=0^\circ\rightarrow5^\circ$)................................................................................................. 87

Figure 4-34: Variation of fan thrust with time. (Forward Flight, $V=150$ knots, 
$\theta_{.75}=0^\circ\rightarrow5^\circ$)................................................................................................. 88

Figure 4-35: Variation of total yawing moment with time. (Forward Flight, 
$V=150$ knots, $\theta_{.75}=0^\circ\rightarrow5^\circ$)................................................................................................. 88

Figure 4-36: Variations of yawing moment components with time. (Forward Flight, 
$V=150$ knots, $\theta_{.75}=0^\circ\rightarrow5^\circ$)................................................................................................. 89

Figure 4-37: Variation of average inflow velocity with time. (Forward Flight, 
$V=150$ knots, $\theta_{.75}=0^\circ\rightarrow5^\circ$)................................................................................................. 89

Figure 4-38: Pressure distribution around FANTAIL™. (Forward flight, $V = 150$ 
knots, $t = 0.011$ sec)................................................................................................. 90
Figure 4-39: Pressure distribution around FANTAIL™. (Forward flight, $V = 150$ knots, $t = 0.034$ sec) ................................................................. 91

Figure 4-40: Pressure distribution around FANTAIL™. (Forward flight, $V = 150$ knots, $t = 0.058$ sec) ................................................................. 91

Figure 4-41: Pressure distribution around FANTAIL™. (Forward flight, $V = 150$ knots, $t = 0.077$ sec) ................................................................. 92

Figure 4-42: Pressure distribution around FANTAIL™. (Forward flight, $V = 150$ knots, $t = 0.1$ sec) ................................................................. 92

Figure 4-43: Pressure distribution around FANTAIL™. (Forward flight, $V = 150$ knots, $t = 0.128$ sec) ................................................................. 93

Figure 4-44: Variation of collective pitch angle with time. (Hover, $\theta_{75}=0^\circ \rightarrow 5^\circ$) .... 94

Figure 4-45: Variation of fan thrust with time. (Hover, $\theta_{75}=0^\circ \rightarrow 5^\circ$) ....................... 95

Figure 4-46: Variation of total yawing moment with time. (Hover, $\theta_{75}=0^\circ \rightarrow 5^\circ$) .... 95

Figure 4-47: Variations of yawing moment components with time. (Hover, $\theta_{75}=0^\circ \rightarrow 5^\circ$) ........................................................................................................ 96

Figure 4-48: Variation of average inflow velocity with time. (Hover, $\theta_{75}=0^\circ \rightarrow 5^\circ$) .. 96

Figure 4-49: Variation of collective pitch angle with time. (Hover, $\theta_{75}=35^\circ \rightarrow 40^\circ$) .. 97

Figure 4-50: Variation of fan thrust with time. (Hover, $\theta_{75}=35^\circ \rightarrow 40^\circ$) ..................... 97

Figure 4-51: Variation of total yawing moment with time. (Hover, $\theta_{75}=35^\circ \rightarrow 40^\circ$) .. 98

Figure 4-52: Variations of yawing moment components with time. (Hover, $\theta_{75}=35^\circ \rightarrow 40^\circ$) ....................................................................................................... 98

Figure 4-53: Variation of average inflow velocity with time. (Hover, $\theta_{75}=35^\circ \rightarrow 40^\circ$) ....................................................................................................... 99

Figure 5-1: Contours of normalized invariant of deformation tensor ......................... 103

Figure 5-2: Sectional view of the mesh for 6:1 prolate spheroid (1 million cells) ..... 104

Figure 5-3: Close-up view of the cells near the solid boundary (6:1 prolate spheroid) (1 million cells) ........................................................................ 104

Figure 5-4: Surface skin friction lines ......................................................................... 105
Figure 5-5: Circumferential Velocity Distribution (Re=6.5x10^6, α=30°, M=0.1322, T_u=0.1%, Re_T=150, x/L=0.4839) ....................................................... 106

Figure 5-6: Circumferential Velocity Distribution (Re=6.5x10^6, α=30°, M=0.1322, T_u=0.1%, Re_T=150, x/L=0.6571) ....................................................... 107

Figure 5-7: Circumferential Velocity Distribution (Re=6.5x10^6, α=30°, M=0.1322, T_u=0.1%, Re_T=150, x/L=0.8279) ....................................................... 107

Figure 5-8: Circumferential Boundary Layer Profiles (Re=6.5x10^6, α=30°, M=0.1322, T_u=0.1%, Re_T=150, x/L=0.4839) ....................................................... 108

Figure 5-9: Circumferential Boundary Layer Profiles (Re=6.5x10^6, α=30°, M=0.1322, T_u=0.1%, Re_T=150, x/L=0.6571) ....................................................... 108

Figure 5-10: Circumferential Boundary Layer Profiles (Re=6.5x10^6, α=30°, M=0.1322, T_u=0.1%, Re_T=150, x/L=0.8279) ....................................................... 108

Figure 5-11: Variation of Turbulent Kinetic Energy (Re=6.5x10^6, α=30°, M=0.1322, T_u=0.1%, Re_T=150, x/L=0.4839) ....................................................... 109

Figure 5-12: Variation of Turbulent Kinetic Energy (Re=6.5x10^6, α=30°, M=0.1322, T_u=0.1%, Re_T=150, x/L=0.6571) ....................................................... 109

Figure 5-13: Variation of Turbulent Kinetic Energy (Re=6.5x10^6, α=30°, M=0.1322, T_u=0.1%, Re_T=150, x/L=0.8279) ....................................................... 109

Figure 5-14: Circumferential Velocity Distribution (Re=6.5x10^6, α=30°, M=0.1322, T_u=1%, Re_T=150, x/L=0.4839) ....................................................... 112

Figure 5-15: Circumferential Velocity Distribution (Re=6.5x10^6, α=30°, M=0.1322, T_u=1%, Re_T=150, x/L=0.6571) ....................................................... 112

Figure 5-16: Circumferential Velocity Distribution (Re=6.5x10^6, α=30°, M=0.1322, T_u=1%, Re_T=150, x/L=0.8379) ....................................................... 113

Figure 5-17: Surface skin friction lines (Re=6.5x10^6, α=30°, M=0.1322, T_u=1%, Re_T=150) ..................................................................................... 114

Figure 5-18: Turbulent Kinetic Energy Contours (Re=6.5x10^6, α=30°, M=0.1322, T_u=0.1%, Re_T=150) ................................................................. 115

Figure 5-19: Turbulent Kinetic Energy Contours (Re=6.5x10^6, α=30°, M=0.1322, T_u=1%, Re_T=150) ................................................................. 115
Figure 5-20: Surface skin friction lines (Re=6.5\times10^6, \alpha=30^\circ, T_u=1\%, Re_T=1.5 million, M=0.1322) ................................................................. 116

Figure 5-21: Circumferential Velocity Distribution (Re=6.5\times10^6, \alpha=30^\circ, M=0.1322, T_u=1\%, Re_T=1.5 million, x/L=0.4839) ................................................................. 117

Figure 5-22: Circumferential Velocity Distribution (Re=6.5\times10^6, \alpha=30^\circ, M=0.1322, T_u=1\%, Re_T=1.5 million, x/L=0.6571) ................................................................. 117

Figure 5-23: Circumferential Velocity Distribution (Re=6.5\times10^6, \alpha=30^\circ, M=0.1322, T_u=1\%, Re_T=1.5 million, x/L=0.8279) ................................................................. 118

Figure 5-24: Circumferential Velocity, x/L = 0.4839, \varphi = 30^\circ (Re=6.5\times10^6, \alpha=30^\circ, M=0.1322) ................................................................. 120

Figure 5-25: Circumferential Velocity, x/L = 0.4839, \varphi = 60^\circ (Re=6.5\times10^6, \alpha=30^\circ, M=0.1322) ................................................................. 121

Figure 5-26: Circumferential Velocity, x/L = 0.4839, \varphi = 90^\circ (Re=6.5\times10^6, \alpha=30^\circ, M=0.1322) ................................................................. 121

Figure 5-27: Circumferential Velocity, x/L = 0.4839, \varphi = 120^\circ (Re=6.5\times10^6, \alpha=30^\circ, M=0.1322) ................................................................. 122

Figure 5-28: Circumferential Velocity, x/L = 0.4839, \varphi = 150^\circ (Re=6.5\times10^6, \alpha=30^\circ, M=0.1322) ................................................................. 122

Figure 5-29: Circumferential Velocity, x/L = 0.6571, \varphi = 30^\circ (Re=6.5\times10^6, \alpha=30^\circ, M=0.1322) ................................................................. 123

Figure 5-30: Circumferential Velocity, x/L = 0.6571, \varphi = 60^\circ (Re=6.5\times10^6, \alpha=30^\circ, M=0.1322) ................................................................. 124

Figure 5-31: Circumferential Velocity, x/L = 0.6571, \varphi = 90^\circ (Re=6.5\times10^6, \alpha=30^\circ, M=0.1322) ................................................................. 124

Figure 5-32: Circumferential Velocity, x/L = 0.6571, \varphi = 120^\circ (Re=6.5\times10^6, \alpha=30^\circ, M=0.1322) ................................................................. 125

Figure 5-33: Circumferential Velocity, x/L = 0.6571, \varphi = 150^\circ (Re=6.5\times10^6, \alpha=30^\circ, M=0.1322) ................................................................. 125

Figure 5-34: Circumferential Velocity, x/L = 0.8279, \varphi = 30^\circ (Re=6.5\times10^6, \alpha=30^\circ, M=0.1322) ................................................................. 126
Figure 5-35: Circumferential Velocity, $x/L = 0.8279$, $\phi = 60^\circ$ ($Re=6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$) ............................................................127

Figure 5-36: Circumferential Velocity, $x/L = 0.8279$, $\phi = 90^\circ$ ($Re=6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$) ............................................................127

Figure 5-37: Circumferential Velocity, $x/L = 0.8279$, $\phi = 120^\circ$ ($Re=6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$) ............................................................128

Figure 5-38: Circumferential Velocity, $x/L = 0.8279$, $\phi = 150^\circ$ ($Re=6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$) ............................................................128

Figure 5-39: Surface skin friction lines ($Re=7.2\times10^6$, $\alpha=30^\circ$, $Tu=0.1\%$, $Re_T=150$, $M=0.1322$) ............................................................130

Figure 5-40: Axial Position of Separation Locations ($Re=7.2\times10^6$, $\alpha=30^\circ$, $Tu=0.1\%$, $Re_T=150$, $M=0.1322$) ............................................................130

Figure 5-41: Townsend’s Structural Parameter ($Re=4.0\times10^6$, $\alpha=10^\circ$, $Tu=0.03\%$, $Re_T=150$, $M=0.1322$, $x/L=0.7$, $\phi=120^\circ$) ............................................................132

Figure 5-42: Townsend’s Structural Parameter ($Re=4.0\times10^6$, $\alpha=10^\circ$, $Tu=0.03\%$, $Re_T=150$, $M=0.1322$, $x/L=0.7$, $\phi=150^\circ$) ............................................................133

Figure 5-43: Sectional view of the fine mesh for 6:1 prolate spheroid (4.7 million cells) ..................................................................................134

Figure 5-44: Surface skin friction lines ($Re=6.5\times10^6$, $\alpha=30^\circ$, $Tu=0.1\%$, $Re_T=150$, $M=0.1322$) ............................................................135

Figure 5-45: Circumferential Velocity Distribution ($Re=6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$, $Tu=0.1\%$, $Re_T=150$, $x/L=0.4839$) ............................................................137

Figure 5-46: Circumferential Velocity Distribution ($Re=6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$, $Tu=0.1\%$, $Re_T=150$, $x/L=0.6571$) ............................................................137

Figure 5-47: Circumferential Velocity Distribution ($Re=6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$, $Tu=0.1\%$, $Re_T=150$, $x/L=0.8379$) ............................................................138

Figure 5-48: Circumferential Boundary Layer Profiles ($Re=6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$, $Tu=0.1\%$, $Re_T=150$, $x/L=0.4839$) ............................................................139

Figure 5-49: Circumferential Boundary Layer Profiles ($Re=6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$, $Tu=0.1\%$, $Re_T=150$, $x/L=0.6571$) ............................................................139
Figure 5-50: Circumferential Boundary Layer Profiles (Re=6.5x10^6, α=30°, M=0.1322, Tₜ=0.1%, Reₜ=150, x/L=0.8279) ........................................................................................................140

Figure 5-51: Circumferential Velocity, x/L = 0.4839, φ = 30° (Re=6.5x10^6, α=30°, M=0.1322) ........................................................................................................................................141

Figure 5-52: Circumferential Velocity, x/L = 0.4839, φ = 60° (Re=6.5x10^6, α=30°, M=0.1322) ........................................................................................................................................142

Figure 5-53: Circumferential Velocity, x/L = 0.4839, φ = 90° (Re=6.5x10^6, α=30°, M=0.1322) ........................................................................................................................................142

Figure 5-54: Circumferential Velocity, x/L = 0.4839, φ = 120° (Re=6.5x10^6, α=30°, M=0.1322) ........................................................................................................................................143

Figure 5-55: Circumferential Velocity, x/L = 0.4839, φ = 150° (Re=6.5x10^6, α=30°, M=0.1322) ........................................................................................................................................143

Figure 5-56: Circumferential Velocity, x/L = 0.6571, φ = 30° (Re=6.5x10^6, α=30°, M=0.1322) ........................................................................................................................................144

Figure 5-57: Circumferential Velocity, x/L = 0.6571, φ = 60° (Re=6.5x10^6, α=30°, M=0.1322) ........................................................................................................................................145

Figure 5-58: Circumferential Velocity, x/L = 0.6571, φ = 90° (Re=6.5x10^6, α=30°, M=0.1322) ........................................................................................................................................145

Figure 5-59: Circumferential Velocity, x/L = 0.6571, φ = 120° (Re=6.5x10^6, α=30°, M=0.1322) ........................................................................................................................................146

Figure 5-60: Circumferential Velocity, x/L = 0.6571, φ = 150° (Re=6.5x10^6, α=30°, M=0.1322) ........................................................................................................................................146

Figure 5-61: Circumferential Velocity, x/L = 0.8279, φ = 30° (Re=6.5x10^6, α=30°, M=0.1322) ........................................................................................................................................147

Figure 5-62: Circumferential Velocity, x/L = 0.8279, φ = 60° (Re=6.5x10^6, α=30°, M=0.1322) ........................................................................................................................................148

Figure 5-63: Circumferential Velocity, x/L = 0.8279, φ = 90° (Re=6.5x10^6, α=30°, M=0.1322) ........................................................................................................................................148

Figure 5-64: Circumferential Velocity, x/L = 0.8279, φ = 120° (Re=6.5x10^6, α=30°, M=0.1322) ........................................................................................................................................149
Figure 5-65: Circumferential Velocity, $x/L = 0.8279$, $\varphi = 150^\circ$ ($Re=6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$) ..............................................................149

Figure 5-66: Circumferential Pressure Distribution ($Re=6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$, $T_u=0.1\%$, $Re_T=150$, $x/L=0.6062$). ..............................................................151

Figure 5-67: Variation of Turbulent Kinetic Energy ($Re=6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$, $T_u=0.1\%$, $Re_T=150$, $x/L=0.4839$) ..............................................................152

Figure 5-68: Variation of Turbulent Kinetic Energy ($Re=6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$, $T_u=0.1\%$, $Re_T=150$, $x/L=0.6571$) ..............................................................153

Figure 5-69: Variation of Turbulent Kinetic Energy ($Re=6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$, $T_u=0.1\%$, $Re_T=150$, $x/L=0.8279$) ..............................................................153

Figure 5-70: Sectional view of the mesh for a sphere (3.8 million cells) ........................................154

Figure 5-71: Close-up view of the cells near the solid boundary (sphere) (3.8 million cells) ........................................................................................................155

Figure 5-72: Circumferential Pressure Distribution ($Re=1.14\times10^6$, $M=0.1763$, $T_u=0.45\%$, $Re_T=150$) ..............................................................156

Figure 5-73: Velocity Distribution with surface skin friction lines ($Re=1.14\times10^6$, $M=0.1763$, $T_u=0.45\%$, $Re_T=150$) ..............................................................157

Figure 5-74: Circumferential Skin Friction Distribution ($Re=1.14\times10^6$, $M=0.1763$, $T_u=0.45\%$, $Re_T=150$) ..............................................................158

Figure 5-75: Iso-vorticity surface in the wake of a sphere ($Re=1.14\times10^6$, $M=0.1763$, $T_u=0.45\%$, $Re_T=150$) ..............................................................159

Figure 5-76: $a_{xx}$ Contours ($Re=1.14\times10^6$, $M=0.1763$, $T_u=0.45\%$, $Re_T=150$) ......................160

Figure 5-77: $a_{zz}$ Contours ($Re=1.14\times10^6$, $M=0.1763$, $T_u=0.45\%$, $Re_T=150$) ......................160

Figure 5-78: $a_{xz}$ Contours ($Re=1.14\times10^6$, $M=0.1763$, $T_u=0.45\%$, $Re_T=150$) ......................161

Figure 5-79: Sectional view of the computational mesh (Bell 214ST) (2.9 million cells) ..............................................................162

Figure 5-80: Sectional view of the computational mesh (Bell 214ST) (2.9 million cells) ........................................................................................................163

Figure 5-81: Actuator disks for rotor modeling ..................................................................................164
Figure 5-82: Surface Pressure Distribution ($\alpha = -2.28^\circ, \psi = 0^\circ$) ........................................... 165

Figure 5-83: Dorsal Centerline Pressure Distribution ($\alpha = -2.28^\circ, \psi = 0^\circ$) ....................... 166

Figure 5-84: Normalized $\tau_{xz}$ contours ($\alpha = -2.28^\circ, \psi = 0^\circ$) ........................................ 167

Figure 5-85: Normalized $\tau_{yz}$ contours ($\alpha = -2.28^\circ, \psi = 0^\circ$) ........................................ 167

Figure 5-86: Surface Pressure Distribution ($\alpha = 17.4^\circ, \psi = 0^\circ$) ..................................... 168

Figure 5-87: Dorsal Centerline Pressure Distribution ($\alpha = 17.4^\circ, \psi = 0^\circ$) ....................... 169

Figure 5-88: Normalized $\tau_{xz}$ contours ($\alpha = 17.4^\circ, \psi = 0^\circ$) ...................................... 170

Figure 5-89: Normalized $\tau_{yz}$ contours ($\alpha = 17.4^\circ, \psi = 0^\circ$) ...................................... 170

Figure 5-90: Normalized $\tau_{xz}$ contours computed using Boussinesq approximation
($\alpha = 17.4^\circ, \psi = 0^\circ$) ........................................................................................................ 171

Figure 5-91: Surface Pressure Distribution ($\alpha = -1.6^\circ, \psi = 16.4^\circ$) ................................... 172

Figure 5-92: Dorsal Centerline Pressure Distribution ($\alpha = 1.6^\circ, \psi = 16.4^\circ$) ................. 172

Figure 5-93: Normalized $\tau_{xz}$ contours ($\alpha = -1.6^\circ, \psi = 16.4^\circ$) .................................... 173

Figure 5-94: Normalized $\tau_{yz}$ contours ($\alpha = -1.6^\circ, \psi = 16.4^\circ$) .................................... 173

Figure 5-95: Pressure Distribution at $x/L = 0.32$. ($\alpha = -2.28^\circ, \psi = 0^\circ$, $T = 17500$ lbs, $T_{tr} = 1104$ lbs) ................................................................. 174

Figure 5-96: Pressure Distribution at $x/L = 0.32$. ($\alpha = -2.28^\circ, \psi = 0^\circ$, without rotors) ................................................................. 175

Figure 5-97: Pressure Distribution at $x/L = 0.95$. ($\alpha = -2.28^\circ, \psi = 0^\circ$, $T = 17500$ lbs, $T_{tr} = 1104$ lbs) ................................................................. 175

Figure 5-98: Pressure Distribution at $x/L = 0.95$. ($\alpha = -2.28^\circ, \psi = 0^\circ$, without rotors) ................................................................. 176

Figure 5-99: Normalized $w$ distribution at the main rotor plane ($\alpha = -2.28^\circ, \psi = 0^\circ$, $T = 17500$ lbs, $T_{tr} = 1104$ lbs) ................................................................. 176

Figure 5-100: Normalized $w$ distribution at the main rotor plane ($\alpha = -2.28^\circ, \psi = 0^\circ$, without rotors) ................................................................. 177
Figure 5-101: Normalized $\tau_{xz}$ contours ($\alpha=-2.28^\circ$, $\psi=0^\circ$, $T=17500$ lbs, $T_r=1104$ lbs) ...................................................................................................................................................................................... 178

Figure 5-102: Normalized $\tau_{yz}$ contours at $x/L = 0.32$ ($\alpha=-2.28^\circ$, $\psi=0^\circ$, $T=17500$ lbs, $T_r=1104$ lbs) ...................................................................................................................................................................................... 179

Figure 5-103: Normalized $\tau_{yz}$ contours at $x/L = 0.95$ ($\alpha=-2.28^\circ$, $\psi=0^\circ$, $T=17500$ lbs, $T_r=1104$ lbs) ...................................................................................................................................................................................... 179

Figure 5-104: Normalized $\tau_{yz}$ contours at $x/L = 0.95$ ($\alpha=-2.28^\circ$, $\psi=0^\circ$, without rotors)...................................................................................................................................................................................... 180

Figure 5-105: Relative Speedup vs. Number of Processors (COCOA3).......................... 186

Figure 5-106: Relative Speedup vs. Number of Processors (MUFASA, coarse mesh)...................................................................................................................................................................................... 187
LIST OF TABLES

Table 3-1: List of Boundary Conditions in PUMA2 ...................................................42
Table 5-1: Primary and secondary separation locations $x/L = 0.738$ .........................136
Table 5-2: Drag predictions and comparisons with wind tunnel data .......................182
Table 5-3: Relative speedup with number of processors (COCOA3) .........................185
Table 5-4: Relative speedup with number of processors (MUFASA) .........................186
ACKNOWLEDGEMENTS

I express sincere appreciation to my adviser Dr. Lyle N. Long for his perfect guidance, never-ending support and understanding throughout my thesis studies. He has been much more than an adviser for me.

I also want to thank my thesis committee members for their precious suggestions and comments about my thesis.

I gratefully thank my family for their extraordinary support, understanding and assistance. I could not have achieved anything without them.

I offer sincere thanks to my office mates and to my friends for their valuable support and friendship.

Finally, special thanks to Gulkiz Dogan.
to my dear parents Hulya and Adil ALPMAN…
Chapter 1

Introduction

1.1 Bluff Body Aerodynamics

Flow around bluff bodies is a very challenging phenomenon in aerodynamics. Some of the basic features of these flows are strong pressure gradients, vortex shedding, three-dimensionality and massive separation. Separated flow causes many phenomena in aerodynamics such as lift loss, drag increase, unsteadiness and three-dimensional effects. Even separated flows around two-dimensional bluff bodies may turn out to be three-dimensional. Figure 1-1 shows an iso-surface of absolute vorticity for flow over a circular cylinder from Ref.[1]. The three-dimensionality of the vortical wake is evident from the figure.

Drag of a bluff body is a very challenging aerodynamic characteristic to determine accurately. Unlike streamlined bodies, where drag is dominated by skin friction, the drag of bluff bodies is mainly due to pressure drag, which occurs because of pressure losses in the wake due to vorticity formation. Therefore, it is very important for separated flows and the correct prediction of the drag of bluff bodies requires the ability to accurately simulate separated flows.
The aerodynamic drag of a bluff body is important in many applications. Sports balls are designed so that they can take advantage of the flowfield around them. For example, in cricket and baseball the major aim is to deliberately curve the ball. On the other hand, in golf, the ball should cover the maximum distance in flight; therefore a maximum lift-to-drag ratio is needed [2]. In cricket, seams on the ball can be used to trip the boundary layer on one side for asymmetric boundary layer separation. This results in an asymmetric pressure distribution that produces the side force necessary for spin. Figure 1-2, taken from Ref. [2], shows a stationary cricket ball in a wind tunnel with the seam set at an incidence angle of 40 degrees to the freestream. It is evident from this figure that the turbulent boundary layer of the lower side separated later than the laminar boundary layer of the upper side; causing an asymmetric pressure distribution on the ball.

In golf ball aerodynamics lift is generated by spinning the ball, which causes asymmetric boundary layer separation and provides an upward force. This situation can
be seen in Figure 1-3, taken from Ref. [2], where flow over a clockwise spinning golf ball is displayed. The dimples on the ball, seen in this figure, trip the boundary layer and help keep the flow attached and reduce the drag. In Figure 1-4 the variation of golf ball drag with Reynolds number is compared with the drag of smooth and rough spheres. In this figure $k$ is the roughness height and $d$ is the diameter of the ball (Ref. [2]). This reference also suggests that at its highest speed, a golf ball operates at a Reynolds number on the order of $10^5$. Thus, the figure clearly shows the effectiveness of dimples in reducing the drag.

Figure 1-2: Flow over a cricket ball. (Ref [2])
For automobiles fuel consumption is an important issue. Low fuel consumption is desired for both economic and environmental reasons. In automobile aerodynamics,
resistance is mostly due to aerodynamic drag especially at higher speeds. Aerodynamic optimization of an automobile may not always be possible because of the design constraints based on external view, passenger and cargo requirements. Therefore, automobiles can be regarded as complex bluff bodies operating in strong ground effect. Flow around them can be very complex. Figure 1-5, taken from Ref. [3], shows the surface flow pattern of an automobile viewed from the driver’s side. Visualization was performed at the NASA Langley Basic Aerodynamics Research Tunnel. The figure clearly shows massive flow separation in the vicinity of the rear windows and deck lid.

![Figure 1-5: Surface flow pattern of an automobile viewed from the driver’s side (Ref [3])](image)

Like automobiles, the design of a helicopter fuselage is also constrained by non-technical parameters such as shape, size, and type of payload. Since a helicopter spends considerable time in hover, and major control forces and moments are provided by the
rotors rather than fuselage components; fuselage designs may be performed according to the mission requirements rather than aerodynamic efficiency. Therefore, helicopter fuselages can be very complex and non-streamlined. One can refer to Figure 1-6 for the complex geometry of AH-64 Apache helicopter. The helicopter fuselage, however, significantly affects the overall performance of a helicopter in all flight conditions. In hover, the drag of the fuselage is subtracted from the rotor thrust. In forward flight, drag must be compensated for by the forward component of the rotor thrust. Hence, fuselage drag affects the whole rotor configuration for trim [4]. Therefore, understanding and predicting fuselage aerodynamics is important for future helicopter designs. The geometry of the fuselage and the wide range of flight conditions ensure that some amount of separation will be present. Hence, accurate prediction of fuselage aerodynamics requires accurate modeling of separated flows.

---

Figure 1-6: AH-64 Apache
The relatively high computational speed and simplicity of potential-flow-based panel methods make them suitable to be used in design processes. Berry et al., [4, 5, 6] and Chaffin et al., [7] employed panel methods to simulate flow over a ROtor Body INteraction (ROBIN) geometry, which can be seen in Figure 1-7 (Ref [7]). It is a generic geometry designed to represent a wide range of helicopter fuselages. But it is very streamlined compared to new generation helicopters. Their results showed that even with boundary layer correction, panel methods could not simulate separated flow correctly. Methods failed to predict separation locations, and strength and trajectory of the vortical wake [4]. For accurate solutions panel methods required the separation location to be known a priori. This makes them insensitive to geometry changes and thus makes them unsuitable for helicopter fuselage problems. Methods based on Navier-Stokes equations are computationally much more complex and expensive than panel methods but they are known to handle separated and vortical flows. Berry et al.[4, 5, 6] and Chaffin et al. [7] performed Navier-Stokes simulations for the flow over ROBIN and compared with panel methods to observe the improvements. Narramore and Brand [8] performed drag predictions for a Bell 214ST fuselage using a Navier-Stokes solver and the Baldwin-Lomax algebraic turbulence model. Duque et al. [9, 10] performed Navier-Stokes solutions for an AH-66 Comanche fuselage using Baldwin-Lomax algebraic and Baldwin-Barth one-equation turbulence models. They mentioned the weakness of the algebraic turbulence model for separated flows and overset grids. The one-equation model proved to be more suitable for overset grids but still could not make significant improvements.
In Europe researchers from three national research centers (CIRA, DLR, ONERA), three helicopter manufacturers (Agusta, Eurocopter, and GKN-Westland), one software company (SIMULOG), and two university laboratories (DTU and IAG) were involved in a very extensive helicopter drag prediction program called HELIFUSE [11, 12, 13]. Predictions were performed for a EUROCOPTER DGV fuselage. Figure 1-8 shows two configurations of this fuselage used in wind tunnel tests [14] and computations [11, 12, 13]. For their computations they employed various Navier-Stokes solvers equipped with algebraic and two-equation turbulence models. Figure 1-9 displays drag predictions obtained for the simpler fuselage configuration [13]. Large discrepancies between various computations were observed and in general predictions underestimated the measurements. Note that the simpler fuselage configuration is highly streamlined and very small separation was present during the predictions. Their results showed that
simpler turbulence models may not be able to predict aerodynamic forces on complex helicopter geometries.

Figure 1-8: EUROCOPTER DGV Fuselage Configurations (Ref [14])

Figure 1-9: Drag Predictions for the simpler DGV configuration (Ref. [13])
At the Pennsylvania State University, Souliez and Long [15] performed Large Eddy Simulation solutions for a Bell 214ST fuselage in order to predict drag and simulate unsteady flowfield around the helicopter. Figure 1-10 shows an instantaneous picture of an iso-vorticity sheet [15]. This illustrates the vortical wake shedding from main rotor hub and fuselage side walls. The downstream traveling wake impinges on horizontal and vertical stabilizers. Being a dynamic model, Large Eddy Simulation proved to be very powerful to simulate the unsteady turbulent flow. But the very high computational cost required huge amount of CPU time and forced the researchers to use relatively coarse meshes.

Alpman and Long [16] performed drag predictions for the same Bell 214ST fuselage using Reynolds Stress Modeling (seven-equation model) and compared with the results obtained in Ref. [15]. Their Reynolds Stress Model was able to predict drag with reasonable accuracy and was nearly ten times faster than Large Eddy Simulation.
In addition to the fuselage, the rotor hub can also significantly affect the aerodynamic drag and performance of other fuselage components. Figure 1-11 shows vortex shedding from the hub of RAH-66 Comanche helicopter (Ref [17]). As mentioned before, the vortical wake behind a bluff body causes pressure drop and generates pressure drag. Note also that the vertical tail and the FANTAIL™ of the helicopter are in the vortical wake.

![Figure 1-11: Vortex shedding from the main rotor hub of RAH-66 Comanche (Ref. [17])](image)

### 1.2 Helicopter Aerodynamics

A helicopter is an aircraft which can be defined as a flying machine using rotating wings to provide lift, propulsion, and control forces that enable the aircraft to hover relative to the ground without forward flight speed [18]. It is the relatively low amount of power required to lift the aircraft compared to other vertical take off and landing (VTOL) aircraft that makes the helicopter unique. But efficient hovering flight with low power requirements comes with the price of aerodynamic and mechanical complexity. Consequently, helicopter aerodynamics has always been one of the most challenging problems in aerospace. In addition to the challenge of complex lifting surfaces operating
in unsteady environments, the interaction between the rotating surfaces and the helicopter fuselage components amplifies the complexity [5].

Many of the numerical studies on helicopter aerodynamics are based on coupling the isolated rotor aerodynamics with linear aerodynamic fuselage analyses or measured isolated fuselage data using superposition. But the idea of linear superposition was shown to have weaknesses [9, 19]. Therefore, models which can handle nonlinear interaction effects are needed for accurate prediction of interactional rotorcraft aerodynamics. Computational fluid dynamics (CFD), which allows a more complete mathematical model to make quantitative predictions of complex flows dominated by nonlinear effects [20], can be employed for this purpose. The most complete model for rotor fuselage interaction prediction is to directly solve for rotor and fuselage flowfields using overset grids [21,22,23]. The method is capable of capturing realistic unsteady flow features such as tip-vortex generation, blade-vortex interactions, or tip-vortex impingement on the fuselage. But it requires a good sliding mesh algorithm for convection of flow variables across the boundary between rotating and stationary meshes [24]. In addition to this, the method also requires vast amounts of computer resources for predictions. An alternative way is to predict time-averaged rotor-fuselage interaction by modeling the rotor flow. The most common approach is to model the rotor as an actuator disk, in which the rotor is assumed to have an infinite number of blades and zero thickness [18]. In this model the pressure undergoes a discontinuity across the actuator-disk while the other flow parameters remain continuous. The simplest version of this model is the uniform actuator disk model where the pressure jump is constant all over the rotor disk [18, 19, 25, 26, 27]. Although this representation greatly simplifies the computational process, it is not able to
account for the blade rotation effects and lateral dissymmetry in the velocity field. An alternative to the uniform model is the non-uniform actuator disk model where the pressure jump depends on the local lift and drag created by the blade passage at the azimuthal and radial positions [25,27-32]. The pressure jump at any position is computed using blade element theory [33]. Although the model is only good for time averaged flow, it can successfully be employed for unsteady problems when the time scales of the problem are much larger when compared to the time scales of the rotor [34].

Besides accurate rotor wake predictions, the flowfields of helicopter fuselages also present a challenging problem. The relatively complex and non-streamlined geometry of helicopter fuselages usually lead to separated flows. Flow separates from the hub or fuselage and then impinges on the tail rotor, empennage and control surfaces.

Earlier researchers employed potential flow theory [4, 7, 35] due to its simplicity and speed. The Euler Equations [21,22,23,26,27,34] and the Navier-Stokes equations [4-13, 15, 16, 30, 35, 36] have also been used to define flowfields around helicopters. Each of these methods has an associated computational cost, limitations, and benefits [9]. Panel methods, being the cheapest of all, have been shown to be limited by their inability to predict flow separation. To overcome this problem panel methods are coupled with a boundary layer model [37,38,39]. Although improvements were observed, the model is still not sufficient for helicopter fuselage flows [7]. The Euler equations constitute the most complete description of inviscid, non-heat conducting flows and hence is the highest level of approximation of the non-viscous fluids [40]. In this sense, they stimulate physical flows in the limit of vanishing viscosity. Although inviscid flow models are not universally valid, their accurate modeling of real flows resides in the dominating
convective character of high Reynolds number flows [41]. While it is true that viscous effects are relatively unimportant outside the boundary layer, the presence of the boundary layer can have a drastic influence on the global pattern of the flow [20]. This will be the case when the flow separates. Therefore, in the presence of massive separation, which is usually the case in helicopter fuselage flows, the inviscid flow assumption can be a serious drawback. However, for geometries with sharp edges and known separation points the Euler equations can be quite effective [42]. Separated flow around a body results in many phenomena in aerodynamics, such as drag increase, lift loss, unsteady fluctuations, etc. Therefore, implementation of Navier-Stokes equations with an accurate turbulence model is a key to understanding and predicting separated turbulent flows around aerodynamic devices [15].

Figure 1-12 shows a general view of the turbulence models which are being used in the literature. It is evident from this figure that by using some assumptions and approximations computational cost can be reduced but physics will be lost. The presence of three-dimensionality and curvature introduces changes in the turbulence structure, thus invalidating many of the turbulence models used widely for simple and mildly complex shear layers. Therefore, it becomes extremely important to employ more physics in providing suitable closure models for adequate prediction of these complex flows.
Direct Numerical Simulation (DNS) yields the most complete description of turbulent flows. It consists in solving the Navier-Stokes equations, resolving all the scales of motion, with appropriate initial and boundary conditions. Each simulation produces a single realization of the flow. Conceptually it is the simplest approach and, when it can be applied, it is unrivalled in accuracy and in the level of description provided [43]. But the computational cost of resolving all the scales of turbulence is extremely high and the computer requirements increase rapidly with Reynolds number (∼ Re³) [44]. Therefore, the approach is limited to low Reynolds number and homogeneous turbulent flows, where the properties of turbulence do not change with position [45]. In addition to this, very fine grid requirements to resolve all the scales may generate accuracy and numerical stability problems.
Large Eddy Simulation (LES), where the energy-carrying large-scale turbulent motions are computed, while smaller scale motions are modeled, can be considered as an alternative to DNS [43]. Since small scales tend to be more homogeneous and universal, and less affected by the boundary conditions than the large ones, their modeling can be simpler and require fewer adjustments when applied to different flows [15]. To separate the large scales from the smaller ones, LES is based on the definition of a filtering operation, which decomposes the instantaneous velocity field into the sum of a filtered (or resolved) component and a residual (or sub-grid scale, SGS). It has been shown that filtering introduces errors on non-uniform grids [46]. Therefore, solutions require uniform grid topologies. In the case of near-wall flows, such as boundary layers, this requirement leads to very small mesh spacing in all directions; resulting in a huge number of grid points [47].

In the LES approach, the equations for the evolution of the filtered velocity field are derived from the Navier-Stokes equations. These equations contain the SGS stress tensor which arises from the residual motions. Closure is obtained by modeling the SGS stress tensor, mostly by an eddy-viscosity model. It was shown that eddy-viscosity models can reproduce the SGS dissipation quite well, but not the SGS forces entering the momentum equation, therefore, making this approach less-suited for complex high Reynolds number flows [48]. In addition to this, the time dependent nature of LES makes it a 4-dimensional problem (space and time). For three-dimensional High Reynolds number flows, the solutions require thousands or millions of time steps. Souliez employed LES for unsteady flow predictions around a Bell 214ST fuselage in his PhD
thesis [15]. The solution for a physical time of 1.25 seconds took 60 days on sixteen 800 MHz Pentium III processors.

Detached Eddy Simulation (DES) is a hybrid method which combines LES with an eddy-viscosity model. The major goal is to benefit from often sufficient performance of an eddy-viscosity model in the attached boundary layer region and to use LES away from the boundaries. It is suitable for massively separated flows [49]. DES reduces to an eddy-viscosity model for attached flows and its advantages of describing three-dimensional unsteady turbulent motions may degrade when the flow is not massively separated [50].

The other branch of approximate turbulence models in Figure 1.1 is the one which employs time averaging. Although these models do not completely represent the unsteady behavior of turbulence, they considerably decrease the computational cost by reducing the 4-dimensional turbulence problem to a three-dimensional one (space only). These models can be divided into two subgroups; eddy viscosity models with the Boussinesq approximation and Reynolds stress models. Many of the turbulence models in the literature (e.g. k-ε, k-ω, Spalart-Allmaras, Baldwin-Lomax) are based on the Boussinesq approximation which assumes that the principal axes of the Reynolds stress tensor are coincident with those of the mean strain rate at all points in a turbulent flow [43]. According to the Boussinesq hypothesis turbulent stresses are defined as:

\[ \tau_{ij} \approx -\mu_T \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) + \frac{2}{3} k \delta_{ij} \]  

1.1
Where $\mu_r$ is the eddy viscosity, $\bar{u}_i$ is the mean velocity component in the $i^{th}$ direction and $k$ is the turbulent kinetic energy. For complex three-dimensional flows with sudden changes in mean strain rate the Boussinesq hypothesis can be seriously in error because the Reynolds stresses will adjust to such changes at a rate unrelated to mean flow processes and time scales [51].

The Boussinesq approximation also yields the necessity of defining length and time scales of turbulence for eddy viscosity calculations. In zero and one equation models, these scales are defined in an ad hoc fashion for different flows. The models are isotropic in nature and are only valid for two-dimensional simple shear flows. Therefore, two-equation models constitute the minimum level of closure that is physically acceptable [51]. Although two-equation models are superior to zero and one equation models, without modification, they still fail to capture many of the features associated with complex flows [52]. While these models can be modified to improve their predictive accuracy, the modifications are largely ad hoc and cannot be easily generalized [51]. Moreover, in the case of strong separation, even the modified two-equation models were shown to fail to predict flow physics due to their isotropic nature [53]. Therefore, anisotropic models such as full Reynolds stress transport models (RSM) [54] are necessary for accurate prediction of three-dimensional separated flows. In references 23 and 26, a Baldwin-Lomax model [55] is used to predict the flowfield around the rotor body interaction (ROBIN) geometry [56]. Considerable discrepancies were observed at the rear fuselage stations where the flow is separated. Duque et al. also mentioned the non-suitability of Baldwin-Lomax model for separated flow fields [9, 10].
Reynolds stress models are second-moment closures based on the differential equations governing the transport of the six Reynolds stress components. In these models there is no need to define an eddy viscosity, which is purely phenomenological and has no mathematical basis [52]. Therefore, the Reynolds stress model constitutes the highest level of closure. Since the model contains the convection and diffusion of the Reynolds stress, it automatically incorporates the history effects of the flow. In addition to this, the model is anisotropic. Therefore, it can accommodate the effects of rotation and curvature [51]. Although RSM does not represent the unsteady nature of turbulence as do dynamic models such as LES, it is very effective in computing the time averaged quantities and is an order of magnitude cheaper than LES.

Based on the above discussion, this study is divided into two parts. The first part consists of inviscid flow predictions around an RAH-66 Comanche fuselage, using actuator disk models with uniform and non-uniform loadings for the fan-in-fin. The main rotor is excluded in this study. Solutions are obtained by modifying the computer code PUMA2 (Parallel Unstructured Maritime Aerodynamics) [15,57-60], and using an unstructured grid of 2.8 million tetrahedral cells. PUMA2, written in ANSI C++, is an in-house computational fluid dynamics code, which has been used and validated by Long et al. for numerical solution of numerous problems [1, 15, 27,34,57-66]. Static and dynamic thrust and antitorque moment response to applied collective pitch in hover, forward flight and sideward flight are presented and discussed.

The second part of the research consists of the implementation and application of Reynolds Stress Modeling (RSM) for numerical simulation of separated turbulent flows around three-dimensional bodies. The anisotropic nature of RSM brings many advantages
over eddy viscosity models, which can be in error for complex turbulent flows. But the complexity of the Reynolds Stress equations and high computer requirements drove researchers to simpler turbulence models and prevented RSM from becoming popular. Increasing computing power has now allowed more physics to be added to the models and RSM has started to be used to simulate more turbulent flows. The objectives of the research described in this dissertation are to implement RSM which has the potential of better capturing the physics of complex turbulent flows and to evaluate the capabilities of the solver by predicting turbulent flows around three-dimensional bodies where extensive experimental data are available. To the author’s knowledge, RSM has not been used before on a complex geometry such as a helicopter fuselage. In this study solutions are obtained using the computer code PUMA_RSM, which is a modified version of PUMA2, and unstructured grids. High Reynolds number turbulent flow solutions around a 6:1 prolate spheroid, a sphere and a Bell 214ST fuselage are presented and discussed. PUMA_RSM is not only of value for helicopters. It can also be used to predict flow around any complex geometry such as automobiles and ships.

Even today, three-dimensional solutions of Euler and Navier-Stokes equations require a considerable amount of computer time and memory. The power of conventional serial and vector computers is inadequate to allow solutions of the equations routinely and quickly. Parallel processing, in which the computational job is split and distributed over multi-processors, can alleviate the problem of computer resources. In the parallel processing approach the computational work is distributed over more than one processor to be performed simultaneously [67]. The essential element of parallel processing is data communication between the processes. In this study, data transfer between the processes
is performed using the Message Passing Interface (MPI) [68] communication standard. The codes are run on locally available computing facilities.
Chapter 2

Governing Equations

2.1 Navier-Stokes Equations

The Navier-Stokes equations are the mass, momentum and energy conservation equations written for a Newtonian fluid, and is the most general description of fluid flow in thermodynamic equilibrium or slightly nonequilibrium [20, 69].

\[
\frac{\partial \rho}{\partial t} + \sum_{i} \frac{\partial \rho u_i}{\partial x_i} = 0
\]

2.1

\[
\frac{\partial \rho u_i}{\partial t} + \sum_{j} \frac{\partial \rho u_i u_j}{\partial x_j} = -\sum_{j} \frac{\partial p}{\partial x_j} + \frac{\partial t_{ij}}{\partial x_j}
\]

2.2

\[
\frac{\partial \rho E}{\partial t} + \sum_{i} \frac{\partial \rho u_i H}{\partial x_i} = \sum_{j} \frac{\partial (u_i t_{ij})}{\partial x_j} + \frac{\partial q_i}{\partial x_i}
\]

2.3

where \( \rho \) is the density, \( u_i \) is the velocity vector, \( p \) is the static pressure, \( E \) and \( H \) are total energy and total enthalpy per unit volume, \( t_{ij} \) is the viscous stress tensor, and \( q_i \) is the rate of heat transfer in the \( i^{th} \) direction.

\[
E = \frac{p}{\rho (\gamma - 1)} + \frac{u_i u_i}{2}
\]

2.4

\[
H = E + \frac{p}{\rho}
\]

2.5

\[
t_{ij} = \mu \left( \frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right) + \left( \mu_b - \frac{2}{3} \mu \right) \frac{\partial u_k}{\partial x_k} \delta_{ij}
\]

2.6

\[
q_i = -\kappa \frac{\partial T}{\partial x_i}
\]

2.7
where $T$ is the fluid temperature. In Eq. 2.6 $\mu_b$ is the bulk viscosity. For incompressible flows its value is zero but can be important when density variations are not negligible [69]. In this dissertation the flowfields analyzed are at low speeds and density variations are small. Therefore, value of bulk viscosity is assumed to be zero throughout the thesis. The dynamic viscosity, $\mu$, and thermal conductivity, $k$, appearing in Eq. 2.6 and Eq. 2.7 must also be related to other flow variables. Sutherland’s Formula [70] provides a relation between viscosity and temperature:

$$\mu = 1.458 \times 10^{-6} \left[ \frac{T^{3/2}}{T + 110.4K} \right] \text{kg/(m s K$^{1/2}$)}$$  \hspace{1cm} 2.8

Thermal conductivity is related to the dynamic viscosity through the Prandtl number $Pr$ which describes the ratio of momentum and thermal diffusivities [71].

$$\kappa = \frac{\mu C_p}{Pr}$$  \hspace{1cm} 2.9

### 2.2 Euler Equations

The Euler equations, which give the most general description of inviscid, non-heat conducting fluids, are derived from the Navier-Stokes equations simply by dropping the viscous and heat conducting terms.

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0$$  \hspace{1cm} 2.10

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i}$$  \hspace{1cm} 2.11

$$\frac{\partial \rho E}{\partial t} + \frac{\partial \rho u_i H}{\partial x_i} = 0$$  \hspace{1cm} 2.12
2.3 Favre-Reynolds Averaged Navier-Stokes Equations

It is possible to derive the evolution of a time averaged turbulent velocity field starting from the Navier-Stokes equations. The most basic of these equations (first derived by Reynolds (1894)) are those that govern the mean velocity field [43]. The original Reynolds Averaged Navier-Stokes equations were written for incompressible flows. For flows where the density variations cannot be neglected, it is better to use Favre averaging [51] rather than standard time averaging which will create additional terms that have no analogs in the laminar equations. Favre averaging for a velocity component is:

\[
\tilde{u}_i = \frac{1}{\bar{\rho}T} \int_t^{t+T} \rho u_i \, dt
\]

where \( \bar{\rho} \) is the time averaged density. \( T \) is the interval over which individual fluctuating terms vanish, i.e.,

\[
\frac{1}{\bar{\rho}T} \int_t^{t+T} \rho u'_i \, dt = 0
\]

For steady state flows this value is taken to be infinity [43]. For unsteady flows it is restricted by the time scale of the wave propagation under which no turbulence scales can be resolved [72].

The Favre-Reynolds Averaged Navier-Stokes Equations are written as [73]:

\[
\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial \bar{\rho} \bar{u}_i}{\partial x_i} = 0
\]  \hspace{1cm} 2.15

\[
\frac{\partial \bar{\rho} \bar{u}_i}{\partial t} + \frac{\partial (\bar{\rho} \bar{u}_i \bar{u}_j + \bar{\rho} \delta_{ij})}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \bar{\tau}_{ij} - \bar{\tau}_{ij} \right] = 0
\]  \hspace{1cm} 2.16

\[
\frac{\partial \bar{\rho} \bar{E}}{\partial t} + \frac{\partial}{\partial x_j} \left[ \bar{\rho} \bar{u}_j \bar{H} \right] - \frac{\partial}{\partial x_j} \left[ \tilde{u}_i \left( \bar{\tau}_{ij} - \tau_{ij} \right) - \left( \bar{q}_j + \bar{q}_j \right) \right] = S
\]  \hspace{1cm} 2.17
where \( \tilde{u}_i \) are mean velocity components, \( \bar{E} \) is the total energy per unit mass, \( \tau_{ij} \) are the Reynolds stresses, \( q_{ij} \) is the turbulent heat transfer rate in the \( j^{th} \) direction and \( \delta_{ij} \) is the Kronecker’s delta. In the equations (\( \overline{\cdot} \)) denotes Favre averaging [51], (\( \cdot \)) denotes non-weighted averaging and (\( \overline{\cdot} \)) denotes an average quantity that is neither a Favre average nor a non-weighted average [74]. Here total energy is different than the Favre averaged total energy which contains turbulent kinetic energy.

\[
\bar{E} = \frac{\bar{p}}{\bar{\rho}(\gamma - 1)} + \frac{\tilde{u}_i \tilde{u}_i}{2} \tag{2.18}
\]

The Reynolds stresses appear in Eq. 2.16 and Eq. 2.17 are defined as follows:

\[
\tau_{ij} = \frac{1}{T} \int^{t+\Delta t}_t \rho \tilde{u}_i \tilde{u}_j \, d\tau \tag{2.19}
\]

where \( u'_i \) are the fluctuating velocity components. The source term \( S \) in the mean energy equation (2.16) is [75]:

\[
S = - \left( P - \bar{p} \varepsilon + p \frac{\partial \bar{u}_i}{\partial x_i} \right) + \frac{\partial}{\partial x_i} \left( \rho u_i \frac{\partial u_i}{\partial x_i} \right) + \left( - \bar{p} \delta_{ij} + \bar{T}_{ij} \right) \frac{\partial \bar{u}_i}{\partial x_j} \tag{2.20}
\]

where \( P \) is the turbulence kinetic energy production, \( \varepsilon \) is the turbulent dissipation rate and \( P' \) is the fluctuating static pressure. The mean viscous stresses and laminar heat transfer rates are approximated as follows [76]:

\[
\bar{T}_{ij} \approx \mu \left( \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right) - \frac{2}{3} \mu \frac{\partial \tilde{u}_k}{\partial x_k} \delta_{ij} \tag{2.21}
\]

\[
\bar{q}_i \equiv - \kappa \frac{\partial \bar{T}}{\partial x_i} \tag{2.22}
\]
The turbulent heat transfer rates are approximated using a simple gradient model [74].

\[
\overline{q}_{ri} \approx -\frac{\mu_r C_p}{Pr_r} \frac{\partial T}{\partial x_i}
\]

2.23

In the present work, turbulent viscosity \( \mu_r \) is computed in the same way as it is done in a \( k-\varepsilon \) model.

\[
\mu_r = C_\mu \bar{\mu} Re_r
\]

2.24

\[
Re_r = \frac{\bar{\rho} k^2}{\bar{\mu} \varepsilon}
\]

2.25

where the coefficient \( C_\mu \) is proposed by Launder and Sharma as [77]:

\[
C_\mu = 0.09 \exp \left[\frac{-2.5}{(1 + 0.02 Re_r)}\right]
\]

2.26

Here \( C_p \) is the specific heat at constant pressure, \( Pr_r \) is the turbulent Prandtl number. In this work, turbulent Prandtl number is taken as 0.9 to obtain the correct recovery temperature for turbulent flow over an adiabatic wall [73].

### 2.4 Reynolds Stress Transport Equations

The averaging approach followed for the derivation of the Favre-Reynolds averaged Navier-Stokes equations introduced a new set of unknowns, the Reynolds stresses, which need to be obtained using additional equations. In eddy viscosity models, the Reynolds stress tensor is obtained using the Boussinesq approximation (Eq. 1.1). In Reynolds Stress Models, model transport equations are solved for Reynolds stresses and
for the dissipation rate. The Favre-Reynolds averaged transport equations for Reynolds stresses obtained from Navier-Stokes equations are [78]:

\[
\frac{\partial \tau_{ij}}{\partial t} + \frac{\partial}{\partial x_k} \left( \tau_{ij} \frac{\partial \bar{u}_k}{\partial x_k} \right) = \frac{\partial}{\partial x_k} \left( - \frac{\bar{p}}{\rho} \bar{u}_i \bar{u}_j \bar{u}_k - p \bar{u}_i' \delta_{ik} - p u'_i \delta_{jk} + u'_i t_{jk} + u'_j t_{ik} \right) + \left( - \tau_{ik} \frac{\partial \bar{u}_j}{\partial x_k} - \tau_{jk} \frac{\partial \bar{u}_i}{\partial x_k} \right) + \frac{2}{3} \frac{\partial}{\partial x_k} p \frac{\partial \bar{u}_i'}{\partial x_k} \delta_{ij} - \left( t_{ik} \frac{\partial \bar{u}_j}{\partial x_k} + t_{jk} \frac{\partial \bar{u}_i}{\partial x_k} \right) + \frac{2}{3} \frac{\partial}{\partial x_k} \frac{\partial \bar{u}_k'}{\partial x_k} \delta_{ij} + p \left( - \bar{u}_i \frac{\partial \bar{p}}{\partial x_j} - \bar{u}_j \frac{\partial \bar{p}}{\partial x_i} + \bar{u}_i \frac{\partial \bar{\tau}_{jk}}{\partial x_k} + \bar{u}_j \frac{\partial \bar{\tau}_{ik}}{\partial x_k} \right) \]

In Eq. 2.27, the convection and production terms are exact. The modeling of the remaining terms is discussed in the following sections. According to Morkovin’s hypothesis [51], for non-hypersonic flows the effect of density fluctuations on turbulence are small provided they change slowly relative to the mean density. Therefore in the present work, density fluctuation effects \( K_{ij} \), pressure dilatation, and pressure diffusion terms are neglected:

\[
K_{ij} \approx 0, \quad p \frac{\partial \bar{u}_i'}{\partial x_i} \approx 0, \quad p u'_i \approx 0, \quad \bar{u}_i \approx 0 \]
As a result, the source term $S$ in Eq. 2.20 reduces to:

$$S \approx -(P - \bar{p} \varepsilon) \quad 2.29$$

### 2.4.1 Diffusion Term

The diffusion term in Eq. 2.27 is approximated by:

$$d_i \approx \frac{\partial}{\partial x_k} \left[ -\bar{p} \bar{u}_i \bar{u}_j \bar{u}_k + \nu \frac{\partial \tau_{ij}}{\partial x_k} \right] - 2.30$$

where the triple correlations are modeled by Daly and Harlow as [79]

$$-\bar{p} \bar{u}_i \bar{u}_j \bar{u}_k \approx -C_s \frac{k}{\bar{p}} \varepsilon \left[ \tau_{kl} \frac{\partial \tau_{ij}}{\partial x_l} \right] \quad 2.31$$

with $C_s = 0.22$. Here $k$ is the turbulent kinetic energy and $\varepsilon$ is the turbulent dissipation rate.

$$k = \frac{1}{2 \bar{p}} \langle \tau_{ii} \rangle \quad 2.32$$

### 2.4.2 Dissipation Term

The dissipation term is modeled by a local isotropy assumption [80].

$$\bar{p} \varepsilon_{ij} \approx \frac{2}{3} \bar{p} \varepsilon \delta_{ij} \quad 2.33$$
Although this approximation was shown to be in-correct in the near-wall region [80], anisotropy effects are included in the redistribution terms by modifying the model coefficients. This approach is suggested by Lumley [81] to keep the number of empirical functions to be optimized to a minimum.

### 2.4.3 Redistribution Term

The redistribution term is the most important item in the closure because it controls both the separation and reattachment processes [73,74,82,83]. It was shown that there are two distinct kinds of interaction which appear in the redistribution correlation; one involving fluctuating quantities (slow terms) and another arising from the presence of mean strain rate (rapid terms) [54]. The slow part is modeled by a simple quasilinear return to isotropy model originally proposed by Rotta (1951).

\[
\phi_{i,j,1} \cong -C_1 \bar{\rho} \varepsilon a_{ij}
\]

The coefficient \( C_1 \) is optimized by Launder and Shima to include the dissipation tensor anisotropy [80].

\[
C_1 = 1 + 2.58 A_2 A_2^{0.25} \left\{ 1 - \exp \left[ -\left( \frac{\text{Re}_T}{150} \right)^2 \right] \right\}
\]
where,

\[
A = 1 - \frac{9}{8} [A_2 - A_3] \quad 2.36
\]

\[
A_2 = a_{ik} a_{ki} \quad 2.37
\]

\[
A_3 = a_{ij} a_{ij} \quad 2.38
\]

\[
a_{ij} = \frac{\tau_{ij}}{\rho k} - \frac{2}{3} \delta_{ij} \quad 2.39
\]

Launder et al. [54] expressed the complete influence of the mean strain rate on the redistribution correlation as follows:

\[
\phi_{y,2} \simeq -\frac{C_2'}{11} + 8 \left( P_{yj} - \frac{1}{3} P_{mn} \delta_{yj} \right) - \frac{30 C'_2 - 2}{55} k \left[ \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} - \frac{2}{3} \frac{\partial \tilde{u}_k}{\partial x_k} \delta_{ij} \right]
\]

\[
- \frac{8 C'_2 - 2}{11} \left( D_{yj} - \frac{1}{3} P_{mn} \delta_{yj} \right) \quad 2.40
\]

where \( C'_2 \) is a constant, \( P_{yj} \) is the production term in Eq. 2.27 and \( D_{yj} \) is:

\[
D_{yj} = -\tau_{jk} \frac{\partial \tilde{u}_k}{\partial x_j} - \tau_{jk} \frac{\partial \tilde{u}_k}{\partial x_j} \quad 2.41
\]

It was shown that the first term on the right hand side of Eq. 2.40 is the dominant term hence a simpler expression can be written as [42]:

\[
\phi_{y,2} \simeq -C_2 \left( P_{yj} - \frac{1}{3} P_{mn} \delta_{yj} \right) \quad 2.42
\]

where \( C_2 \) is different in magnitude from the coefficient of the first term in Eq. 2.40 to compensate in part for the neglected terms. \( C_2 \) is optimized by Launder and Shima [80]:

\[
C_2 = 0.75 \cdot \sqrt{A} \quad 2.43
\]
Although the coefficients of slow and rapid terms have been optimized to include the dissipation tensor anisotropy in the near-wall regions, wall correction terms must still be added to the expressions to compensate for the reflection of pressure fluctuations from the rigid wall. Launder, Reece and Rodi [54] have shown that there should be two contributions to the near-wall effect, corresponding to the reflected wall-influence of slow and rapid terms. The models for slow [84] and rapid [85] wall echo terms are displayed in the following equations:

\[
\phi_{y,1}^{w} \simeq C_1^{w} \frac{\varepsilon}{k} \left[ \tau_{km} n_k n_m \delta_{ij} - \frac{3}{2} \tau_{kl} n_k n_l \right] f \\
\phi_{y,2}^{w} \simeq C_2^{w} \left[ \phi_{km,2} n_k n_m \delta_{ij} - \frac{3}{2} \phi_{kl,2} n_k n_l \right] f 
\]

where \( \boldsymbol{n} \) is the surface normal. The near wall damping function \( f \) is taken as \( 0.4 k^{3/2} / y_n^{1/2} \), \( y_n \) being the normal distance to the wall. The coefficients \( C_1^{w} \) and \( C_2^{w} \) are optimized by Launder and Shima [80].

\[
C_1^{w} = -\frac{2}{3} C_1 + 1.67 \\
C_2^{w} = \max \left( \frac{2}{3} - \frac{1}{6 C_2} \right) 0 
\]

Gerolymos et al [73,74] suggested that for the computation of three-dimensional complex flows it is important to use models independent of geometric parameters such as the distance from the wall or the surface normal. They proposed an alternative approach of using pseudo-normals instead of surface normals. The echo terms are computed in the usual way without the wall damping function \( f \).
To construct the wall-echo terms, the surface normals are approximated by the gradient of a function of the turbulence length scale. The effect of the distance from the wall is included in the coefficients $C_1^w$ and $C_2^w$.

\[
\phi_{ij,1}^w \approx C_1^w \frac{\tau_{km} n_k n_m \delta_{ij} - \frac{3}{2} \tau_{k\ell} n_k n_j - \frac{3}{2} \tau_{j\ell} n_k n_i}{k} 
\]

\[
\phi_{ij,2}^w \approx C_2^w \left[ \phi_{km,2} n_k n_m \delta_{ij} - \frac{3}{2} \phi_{k\ell,2} n_k n_j - \frac{3}{2} \phi_{j\ell,2} n_k n_i \right] 
\]

Coefficients $C_1^w$ and $C_2^w$ were optimized by Gerolymos and Vallet [73]:

\[
n_i \frac{\partial \tilde{e}_j}{\partial n_i} = \left[ \nabla l_n \right] \frac{\nabla l_n}{\left| \nabla l_n \right|} 
\]

\[
l_i \left\{ 1 - \exp \left[ - \frac{\left( \text{Re}_T \right)}{30} \right] \right\} 
\]

\[
l_n = \frac{1}{1 + 2\sqrt{A_2} + 2A_2^{16}} 
\]

\[
l_i = \frac{k^{3/2}}{\varepsilon} 
\]

\[
C_1^w = \left[ -\frac{2}{3} C_1 + 1.67 \right] \left| \nabla l_n^w \right| 
\]

\[
C_2^w = \max \left[ \frac{2}{3} - \frac{1}{6C_2}, 0 \right] \left| \nabla l_n^w \right| 
\]

where

\[
l_i \left\{ 1 - \exp \left[ - \frac{\left( \text{Re}_T \right)}{30} \right] \right\} 
\]

\[
l_1^w = \frac{1}{1 + 2A_2^{0.8}} 
\]

\[
l_2^w = \frac{1}{1 + 1.8A_2^{\max(0.6, A)}} 
\]
In the present work Eq. 2.48 and Eq. 2.49 are selected as wall-echo terms along with the coefficients given in equations Eq. 2.53 and Eq. 2.54.

### 2.5 Turbulence Dissipation Rate Equation

The models employed for the non-exact terms of the Reynolds stress transport equations introduced a new unknown $\varepsilon$, the turbulence dissipation rate. Hence one more equation is needed to complete the closure. The differential equation for the evolution of turbulence dissipation rate is written as [86]:

\[
\frac{\partial \bar{\varepsilon}}{\partial t} + \frac{\partial \bar{\varepsilon} \bar{u}_i}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ (\bar{\mu} \delta_{ij} + C_\varepsilon \frac{k}{\varepsilon} \tau_{ij}) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{\varepsilon 1} \frac{P \varepsilon}{k} - C_{\varepsilon 2} \frac{\varepsilon^2}{k} \tag{2.57}
\]

where

\[
P = \frac{1}{2} P_n \tag{2.58}
\]

The first term on the right hand side of Eq. 2.57 represents the diffusive transport of $\varepsilon$ while the second and third terms represent the net effect of generation of $\varepsilon$ due to vortex stretching and its destruction by viscosity [54]. The values of the coefficients $C_\varepsilon$ and $C_{\varepsilon 2}$ are 0.18 and 1.9 respectively. $C_{\varepsilon 1}$ is computed as follows [80]:

\[
C_{\varepsilon 1} = 1.45 + \psi_1 + \psi_2 \tag{2.59}
\]

\[
\psi_1 = 2.5 \left( \frac{P}{\varepsilon} - 1 \right) \tag{2.60}
\]

\[
\psi_2 = 0.3 (1 - 0.3 \Delta \varepsilon) \exp(-0.002 \text{Re}_T) \tag{2.61}
\]
Launder et. al. [80] introduced $\psi_1$ to prevent too small values of turbulence dissipation rate in near-wall separated flows and $\psi_2$ to prevent too rapid decay toward laminar flow in strong accelerations.

Launder and Sharma [77] proposed an alternative equation for the modified dissipation rate $\varepsilon^*$, which vanishes at the wall, to avoid the numerical instabilities that might occur because of non-zero dissipation at the wall.

$$
\varepsilon^* = \varepsilon - 2\frac{\mu}{\rho} \left| \nabla \left( \sqrt{k} \right) \right|^2 
$$

$$
\frac{\partial \rho \varepsilon^*}{\partial t} + \frac{\partial \rho \varepsilon^* u_i}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu \delta_{ij} - C_{\varepsilon} \frac{k}{\varepsilon} \tau_{ij} \right) \frac{\partial \varepsilon^*}{\partial x_j} \right] + C_{\varepsilon 1} \frac{\varepsilon^*}{k} - C_{\varepsilon 2} \frac{\varepsilon^{*2}}{k} + 2\frac{\mu_l}{\rho} \left( \nabla \tilde{\nabla} \right)^2 
$$

Although the modified dissipation rate provides homogeneous boundary conditions at the wall, $\varepsilon^*$ varies very rapidly near the wall. It might be difficult to numerically resolve this growth with sufficient accuracy and the solutions may not be grid independent near the wall. These problems are alleviated to some extent by using an equation for $\varepsilon$ since the gradients of $\varepsilon$ are much smaller compared to $\varepsilon^*$ [86]. In the present work Eq. 2.57 is selected to close the system. The selected pressure redistribution model and turbulence dissipation rate equation for this dissertation were shown to yield better correlations when compared to several other second-order models [87].
Chapter 3

Flow Solvers

3.1 Flow Solver PUMA2

The flow solver PUMA2 (Parallel Unstructured Maritime Aerodynamics-2) is a computer program for the analysis of external, non-reacting compressible flows over three-dimensional geometries. The original version of the code (PUMA) was written by Dr. Christopher W. S. Bruner as a part of his doctoral thesis [88]. It was an inviscid Euler code. PUMA2 is written in ANSI C and uses MPI libraries for message transfer between the processors. The code solves the full Navier-Stokes equations using a finite volume technique and supports several unstructured grid topologies such as tetrahedrons, wedges, pyramids, and hexahedrons. It uses dynamic memory allocation; therefore, the problem size is limited by the amount of memory available on the machine [89]. Numerous improvements have been made to PUMA2 by Long et al. [1, 15, 27, 34, 57-66]

3.1.1 Finite Volume Formulation

The integral form of the Navier-Stokes equations is written as:

$$\frac{\partial}{\partial t} \int_{\Omega} \mathbf{q} \, d\Omega + \oint_{s} \mathbf{F} \cdot d\mathbf{S} - \oint_{s} \mathbf{F}_r \cdot d\mathbf{S} = 0$$

3.1
where \( \mathbf{q} \) is the vector containing the conservative variables, \( \mathbf{F} \) is the convective flux tensor and \( \mathbf{F}_v \) is the viscous flux tensor.

\[
\mathbf{q} = \begin{bmatrix} \rho & \rho u & \rho v & \rho w & \rho E \end{bmatrix}^T
\]

\[
\mathbf{F} = \begin{bmatrix} \rho u & \rho v & \rho w \\ \rho u^2 + p & \rho uv & \rho uw \\ \rho uv & \rho v^2 + p & \rho vw \\ \rho uw & \rho vw & \rho w^2 + p \\ \rho uH & \rho vH & \rho wH \end{bmatrix}
\]

\[
\mathbf{F}_v = \begin{bmatrix} t_{xx} & t_{xy} & t_{xz} \\ t_{xy} & t_{yy} & t_{yz} \\ t_{xz} & t_{yz} & t_{zz} \\ (t_{xx}u + t_{xy}v) & (t_{xy}u + t_{yy}v) & (t_{xz}u + t_{yz}v) \\ + t_{xx}w - q_x & + t_{xy}w - q_y & + t_{xz}w - q_z \end{bmatrix}
\]

### 3.1.1.1 Spatial Discretization

PUMA2 is based on a finite volume discretization which breaks up the physical domain into control volumes (or cells). For a given cell \( i \) of volume \( \Omega_i \) and \( S_j \) being the surface areas delimiting its volume, Eq. 3.1 becomes:

\[
\Omega_i \frac{\partial \mathbf{q}}{\partial t} = - \sum_{j=1}^{N_{faces}} \left( \mathbf{F}(\mathbf{q}) \cdot \mathbf{n} S \right)_j + \sum_{j=1}^{N_{faces}} \left( \mathbf{F}_v(\mathbf{q}) \cdot \mathbf{n} S \right)_j = \mathbf{R}(\mathbf{q})
\]

The right hand side of the equation is called the residual vector. In the code solutions are stored at the cell centers. In order to compute fluxes through each face of the cell, one needs the reconstruct solutions at the face centers. For this purpose the solution at a face is expressed as a function flow states on each side of the face:
3.1.1.1 Spatial Discretization of Convective Fluxes

Due to the hyperbolic nature of the convective fluxes, upwinding schemes are necessary for the discretizations [41]. In this work Roe’s scheme is employed for this purpose [90].

\[
\left( \overline{F}(q) \cdot n \right)_{\text{face}} = \frac{1}{2} \left[ \overline{F}(q_L) \cdot n + \overline{F}(q_R) \cdot n - \Delta \overline{F}_1 \cdot n - \Delta \overline{F}_2 \cdot n - \Delta \overline{F}_3 \cdot n \right]  
\]

where

\[
\Delta \overline{F}_1 \cdot n = \left| V_a \right| \left( \begin{array}{c}
\frac{(\Delta \rho - \frac{\Delta \rho}{a_{roe}^2})}{2} \\
\frac{u_{roe}^2}{2} + v_{roe}^2 + w_{roe}^2 \\
\end{array} \right) \left( \begin{array}{c}
1 \\
u_{roe} \\
v_{roe} \\
w_{roe} \\
\frac{u_{roe}^2 + v_{roe}^2 + w_{roe}^2}{2} \\
\end{array} \right) 
\]

\[
\Delta \overline{F}_2 \cdot n = \left| V_a \right| \left( \begin{array}{c}
\Delta u - n_x \Delta V \\
\Delta v - n_y \Delta V \\
\Delta w - n_z \Delta V \\
\end{array} \right) 
\]

\[
\Delta \overline{F}_3 \cdot n = \left| V_a \right| \left( \begin{array}{c}
u_{roe} \Delta u + v_{roe} \Delta v + w_{roe} \Delta w - V_n \Delta V \\
\end{array} \right) 
\]
\[ \Delta \bar{F}_2 \cdot \mathbf{n} = \left| V_n + a_{\text{roe}} \left( \frac{\Delta p + \rho_{\text{roe}} a_{\text{roe}} \Delta V}{2a_{\text{roe}}^2} \right) \right| \begin{bmatrix} 1 \\ u_{\text{roe}} + n_x a_{\text{roe}} \\ v_{\text{roe}} + n_y a_{\text{roe}} \\ w_{\text{roe}} + n_z a_{\text{roe}} \\ H_{\text{roe}} + V_{\text{ave}} a_{\text{roe}} \end{bmatrix} \]

\[ \Delta \bar{F}_3 \cdot \mathbf{n} = \left| V_n - a_{\text{roe}} \left( \frac{\Delta p - \rho_{\text{roe}} a_{\text{roe}} \Delta V}{2a_{\text{roe}}^2} \right) \right| \begin{bmatrix} 1 \\ u_{\text{roe}} - n_x a_{\text{roe}} \\ v_{\text{roe}} - n_y a_{\text{roe}} \\ w_{\text{roe}} - n_z a_{\text{roe}} \\ H_{\text{roe}} - V_{\text{ave}} a_{\text{roe}} \end{bmatrix} \]

with

\[ \rho_{\text{roe}} = \sqrt{\rho_L \rho_R} \]

\[ u_{\text{roe}} = \frac{u_L + u_R \sqrt{\rho_R \rho_L}}{1 + \sqrt{\rho_R \rho_L}} \]

\[ v_{\text{roe}} = \frac{v_L + v_R \sqrt{\rho_R \rho_L}}{1 + \sqrt{\rho_R \rho_L}} \]

\[ w_{\text{roe}} = \frac{w_L + w_R \sqrt{\rho_R \rho_L}}{1 + \sqrt{\rho_R \rho_L}} \]

\[ H_{\text{roe}} = \frac{H_L + H_R \sqrt{\rho_R \rho_L}}{1 + \sqrt{\rho_R \rho_L}} \]

\[ a_{\text{roe}} = (\gamma - 1) \left[ H_{\text{roe}} - \frac{u_{\text{roe}}^2 + v_{\text{roe}}^2 + w_{\text{roe}}^2}{2} \right] \]
3.1.1.1.2 Spatial Discretization of Viscous Fluxes

Viscous fluxes are parabolic partial differential equations in nature. Therefore, they can be computed using a space centered scheme. The laminar stress tensor is defined using Eq. 2.6 and the laminar heat transfer rate is given in Eq. 2.7. Computation of viscous fluxes through each cell face requires the computation of velocity derivatives and temperature gradient at the cell faces. For a given cell $i$ of volume $\Omega_i$ and $S_j$ being the surface areas delimiting its volume, gradient of a velocity component at the cell center is computed using the Divergence Theorem [91].

$$\nabla u = \frac{1}{\Omega_i} \sum_{j=1}^{N_{\text{faces}}} (u_{\text{face}} \cdot n_j) S_j$$

where the values of the velocity components at each face are computed using the volume weighted average of the neighboring cell values at each side of that face.

$$u_{\text{face}} = \frac{u_{\text{cell0}} \Omega_{\text{cell0}} + u_{\text{cell1}} \Omega_{\text{cell1}}}{\Omega_{\text{cell0}} + \Omega_{\text{cell1}}}$$
Derivatives of velocity components at each face are obtained using the same approach given in Eq. 3.25. The temperature gradient at each cell is computed from the equation of state for a perfect gas.

\[ p = \rho RT \]  \hspace{1cm} 3.26
\[ \nabla T = \frac{1}{\rho R} \left( \nabla p - RT \nabla \rho \right) \]  \hspace{1cm} 3.27

### 3.1.1.2 Time Integration

The resulting semi-discrete equations (Eq. 3.5) can be solved using explicit or implicit methods implemented in PUMA2. An m-stage Jameson-style [92, 93, 94, 95] Runge-Kutta explicit scheme with either two or four stages can be used for time-accurate and steady state solutions. A major limitation of this explicit scheme is the maximum allowable CFL (Courant-Friedrichs-Lewy) number of \( 2\sqrt{2} \), which places a great constraint on the time step. The major advantage of this method is that it only requires the storage of two sets of solutions at a time. The details of a four stage Runge-Kutta scheme is given in Eq. 3.28 where \( \Delta t \) is the time step and \( \Omega \) is the volume of the cell.

\[
\begin{align*}
q^0 &= q^n \\
q^1 &= q^0 + \frac{\alpha_1 \Delta t}{\Omega} R(q^0) \\
q^2 &= q^0 + \frac{\alpha_2 \Delta t}{\Omega} R(q^1) \\
q^3 &= q^0 + \frac{\alpha_3 \Delta t}{\Omega} R(q^2) \\
q^4 &= q^0 + \frac{\alpha_4 \Delta t}{\Omega} R(q^3) \\
q^{n+1} &= q^4
\end{align*}
\]  \hspace{1cm} 3.28
The coefficients $(\alpha_1, \alpha_2, \alpha_3, \alpha_4)$ are set to $\left(\frac{1}{4}, \frac{1}{3}, \frac{1}{2}, 1\right)$, respectively.

Implicit time marching methods available in PUMA2, namely Jacobi and Symmetric Successive Overrelaxation (SSOR), allow much larger CFL numbers to be used when fast convergence to steady state is needed. These routines split the matrix of the system of equations (Eq. 3.5) into diagonal, lower diagonal and upper diagonal matrices. The diagonal matrix is inverted while the lower and upper diagonal matrices are moved into the residue of the equation. The difference between Jacobi and SSOR schemes lies in the time at which the most recently updated data are used. The Jacobi method uses the data computed on the previous time step to update the solution. On the other hand, SSOR uses the latest values of the vector of unknowns. Although these methods allow larger time steps than the explicit methods, they are however much more expensive to use in terms of memory use [15,59,65].

3.1.1.2.1 Time Step Computation

Time integration requires an appropriate time step $\Delta t$, limited by the stability bound of the scheme. In PUMA2 the local time step is computed through a stability analysis for the inviscid and viscous flux operators separately. The time step size for a cell $i$ is written as:

$$\Delta t = CFL \times \min[(\Delta t)_{inv}, (\Delta t)_{vis}]$$  \hspace{1cm} 3.29
where

\[
(\Delta t)_{inv} = \Omega_i \sum_{j=1}^{Nfaces} \frac{1}{(c + \mathbf{V} \cdot \mathbf{n}) S_j}
\]

\[
(\Delta t)_{vis} = \frac{0.5 \rho \Pr}{\gamma \mu} \left[ \frac{\Omega_i}{\sum_{j=1}^{Nfaces} S_j} \right] \]

In Eq. 3.30 and Eq. 3.31 \( \mathbf{V} \) is the velocity vector, \( c \) is the local speed of sound and \( \gamma \) is 1.4 for air.

### 3.1.2 Boundary Conditions

The numerical solutions also require appropriate boundary conditions at the relevant boundaries. The boundary conditions implemented in PUMA2 are listed in Table 3-1.

<table>
<thead>
<tr>
<th>Type</th>
<th>Description</th>
<th>Specified Value(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Fixed at freestream</td>
<td>None</td>
</tr>
<tr>
<td>2</td>
<td>Fixed at given values</td>
<td>( \rho, u, v, w, p )</td>
</tr>
<tr>
<td>3</td>
<td>First order interpolation from interior</td>
<td>None</td>
</tr>
<tr>
<td>4</td>
<td>Second order interpolation from interior</td>
<td>None</td>
</tr>
<tr>
<td>5</td>
<td>Farfield</td>
<td>None</td>
</tr>
<tr>
<td>6</td>
<td>Specified total pressure and temperature</td>
<td>( p_0, T_0 )</td>
</tr>
<tr>
<td>7</td>
<td>Fixed back pressure</td>
<td>Exit pressure</td>
</tr>
<tr>
<td>8</td>
<td>Tangency (inviscid solid wall)</td>
<td>None</td>
</tr>
<tr>
<td>9</td>
<td>No slip, adiabatic wall</td>
<td>None</td>
</tr>
<tr>
<td>10</td>
<td>No slip, fixed wall temperature</td>
<td>( T_{wall} )</td>
</tr>
<tr>
<td>11</td>
<td>Actuator disc</td>
<td>Rotor thrust or blade pitch angle</td>
</tr>
<tr>
<td>17</td>
<td>Zero u-velocity</td>
<td>None</td>
</tr>
<tr>
<td>18</td>
<td>Zero v-velocity</td>
<td>None</td>
</tr>
<tr>
<td>19</td>
<td>Zero w-velocity</td>
<td>None</td>
</tr>
</tbody>
</table>
In PUMA2 farfield boundary conditions are based on the one-dimensional Riemann invariants [96]. For a subsonic farfield, the fixed and the extrapolated Riemann invariants are defined as:

\[
R_\infty = V_\infty - \frac{2c_e}{\gamma - 1} \quad 3.32
\]

\[
R_e = V_e - \frac{2c_e}{\gamma - 1} \quad 3.33
\]

where subscripts \(\infty\) and \(e\) denote freestream values and the values extrapolated from the interior cells, and let \(V_n\) and \(c\) be the velocity component normal to the boundary and the local speed of sound, respectively.

These invariants are used to obtain the actual normal velocity component and the speed of sound at the farfield boundary:

\[
V_n = \frac{1}{2}(R_e + R_\infty) \quad 3.34
\]

\[
c = \frac{1}{2}(\gamma - 1)(R_e - R_\infty) \quad 3.35
\]

At the outflow boundary tangential velocity components and entropy are extrapolated from the interior cells while at an inflow boundary they are specified as having farfield values.

Actuator disc boundary conditions are implemented in the code to model the antitorque system of RAH-66 Comanche helicopter. The details are given in the next section.
3.1.3 FANTAIL™ Modeling

The antitorque system of an RAH-66 Comanche helicopter is a ducted fan called the FANTAIL™ [30] which can be seen in Figure 3-1.

In this study, the FANTAIL™ is modeled as an actuator disk with zero thickness. Its effects are introduced into the flow as boundary conditions in which the pressure undergoes a discontinuity while the other flow parameters remain continuous. Momentum theory with uniform loading [18] and blade element theory [33] are used for the fain-in-fin model.

Figure 3-1: RAH-66 Comanche FANTAIL™
3.1.3.1 Momentum Theory with Uniform Loading

In this approach fan thrust is set as an input and the uniform pressure jump is computed as follows:

\[ \Delta p = \frac{T_{\text{fan}}}{A_{\text{disk}}} \]  \hspace{1cm} (3.36)

where \( T_{\text{fan}} \) is the fan thrust and \( A_{\text{disk}} \) is the rotor disk area.

\[ A_{\text{disk}} = \pi \left( R^2 - R_{cb}^2 \right) \]  \hspace{1cm} (3.37)

In Eq. 3.37, \( R_{cb} \) is the radius of the center body and \( R \) is the radius of the duct.

3.1.3.2 Blade Element Theory

In this approach the collective pitch angle of the blades is set as an input and the corresponding fan thrust is computed using blade element theory, which relates the local lift on a differential element of the blade to the local velocity and the local blade pitch. The external velocity vector is decomposed into a component normal to the disk, \( V_N(r,\psi) \), and a component in the plane of the disk, \( V_T(r,\psi) \). A schematic of the blade element and the corresponding velocities and forces can be seen in Figure 3-2. The lift coefficient is assumed to be a unique function of angle of attack by neglecting the Mach number and Reynolds number effects. The resulting equations for local angle of attack and local section lift are:
An often used approximation is that the linear lift coefficient is given by 

\[ C_l = a \alpha \]

where \( a = 5.73 \text{ rad}^{-1} \) (0.1 deg\(^{-1}\)). This lift curve slope value is slightly lower than the theoretical value of \( 2\pi \) (Thin Airfoil Theory) but it is sufficiently accurate for many purposes [97]. Computations also do not include any stall model for blade section lift. The blade pitch is typically approximated by linear twist with respect to the blade pitch at 75% radius. The fan has eight blades; each has a twist angle of 7 degrees.

\[ \theta(r) = \theta_0 + \theta_1 \left( \frac{r}{R} - 0.75 \right) \]

In the usual simplified theory, the blade element expressions for lift, or more commonly the integrated thrust and moments, are used as inputs to momentum theory to estimate the external flow velocities, \( V_N(r, \psi) \) and \( V_T(r, \psi) \). In the present work, momentum theory is replaced by a direct numerical simulation of the external flow. Also the individual blades are not included in the definition of the body boundary. Instead, an
actuator disk is used at the nominal plane of the rotor disk to apply pressure-jump boundary conditions to the flow solution.

While the actual pressure jump at a given location on the disk will vary between zero (when no blade is present) and the maximum pressure difference over the blade chord, a useful approximation is to take the average pressure jump during a single blade passage time:

\[
\Delta p(r, \psi) = \frac{1}{T} \int_0^T \Delta p_{\text{blade}}(t) \cdot dt
\]

where

\[
T = \frac{2 \pi}{N_b \Omega}
\]

Here \( N_b \) is the number of blades. Now, as noted above, \( \Delta p_{\text{blade}} \) will vanish except in those parts of the rotor revolution where some portion of the blade is located at the given station. During these times the blade is moving with a speed of \( \Omega r \), so the variable of integration can be replaced using \( \Omega r dt = dx \):

\[
\Delta p(r, \psi) = N_b \frac{\Omega}{2\pi} \int_0^c \Delta p_{\text{blade}}(x) \frac{1}{\Omega r} \cdot dx
\]

Since the integration of blade pressure over the chord is simply the local lift per unit span, the integral can be replaced in favor of the local lift:

\[
\Delta p(r, \psi) = N_b \frac{l(r, \psi)}{2\pi \cdot r}
\]
Given either a lift coefficient look-up table or linear lift-curve slope and a definition of twist, equations 3.38, 3.39, 3.40, 3.44 form a complete set of algebraic equations to compute the pressure jump across the disk as a function of computed external flow and the applied collective pitch input ($\theta_0$).

### 3.2 Flow Solver PUMA_RSM

The in-house computational fluid dynamics code PUMA_RSM is a modified version of PUMA2. It has been designed to perform turbulent flow simulations using RSM. The model consists of coupling the transport equations for Reynolds stresses and turbulence dissipation rate with the Favre-Reynolds averaged Navier-Stokes equations, which results in a system of twelve transport equations. Therefore; PUMA2, which was designed to solve for five flow variables, is modified to handle twelve variables. Additional subroutines are introduced for the additional turbulence terms.

#### 3.2.1 Finite Volume Formulation

Favre averaging applied to the transport equations makes it possible to predict the variations in mean density without explicitly modeling the parts related to density variations. Although the Reynolds stress transport equations are written for time averaged quantities, the method can still be used for unsteady flow simulations when the characteristic frequencies of the phenomenon are sufficiently low compared to the
characteristic frequencies of turbulence [83]. The integral form of the governing
equations (RSM) can be written as:

\[
\frac{\partial}{\partial t} \int_{\Omega} q \, d\Omega + \oint_{s} \overline{F} \cdot dS - \oint_{s} \overline{F}_{\nu} \cdot dS - \oint_{s} \overline{F}_{r} \cdot dS - \int_{\Omega} Q \, d\Omega = 0
\] 3.45

where \( q \) is the vector containing the conservative variables, \( \overline{F} \) is the convective flux
tensor, \( \overline{F}_{\nu} \) is the laminar viscous flux tensor, \( \overline{F}_{r} \) is the turbulent flux tensor and \( Q \) is the
vector containing turbulent source terms. The conservative variable vector \( q \) can be
written as:

\[
q = [\rho \quad \rho \tilde{u} \quad \rho \tilde{v} \quad \rho \tilde{w} \quad \rho \tilde{E} \quad \tau_{xx} \quad \tau_{yy} \quad \tau_{zz} \quad \tau_{xy} \quad \tau_{xz} \quad \tau_{yz} \quad \rho \tilde{e}]^T
\] 3.46

Convective and laminar viscous flux tensors, which contain exact terms, are
given by the following relations:

\[
\overline{F} = \begin{bmatrix}
\rho \tilde{u} \\
\rho \tilde{u} \tilde{u} + \tilde{p} \\
\rho \tilde{u} \tilde{v} \\
\rho \tilde{u} \tilde{w} \\
\rho \tilde{u} \tilde{H} \\
\tilde{u} \tau_{xx} \\
\tilde{u} \tau_{xy} \\
\tilde{u} \tau_{xz} \\
\tilde{u} \tau_{yz} \\
\tilde{u} \tilde{p} \tilde{e}
\end{bmatrix}
\]

3.47
Turbulent flux tensor and turbulent source vector are constructed in conjunction with the models employed for turbulent diffusion, dissipation and redistribution terms. According to the models selected in the previous sections, turbulent flux tensor and turbulent source vector are constructed as:

\[
\overline{F}_v = \begin{bmatrix}
0 & 0 & 0 \\
\tilde{t}_{xx} & \tilde{t}_{xy} & \tilde{t}_{xz} \\
\tilde{t}_{xy} & \tilde{t}_{yy} & \tilde{t}_{yz} \\
\tilde{t}_{xz} & \tilde{t}_{yz} & \tilde{t}_{zz} \\
\left(\tilde{t}_{xx}\tilde{u} + \tilde{t}_{xy}\tilde{v}\right) + \tilde{t}_{xz}\tilde{w} - \tilde{q}_x & \left(\tilde{t}_{xy}\tilde{u} + \tilde{t}_{yy}\tilde{v}\right) + \tilde{t}_{yz}\tilde{w} - \tilde{q}_y & \left(\tilde{t}_{xz}\tilde{u} + \tilde{t}_{yz}\tilde{v}\right) + \tilde{t}_{zz}\tilde{w} - \tilde{q}_z
\end{bmatrix}
\]

3.48

\[
\overline{F}_T = \begin{bmatrix}
0 & 0 & 0 \\
-\tau_{xx} & -\tau_{xy} & -\tau_{xz} \\
-\tau_{xy} & -\tau_{yy} & -\tau_{yz} \\
-\tau_{xz} & -\tau_{yz} & -\tau_{zz} \\
-\tau_{xx}\tilde{u} - \tau_{xy}\tilde{v} & -\tau_{xy}\tilde{u} - \tau_{yy}\tilde{v} & -\tau_{xz}\tilde{u} - \tau_{yz}\tilde{v} \\
\left(d_{xx}\right)_1 & \left(d_{xx}\right)_2 & \left(d_{xx}\right)_3 \\
\left(d_{xy}\right)_1 & \left(d_{xy}\right)_2 & \left(d_{xy}\right)_3 \\
\left(d_{xz}\right)_1 & \left(d_{xz}\right)_2 & \left(d_{xz}\right)_3 \\
\left(d_{yy}\right)_1 & \left(d_{yy}\right)_2 & \left(d_{yy}\right)_3 \\
\left(d_{yz}\right)_1 & \left(d_{yz}\right)_2 & \left(d_{yz}\right)_3 \\
\left(d_{z}\right)_1 & \left(d_{z}\right)_2 & \left(d_{z}\right)_3
\end{bmatrix}
\]

3.49
where

\[
\begin{align*}
(d_{ij})_n &= C_x \frac{k}{\bar{\rho} \varepsilon} \left[ \tau_{nm} \frac{\partial \tau_{ij}}{\partial x_m} \right] + \tilde{v} \frac{\partial \tau_{ij}}{\partial x_n} \quad 3.51 \\
(d_{\varepsilon})_n &= C_x \frac{k}{\varepsilon} \left[ \tau_{km} \frac{\partial \varepsilon}{\partial x_m} \right] + \tilde{v} \frac{\partial \varepsilon}{\partial x_n} \quad 3.52 \\
\phi_{ij} &= \phi_{ij,1} + \phi_{ij,2} + \phi_{ij}^w + \phi_{ij,2}^w \quad 3.53 \\
S_{\varepsilon} &= C_{x1} \frac{\varepsilon}{k} \tau_{ij} \frac{\partial \varepsilon}{\partial x_j} - C_{x2} \frac{\bar{\rho} \varepsilon^2}{k} \quad 3.54
\end{align*}
\]

### 3.2.1.1 Spatial Discretization

The discretized version of Equation (2.78) using the finite volume technique written for a cell \(i\) of volume \(\Omega_i\) and \(S_j\) being the surface areas delimiting its volume is:
Laminar viscous and turbulent fluxes are parabolic in nature. Therefore, they are computed using a similar approach described in section 3.1.1.2. The hyperbolic nature of the convective fluxes requires upwinding methods such as Roe’s scheme. In this work, Roe’s scheme described in section 3.1.1.1 and is extended for twelve equations. The details are given in the next section.

3.2.1.1 Spatial Discretization of Convective Fluxes

Roe’s scheme employed in this work has the same formulation given in Eq. 3.7 but with different components.

\[
\Omega_i \frac{\partial \mathbf{q}_i}{\partial t} = - \sum_{j=1}^{N_{\text{faces}}} \left( \mathbf{F}(\mathbf{q}) \cdot \mathbf{n}_j S_j \right) + \sum_{j=1}^{N_{\text{faces}}} \left( \mathbf{F}_T(\mathbf{q}) \cdot \mathbf{n}_j S_j \right) + \Omega_i \mathbf{Q}_i
\]

\[3.55\]

\[
\Delta \mathbf{F}_i \cdot \mathbf{n} = \left| \frac{\Delta \mathbf{p}}{\Delta \mathbf{u}_{\text{ro}}} \right| \left( \frac{1}{\Delta \mathbf{u}_{\text{ro}}} + \frac{2}{\Delta \mathbf{v}_{\text{ro}}} + \frac{2}{\Delta \mathbf{w}_{\text{ro}}} \right) + \frac{\Delta \mathbf{p}_{\text{ro}}}{\Delta \mathbf{u}_{\text{ro}}}
\]

\[3.56\]

\[
\begin{bmatrix}
\Delta \mathbf{u}_{\text{ro}} = \mathbf{u}_{\text{ro}} - \mathbf{u}_{\text{ro}} \\
\Delta \mathbf{v}_{\text{ro}} = \mathbf{v}_{\text{ro}} - \mathbf{v}_{\text{ro}} \\
\Delta \mathbf{w}_{\text{ro}} = \mathbf{w}_{\text{ro}} - \mathbf{w}_{\text{ro}}
\end{bmatrix}
\]

\[
\begin{bmatrix}
0 \\
\Delta \mathbf{u} - n_x \Delta \mathbf{V} \\
\Delta \mathbf{v} - n_y \Delta \mathbf{V} \\
\Delta \mathbf{w} - n_z \Delta \mathbf{V}
\end{bmatrix}
\]

\[
\begin{bmatrix}
\Delta \mathbf{u}_{\text{ro}} = \mathbf{u}_{\text{ro}} - \mathbf{u}_{\text{ro}} \\
\Delta \mathbf{v}_{\text{ro}} = \mathbf{v}_{\text{ro}} - \mathbf{v}_{\text{ro}} \\
\Delta \mathbf{w}_{\text{ro}} = \mathbf{w}_{\text{ro}} - \mathbf{w}_{\text{ro}}
\end{bmatrix}
\]

\[
\begin{bmatrix}
\Delta \mathbf{u}_{\text{ro}} = \mathbf{u}_{\text{ro}} - \mathbf{u}_{\text{ro}} \\
\Delta \mathbf{v}_{\text{ro}} = \mathbf{v}_{\text{ro}} - \mathbf{v}_{\text{ro}} \\
\Delta \mathbf{w}_{\text{ro}} = \mathbf{w}_{\text{ro}} - \mathbf{w}_{\text{ro}}
\end{bmatrix}
\]
The mean flow terms appearing in equations 3.56, 3.57 and 3.58 are computed using equations 3.11 through 3.23. Turbulent terms are obtained using the following relations:

\[
\Delta \vec{F}_2 \cdot \mathbf{n} = \vec{V}_n + \bar{a}_\text{ro} \left( \frac{\Delta \vec{p} + \bar{p}_\text{ro} \bar{a}_\text{ro} \Delta \vec{V}}{2\bar{a}_\text{ro}^2} \right) \\
\begin{bmatrix}
1 \\
\tilde{u}_\text{ro} + n_x \bar{a}_\text{ro} \\
\tilde{v}_\text{ro} + n_y \bar{a}_\text{ro} \\
\tilde{w}_\text{ro} + n_z \bar{a}_\text{ro} \\
\tilde{H}_\text{ro} + \tilde{V}_n \bar{a}_\text{ro} \\
\tau_{xx} / \bar{\rho}_\text{ro} \\
\tau_{xy} / \bar{\rho}_\text{ro} \\
\tau_{xz} / \bar{\rho}_\text{ro} \\
\tau_{yz} / \bar{\rho}_\text{ro} \\
\varepsilon_\text{ro} \\
\end{bmatrix}
\]

\[
\Delta \vec{F}_3 \cdot \mathbf{n} = \vec{V}_n - \bar{a}_\text{ro} \left( \frac{\Delta \vec{p} - \bar{p}_\text{ro} \bar{a}_\text{ro} \Delta \vec{V}}{2\bar{a}_\text{ro}^2} \right) \\
\begin{bmatrix}
1 \\
\tilde{u}_\text{ro} - n_x \bar{a}_\text{ro} \\
\tilde{v}_\text{ro} - n_y \bar{a}_\text{ro} \\
\tilde{w}_\text{ro} - n_z \bar{a}_\text{ro} \\
\tilde{H}_\text{ro} - \tilde{V}_n \bar{a}_\text{ro} \\
\tau_{xx} / \bar{\rho}_\text{ro} \\
\tau_{xy} / \bar{\rho}_\text{ro} \\
\tau_{xz} / \bar{\rho}_\text{ro} \\
\tau_{yz} / \bar{\rho}_\text{ro} \\
\varepsilon_\text{ro} \\
\end{bmatrix}
\]
3.2.1.2 Time Integration

The time integration technique of PUMA_RSM is essentially the same as PUMA2. However, due to the complexity of the problem, only m-stage Runge-Kutta type explicit schemes are adopted.

3.2.1.2.1 Time Step Computation

In PUMA_RSM the local time step is computed through a stability analysis for the inviscid convection viscous diffusion, turbulent diffusion and turbulent dissipation.
[82, 83, 98]. Viscous and turbulent diffusion is then combined to obtain a single time step for the diffusion process. Thus, the time step for a cell $i$ is written as:

$$\Delta t = CFL \times \min[(\Delta t)_{inv}, (\Delta t)_{diff}, (\Delta t)_{diss}]$$

$(\Delta t)_{inv}$ is computed using equations 3.30. Definitions of $(\Delta t)_{diff}$ and $(\Delta t)_{diss}$ are given below.

$$(\Delta t)_{diff} = \frac{0.5 \rho \Pr}{\gamma \mu} \left( \frac{\bar{\rho} \Pr}{\bar{\mu}} + \frac{\bar{P} \Pr_T}{\mu_T} \right) \left[ \Omega_T \sum_{j=1}^{Nfaces} \frac{1}{S_j} \right]^2$$

$$(\Delta t)_{diss} = \frac{k}{\epsilon}$$

### 3.2.2 Boundary Conditions

The boundary conditions implemented in PUMA_RSM are the same as the ones implemented in PUMA2 with slight modifications for the additional turbulence terms. At the farfield boundaries turbulence is assumed to be isotropic and Reynolds stress components are calculated using the freestream turbulence intensity specified by the user. Freestream turbulence dissipation rate is the most difficult item to assign. It is usually obtained by specifying a turbulent Reynolds number (Eq. 2.25) [43] or a turbulence length scale (Eq. 2.52) [83], neither of which can be directly measured. In PUMA_RSM freestream turbulence dissipation rate is by specifying turbulent Reynolds number.
At the solid boundaries all Reynolds stress components are set to zero and turbulence dissipation rate is computed using Eq. 3.66 where \( n \) is the normal direction to the wall.

\[
\varepsilon_w = 2 \frac{\mu}{\rho} \left( \frac{\partial \sqrt{k}}{\partial n} \right)^2 \tag{3.66}
\]

### 3.2.3 Realizability Constraints

The basic requirement for a turbulence model to be acceptable is that it yields turbulent statistics that are realizable [99, 100]. For second moment closures (RSM) realizability means that the Reynolds stress tensor is positive semi-definite which requires the diagonal elements (energy components) to be non-negative and non-diagonal elements to satisfy Cauchy-Schwartz inequality [101].

\[
\tau_{xx} \geq 0 \quad \tau_{yy} \geq 0 \quad \tau_{zz} \geq 0 \tag{3.67}
\]

\[
\left( \tau_{xy} \right)^2 < \tau_{xx} \tau_{yy} \quad \left( \tau_{xz} \right)^2 < \tau_{xx} \tau_{zz} \quad \left( \tau_{yz} \right)^2 < \tau_{yy} \tau_{zz} \tag{3.68}
\]

In addition to this, the turbulence dissipation rate must also be non-negative. It is possible during the iterations that the solution may yield a Reynolds stress tensor or a dissipation rate that does not satisfy the realizability constraints. In order to prevent such non-physical solutions, Reynolds stresses and dissipation rate are checked for at every iteration. If the realizability constraints are not satisfied at a given cell than all the turbulence quantities at that cell are set to zero.

The Reynolds stress tensor for a turbulent flow must also correspond to a point inside the Lumley triangle [102] which can be seen in Figure 3-3.
The sides of the triangle correspond to special states of turbulence, namely two component turbulence, axisymmetric turbulence with one large diagonal element and axisymmetric turbulence with one small diagonal element [43]. These cases act as constraints and $A_2 - A_3$ couples (equations 2.37 and 2.38) at every point in a turbulent flow must lie inside the triangle. This condition is also checked for in PUMA_RSM at every iteration and if the constraints are not satisfied at a given cell, then all the turbulent quantities at that cell are set to zero.
3.3 Grid Generation and Partitioning

Computationally, unstructured meshes are more difficult to handle than the structured meshes because unlike the latter they have no implicit connectivity and the storage requirements are larger. Each grid point, face and cell are numbered and the coordinates of grid points, the nodes that from the faces and cells, and the neighboring cell information has to be stored [89]. However, unstructured grids are relatively easy to construct around complex three-dimensional bodies such as helicopter fuselages which require highly localized regions of grid refinement. Since unstructured meshes can refine the cells in the areas of interest they can offer very efficient cell distributions for accurate solutions of complex real life problems. Therefore, unstructured meshes are employed for all computations of this dissertation.

In this thesis unstructured meshes around the bodies of interest are generated using Gridgen which is a complete meshing toolkit used to generate three-dimensional grids for complex geometries [103]. It can generate structural hexahedral, unstructured tetrahedral and hybrid meshes.

In order to run the codes on parallel computers, a partitioning algorithm is required to distribute the computational domain of interest on several CPUs. The Gibbs-Poole-Stockmeyer (GPS) [104] algorithm is the scheme that is used in this dissertation. The underlying principle of the GPS program is to minimize the bandwidth of the mesh connectivity matrix, in order to reduce the amount of inter-processor communication. The objective is to reduce the time spent on message passing by having fewer messages of greater length. This adequately fits the parallel computing platforms such as Beowulf
clusters whose latency is usually high but whose bandwidth can be comparable to that of supercomputers [15].

### 3.4 Computing Platforms

This section describes some locally available computer facilities which were used for the computations in this thesis.

The COst effective COmputing Array (COCOA2) is a cluster of off-the-shelf PCs connected via fast ethernet. It contains 20 dual 800 MHz Pentium III processors each having 1 GB RAM. These PCs run Redhat Linux with MPI.

COCOA3 is the successor of COCOA2. It contains 60 2.4 GHz. dual Intel Xeon processors, each having 2 GB of RAM and dual 100 Mbps fast Ethernet cards. It also runs Redhat Linux with MPI.

Mufasa [105] is a Linux Beowulf cluster having 85 dual MP2200+ MHz Athlon processors and 1GB RAM. The nodes are connected by high-speed SCI network cards. The high speed Dolphin network is extremely high performance with point-to-point latencies and bandwidth of 1.46 µs and 250 Mbytes/s, respectively. It runs Redhat Linux and the GNU gcc compiler.

LION-XM [106] is a PC cluster with 168 compute nodes each having dual 3.2 GHz Intel Xeon processors and 4 GB of ECC RAM. The nodes are connected by high-speed Myrinet network. LION-XM runs GNU/Linux operating system.

LION-XL [107] is a PC cluster with 128 compute nodes each having dual 2.4 GHz Intel P4 processors and 4 GB of ECC RAM. The nodes are connected by quadrics
very high bandwidth, ultra low latency network. LION-XL runs GNU/Linux operating system.
4.1 Motivation

On helicopters, a ducted tail rotor offers a number of important benefits over a conventional tail rotor. Safety is notably improved through virtual elimination of tail rotor strike events. Noise reductions are also dramatic. From a handling-qualities perspective, the ducted rotor provides the loads capacity required for very aggressive maneuvers and allows unrivaled sideward flight and sideslip envelopes, among other important advantages [108-111].

Despite these significant advantages, the ducted tail rotor presents a design challenge in forward flight. To reduce the momentum drag caused by turning the flow through the fan, the nominal operating condition of the ducted rotor is chosen to be near-zero effective mass flux. In contrast to a conventional rotor, for which the large effective mass flux in forward flight tends to linearize thrust response to collective pitch and reduce the importance of inflow dynamics, near-zero effective mass flux through the duct tends to increase the importance of dynamic inflow and render the approximations of conventional momentum theory inadequate.

Experience on the RAH-66 Comanche has shown that, despite substantial improvements in momentum-type models of the steady thrust response of a ducted tail rotor [112], the dynamics of the total (fan + shroud) thrust response in forward flight are
not yet well understood. The unexpected thrust response was first clearly observed shortly after the initial engagement of the Core Automatic Flight Control System (CAFCS) mode. Figure 4-1 shows a sustained, large-amplitude 1-Hz yaw oscillation during a shallow turning partial-power descent at 80 knots forward speed. (There were no loads or safety issues associated with this oscillation, but it would obviously adversely affect pilot comfort.) These oscillations are not seen in the steady state results but are only seen in the closed-loop control. Notice that the average (trim) value of the FANTAIL™ the pitch (measured at 75% radius) is near zero, where the mass flux through the duct is near zero.

Figure 4-1: Sustained 1-Hz directional axis oscillation in shallow turning partial power descent at 80 Knots.
After an exhaustive review of possible causes of the oscillations, including a careful audit of digital processing delays and consideration of stiction in the actuators, among many other factors, the conclusion was reached that there must be a significant apparent delay in the development of thrust in response to collective pitch changes.

For the Comanche, combinations of changes in directional-axis feedback with modifications in the empennage configuration were sufficient to satisfy the ADS-33 analytical requirements and also to ensure mission effectiveness [113, 114]. The current effort was undertaken to improve the understanding of the aeromechanics of ducted tail rotors in forward flight with the hope of enabling further improvements in this very successful technology. The results presented in the oncoming sections were published in references [27, 34, 115, 116].

4.2 Methodology

In this study the flow field is assumed to be inviscid and the predictions are made using the Euler equations. The FANTAIL™ is modeled using an actuator disk, in which the fan-in-fin is assumed to be a rotor with zero thickness [18]. While the fan blades could be modeled in more detail, the main purpose is to develop a method that requires minimal CPU time and can be used in engineering settings. For our purposes here, it is not believed that detailed modeling of the tip-gap region or blade swirl is critical. It is also assumed that the time scales of the blades are much smaller than the time scale of the outer flow. Two different methods are followed to introduce the effects of the FANTAIL™ to the overall flowfield. In the first method the fan thrust is set as an input
by applying a uniform pressure jump across the actuator disk. On the other hand, in the second method, the blade collective pitch angle is set as an input and the corresponding fan thrust and aerodynamic forces are computed. Details of FANTAIL™ modeling and numerical solution of the Euler equations are covered in the previous chapter.

Steady state computations were performed for hover, forward flight and sideward flight conditions [27,115] while unsteady simulations were performed for hover and forward flight [34, 116]. The governing equations are solved on an unstructured grid with 2.8 million tetrahedral cells, which can be seen in Figure 4-2. This number of cells is adequate to achieve good agreement with wind tunnel test data and is also a case that can be run in a reasonable time on a Beowulf cluster. For example, the steady state hover cases require approximately 38 hours of computer time on four COCOA2 nodes.

Figure 4-2: Computational mesh for RAH-66 Comanche
4.3 Results with Uniform Momentum Theory

This section includes predictions for flow over an isolated fuselage in forward flight, a FANTAIL™ modeled with uniform momentum theory in forward flight. The study is performed to test the actuator disk boundary condition and to have a preliminary knowledge about the effects of the fan-in-fin on the overall flow.

The pressure distribution over the isolated fuselage in forward flight can be seen in Figure 4-3. For the solutions with uniform momentum theory, the flow conditions are kept the same as the isolated fuselage case and the fan thrust coefficient is set to 0.008. Figure 4-4 shows the surface pressure distribution and Figure 4-5 shows the comparison of computed and experimental [10] dorsal midline, i.e. top of the vehicle, pressure distributions.

Since the fan thrust applied for this case is very small there is not much difference between isolated fuselage and fan-in-fin operating cases. One can see from Figure 3-7 that the computed dorsal midline pressure shows good agreement with the experiment. The slight discrepancies are most likely due to slight geometric differences between the experimental and the computational model, the use of a uniform pressure jump assumption, and viscous effects.
Figure 4-3: Pressure distribution over isolated fuselage ($M_\infty=0.13$, $V_\infty=85.9$ knots, $\alpha=0^\circ$).

Figure 4-4: Pressure distribution over fuselage ($M_\infty=0.13$, $V_\infty=85.9$ knots, $\alpha=0^\circ$, $C_{T\text{fan}}=0.008$).
4.4 Results with Blade Element Theory

Uniform momentum theory may yield reliable results but in reality the FANTAIL\textsuperscript{TM} imposes a non-uniform pressure jump on the rotor disks [10]. In addition to this, a uniform momentum theory cannot account for the pitch angle settings of the blades. Therefore, as a more realistic approach, blade element theory, which relates local pressure jump to local velocity field and blade pitch angle, is coupled with the CFD solution. Unlike the first method, where the fan thrust is given as input, in this case the blade pitch angle is specified and the corresponding fan thrust is computed. Here the pitch angle is measured at the \( \frac{3}{4} \) radius position [110].
4.4.1 Steady State Simulations

This section presents steady state flowfields and static thrust response for different blade pitch settings at different flight conditions.

Figure 4-6 to 4-11 show pressure distributions and velocity vectors in the vicinity of the FANTAIL™ for pitch angles -10, 20 and 40 degrees for the hover condition.

The effect of the blade pitch setting on the pressure distribution and the velocity field are evident in the figures. Also note from the figures that the suction of the fan creates a low pressure region on the shroud around the inlet lip, which leads to additional antitorque moment.

Figure 4-6: Cp contours in the vicinity of FANTAIL™ (Hover, starboard and port views, $\theta_{0.75}=-10^\circ$)
Figure 4-7: Velocity vectors in the vicinity of FANTAIL™, hover, $\theta_{75} = -10^\circ$ (Net thrust is towards port side)

Figure 4-8: Cp contours in the vicinity of FANTAIL™, hover, starboard and port views, $\theta_{75} = 20^\circ$
Figure 4-9: Velocity vectors in the vicinity of FANTAIL\textsuperscript{TM} Hover, $\theta_{75} = 20^\circ$) (Net thrust is towards starboard side)

Figure 4-10: $C_p$ contours in the vicinity of FANTAIL\textsuperscript{TM} Hover, starboard and port views, $\theta_{75} = 40^\circ$)
Pressure and velocity distributions for the helicopter in forward flight with freestream velocity of 150 knots and collective pitch settings of 0, 20 and 40 degrees are shown in Figure 4-12 to 4-17.

Figure 4-11: Velocity vectors in the vicinity of FANTAIl™ (Hover, $\theta_{0.75} = 40^\circ$) (Net thrust is towards starboard side)
Figure 4-12: Cp contours in the vicinity of FANTAIL™ (Forward Flight, starboard and port views, $V_\infty=150$ kts, $\theta_{.75} = 0^\circ$)

Figure 4-13: Velocity vectors in the vicinity of FANTAIL™ (Forward Flight, $V_\infty=150$ kts, $\theta_{.75} = 0^\circ$) (Net thrust is towards port side)
Figure 4-14: Cp contours in the vicinity of FANTAIL™ (Forward Flight, starboard and port views, $V_\infty=150$ kts, $\theta_{.75} = 20^\circ$)

Figure 4-15: Velocity vectors in the vicinity of FANTAIL™ (Forward Flight, $V_\infty=150$ kts, $\theta_{.75} = 20^\circ$) (Net thrust is towards starboard side)
Figure 4-16: Cp contours in the vicinity of FANTAIL™ (Forward Flight, starboard and port views, $V_\infty=150$ kts, $\theta_{75} = 40^\circ$)

Figure 4-17: Velocity vectors in the vicinity of FANTAIL™ (Forward Flight, $V_\infty=150$ kts, $\theta_{75} = 40^\circ$) (Net thrust is towards starboard side)
In forward flight, the low fan thrust is desired to minimize drag and power consumption [110]. Figures 4-12, 4-14 and 4-16 clearly show that, increasing the blade pitch angle creates a high pressure region on the downstream side of the duct. This results in a significant aft force. To give better overall performance, it is desirable to design the vertical tail so that the fan is unloaded in trim. Note that this design approach also tends to push the fan into the region, near zero degrees of pitch, where the trim flow through the duct is not well defined as can be seen from Figure 4-13. In addition to this, a low pressure region develops on the port side of the shroud with increasing pitch angle. This also tends to decrease the total thrust. Unloading the fan also eliminates this effect.

Another important flight condition is sideward flight. Since the flow goes directly through the fan the inflow velocities and consequently the local angle of attack of the blades change drastically. Figures 4-18 and 4-19 show the pressure and velocity distributions around the FANTAIL™ when the helicopter is in a left sideward flight of 45 knots and the collective pitch setting is zero degrees.

Figure 4-18: Cp contours in the vicinity of FANTAIL™ (Left Sideward Flight, starboard and port views, \( V_\infty = 45 \) kts, \( \theta_{75} = 0^\circ \))
Figures 4-7, 4-9, 4-11, 4-13, 4-15, 4-17, and 4-19 show that except for the hover case with positive fan thrust, the flow through the duct is not well defined. Separated flow regions can be easily seen in the vicinity of the duct and the centerbody. This also shows that the Euler equations are able to predict these kinds of flow when the separation is from a sharp edge and is Reynolds number independent. (For Reynolds number dependent flows, one would have to use the Navier-Stokes equations).

In order to analyze the control characteristics of the FANTAIL™, thrust predictions are plotted as a function of the collective pitch angle for different flight conditions and the results are compared with wind tunnel data [30, 110, 117]. Here the total thrust generated by the FANTAIL™ and the thrust generated by the fan only are compared with the experiments separately. Figure 4-20 illustrates the relationship

Figure 4-19: Velocity vectors in the vicinity of FANTAIL™ (Left Sideward Flight, $V_{\infty}=45$ kts, $\theta_{75}=0^\circ$) (Net thrust is towards starboard side)
between the collective pitch angle and the total thrust in hover condition. Comparisons of the thrust generated by the fan only with the wind tunnel data for hover and right sideward flight at a speed of 45 knots are available in Figure 4-21 Figure 4-22. Solidity appearing in the figures is defined as:

\[
\sigma = \frac{N_b \cdot c \cdot (R - R_{cb})}{\pi (R^2 - R_{cb}^2)} = \frac{N_b \cdot c}{\pi (R + R_{cb})}
\]

Another result of interest from the static thrust results is that the ratio of total device thrust to fan thrust is about 1.83 and is very nearly independent of collective pitch. This value is very close to the ideal augmentation factor of 2.0.

---

**Figure 4-20:** Comparison of total thrust with experiment. (Hover)
Fan Thrust vs. Pitch Angle
(Hover)

Figure 4-21: Comparison of fan thrust with experiment. (Hover)

Fan Thrust vs. Pitch Angle
(Right Sideward Flight, $V = 45$ knots)

Figure 4-22: Comparison of fan thrust with experiment, right sideward flight, $V_\infty=45$ kts
The figures show that the results are generally in excellent agreement with the wind tunnel data. The differences are most likely due to the inviscid flow and linear lift curve assumptions.

Figures 4-21 and 4-22 clearly showed that the fan thrust for a given collective pitch setting changes according to the flight condition of the aircraft. In order to display this effect in more detail, fan thrust is plotted as a function of pitch angle for hover, forward flight and sideward flight in Figure 4-23.

![Figures showing fan thrust characteristics in different flight conditions.](image)

**Figure 4-23:** Fan thrust characteristics in different flight conditions.

Results show that the introduction of forward speed causes a dramatic change in the thrust slope (change in thrust per degree of collective) near zero collective, a result which qualitatively agrees with flight data [112] and is critical for the setting of the feedback gain schedule. Right sideward flight is analogous to a main rotor in a vertical climb where the freestream velocity adds to the inflow and decreases the local angle of attack of the blades. This results in less thrust for a given collective pitch setting than in
hover. In left sideward flight this situation is reversed and the fan acts like a rotor in vertical descent. The freestream velocity decreases the inflow and increases the local angle of attack of the blades. For low pitch angles, the fan creates more thrust than in hover. But for high pitch settings this effectiveness will decrease because of the increasing induced flow.

Having validated the code, steady state results can further be used to obtain basic directional stability characteristics of the helicopter. Weathercock stability, which is essentially the variation of yawing moment with sideslip angle, is one of them. Figure 4-24 shows the variation of aircraft yawing moment with sideslip angle in forward flight with zero degrees of collective pitch setting. The yawing moment presented in Figure 4-24 are computed with respect to the nose of the aircraft. For stability, the slope of the line must be positive. Since the yawing moment was not computed with respect to the center of gravity of the aircraft, the predictions do not really give information about stability. But they still give some hints about the behavior of yawing moment with changing sideslip angle.

In forward flight, the fuselage also generates a significant amount of antitorque moment. For a more detailed analysis, variations of yawing moments generated by fan, shroud, horizontal tail, vertical tail and other fuselage surfaces with sideslip angle are displayed in Figure 4-25. As expected, the vertical tail is the dominant contributor of weathercock stability. The word “other” seen in the legend of the figure stands for all other fuselage surfaces except shroud, horizontal tail and vertical tail.
Variation of Total Yawing Moment with Sideslip Angle

Figure 4-24: Variation of yawing moment with sideslip angle, forward flight, $V_{\infty}=150$ kts, $\theta_{75}=0^\circ$.

Variation of Yawing Moment Components with Sideslip Angle

Figure 4-25: Variation of yawing moment components with sideslip angle, forward flight, $V_{\infty}=150$ kts, $\theta_{75}=0^\circ$.
4.4.2 Unsteady Simulations

Understanding the dynamic relationship between the antitorque thrust moment and the applied collective pitch angle is crucial, especially for directional control sensitivity analyses. Although there are many studies in the literature on the steady state behavior of the FANTAIL™ [9, 10, 27, 30, 115], little is known about the transient response and thrust build up. This study was performed to simulate the unsteady flow around the fuselage of an RAH-66 Comanche helicopter and to analyze the effects of varying the collective pitch angle.

The results illustrate the transient response of aerodynamic forces and moments to changes in the collective pitch settings in hover and forward flight. In the studies the pitch angle is changed by 5 degrees from an equilibrium point at a rate of 144 degrees per second and the development of the fan thrust and yawing moment are analyzed. Figures 4-26 to 4-29 show the variations of collective pitch angle, fan thrust, total yawing moment and yawing moment components due to fan and fuselage with time, respectively.

A rapid decrease of the pitch angle decreases the angle of attack of the blades, and consequently the fan thrust. A decrease in fan thrust will also decrease the induced velocity, which will result in increasing the blade angle of attack and the fan thrust. Due to the inertia of the fluid, the induced flow does not respond as quickly as the pitch settings, thus an overshoot and then decay to a steady state value is observed (Figure 4-27). The yawing moment generated by the fuselage also shows a similar behavior which can be seen in Figure 4-29. The transient response of fan thrust and yawing moment in forward flight for the same pitch input are displayed in Figures 4-30, 4-31, and 4-32.
Figure 4-26: Variation of collective pitch angle with time (Hover, $\theta_{75}=20^\circ \rightarrow 15^\circ$)

Figure 4-27: Variation of fan thrust with time (Hover, $\theta_{75}=20^\circ \rightarrow 15^\circ$)
Figure 4-28: Variation of total yawing moment with time. (Hover, $\theta_{75}=20^\circ \rightarrow 15^\circ$)

Figure 4-29: Variations of yawing moment components due to fan and fuselage with time. (Hover, $\theta_{75}=20^\circ \rightarrow 15^\circ$)
Figure 4-30: Variation of fan thrust with time. (Forward Flight, $V=150$ knots, $	heta_{75}=20^\circ \rightarrow 15^\circ$)

Figure 4-31: Variation of total yawing moment with time. (Forward Flight, $V=150$ knots, $	heta_{75}=20^\circ \rightarrow 15^\circ$)
It is evident that with the forward speed the vertical tail generates a significant amount of antitorque moment. But the contribution, and transient behavior, of each component is still not obvious. Therefore, a more detailed analysis was performed to simulate the unsteady response of forces and moments generated by the fan, shroud, horizontal tail, vertical tail, and other fuselage surfaces. Since the nominal operating condition for the FANTAIL™ is near zero pitch angle, a new case is analyzed by changing the collective pitch angle from zero to five degrees at 144 degrees per second. Variations of collective pitch angle, fan thrust, total yawing moment and yawing moments generated by different components (fan, shroud, horizontal tail, vertical tail and other fuselage surfaces) in forward flight are shown in Figures 4-33 to 4-36. The time history of the average inflow velocity can be seen in Figure 4-37. It is clear from Figure 4-34 that fan thrust shows a qualitatively similar response to the previous case but
now with more oscillatory behavior. Inflow velocity also shows a similar oscillatory behavior, which qualitatively explains the oscillatory thrust response. After the collective pitch angle stops changing, the average inflow velocity continues to increase and then to decrease, which effectively first decreases then increases the local blade angle of attack. But when Figures 4-36 and 4-37 are examined, it is seen that between 0.15 and 0.27 seconds the change in inflow is very small, but the shroud force changes a lot. These results suggest that the simple approach that is widely used to model dynamic inflow response cannot be used to capture what is happening for the ducted rotor in forward flight. Shroud thrust still remains unknown, so another degree of freedom is needed in the simplified models to capture this effect [33].

Figure 4-33: Variation of collective pitch angle with time. (Forward Flight, V=150 knots, $\theta_{75} = 0^\circ \rightarrow 5^\circ$)
Figure 4-34: Variation of fan thrust with time. (Forward Flight, \(V=150\) knots,
\(\theta_{75}=0^\circ \rightarrow 5^\circ\))

Figure 4-35: Variation of total yawing moment with time. (Forward Flight, \(V=150\) knots,
\(\theta_{75}=0^\circ \rightarrow 5^\circ\))
Figure 4-36: Variations of yawing moment components with time. (Forward Flight, V=150 knots, $\theta_{.75}=0^\circ \rightarrow 5^\circ$)

Figure 4-37: Variation of average inflow velocity with time. (Forward Flight, V=150 knots, $\theta_{.75}=0^\circ \rightarrow 5^\circ$)
As mentioned earlier, with the introduction of forward speed other components of the fuselage start to generate a significant amount of yawing moment. This situation can be seen in Figure 4-36. It is evident that the shroud and the vertical tail are the dominating components for the yawing moment. They also significantly affect the transient response of the total yaw moment which can be observed in Figure 4-35. For a better understanding of the yawing moment behavior of shroud and the vertical tail, surface pressure distributions of the FANTAIL™ are displayed in Figures 4-38 to 4-43.

Figure 4-38: Pressure distribution around FANTAIL™. (Forward flight, V = 150 knots, t = 0.011 sec)
Figure 4-39: Pressure distribution around FANTAIL™. (Forward flight, V = 150 knots, t = 0.034 sec)

Figure 4-40: Pressure distribution around FANTAIL™. (Forward flight, V = 150 knots, t = 0.058 sec)
Figure 4-41: Pressure distribution around FANTAIL™. (Forward flight, $V = 150$ knots, $t = 0.077$ sec)

Figure 4-42: Pressure distribution around FANTAIL™. (Forward flight, $V = 150$ knots, $t = 0.1$ sec)
An analysis of Figure 4-38 to 4-43 yields several conclusions. Initially, the rapid increase of the blade pitch angle creates suction which results in a low pressure region on the upstream lip of the starboard side. This increases the yawing moment generated by the shroud. But as the flow develops, after the new pitch angle is set, the pressure in the vicinity of the downstream lip of the shroud begins to increase. This not only decreases the side force developed, but it also moves it more forward. This results in a low yawing moment. In addition to this, as time passes the pressure on the downstream part of the port side begins to decrease, which also effectively decreases the antitorque moment. Figure 4-43 shows that the pressure in this region begins to increase and cause an oscillatory behavior.

It can also be observed from the figures that the pressure on the starboard side of the vertical tail decreases gradually, which creates a smooth gradual increase in the antitorque moment created by this component.
The results for forward flight show that, at low pitch angles, convergence to the steady state value is slower. In order to further analyze this situation two additional flowfield solutions were performed for hover. In the first one the collective pitch angle is changed from zero to five degrees and in the second one from 35 to 40 degrees at 144 degrees per second. The time histories of collective pitch angle, fan thrust, yawing moments and average inflow velocity for the first case are shown in Figures 4-44 to 4-48.

Figure 4-44: Variation of collective pitch angle with time. (Hover, $\theta_{75}=0^\circ \rightarrow 5^\circ$)
Figure 4-45: Variation of fan thrust with time. (Hover, $\theta_{75}=0^\circ \rightarrow 5^\circ$)

Figure 4-46: Variation of total yawing moment with time. (Hover, $\theta_{75}=0^\circ \rightarrow 5^\circ$)
Time histories of collective pitch angle, fan thrust, yawing moments and average inflow velocity for the second case are shown in Figures 4-49 to 4-53.
Figure 4-49: Variation of collective pitch angle with time. (Hover, $\theta_{75}=35^\circ \rightarrow 40^\circ$)

Figure 4-50: Variation of fan thrust with time. (Hover, $\theta_{75}=35^\circ \rightarrow 40^\circ$)
Figure 4-51: Variation of total yawing moment with time. (Hover, $\theta_{75}=35^\circ \rightarrow 40^\circ$)

Figure 4-52: Variations of yawing moment components with time. (Hover, $\theta_{75}=35^\circ \rightarrow 40^\circ$)
Figures 4-45, 4-46, 4-50 and 4-51 show that for the same amount of change in the collective pitch angle, a higher overshoot is observed at lower thrust values. In addition to this, convergence to a steady state value is also slower for the low thrust case. These situations were also observed in flight tests. Unlike forward flight, the fan is the dominant component for antitorque moment in hover. This situation becomes more severe as the collective pitch angle increases as can be seen from Figures 4-47 and 4-52.

Figure 4-53: Variation of average inflow velocity with time. (Hover, $\theta_{75}=35^\circ \rightarrow 40^\circ$)

Chapter 5

Reynolds Stress Modeling for Separated Turbulent Flow Simulations

5.1 Motivation

Separated turbulent flow around complex three-dimensional geometries is one of the most challenging problems in aerospace. For example, one of the common problems in helicopter fuselage aerodynamics is separated turbulent flow. Flow separates from the hub or fuselage and then impinges on the tail rotor, empennage and control surfaces. Separated flow around a body results in many phenomena in aerodynamics, such as drag increase, lift loss, unsteady fluctuations, etc. Therefore, accurate turbulence prediction is a key to understanding and predicting separated turbulent flows around aerodynamic devices [15]. The presence of three-dimensionality and curvature introduces changes in the turbulence structure, thus invalidating many of the turbulence models used widely for simple and mildly complex shear layers [52]. Therefore, it becomes extremely important to employ more physics in providing suitable closure models for adequate prediction of these complex flows.

Turbulence models based on the Boussinesq approximation are mainly developed for attached simple shear flows, where there is only one significant mean strain component [52]. While these models can be modified to improve their capabilities, the modifications are largely ad hoc and cannot be easily generalized [51]. Moreover, in the case of strong separation, even the modified two-equation models fail to predict flow
physics due to their isotropic nature [53]. Therefore, anisotropic models such as full Reynolds stress transport models (RSM) are necessary for accurate prediction of three-dimensional separated flows. Although RSM does not represent the unsteady nature of turbulence as does dynamic models such as Large Eddy Simulation (LES) [118], it is very effective in computing the time averaged quantities and is an order of magnitude faster to compute than LES.

5.2 Methodology

The RSM used in this dissertation is composed of coupling the Favre-Reynolds Averaged Navier-Stokes equations with the transport equations for Reynolds stresses and turbulence dissipation rate. The resulting system of 12 coupled non-linear partial differential equations is numerically solved using the computer code PUMA_RSM. The equations are discretized by a second order finite volume method and solved using an explicit four-stage Runge-Kutta type integration technique. Although explicit methods suffer from limitations on the maximum allowable time step size, they require a minimum number of arithmetic operations per iteration step and are easier to parallelize compared to implicit methods [119].

5.3 Turbulent Flow Simulations around a 6:1 Prolate Spheroid

Although it has a very simple geometry, the flow around a prolate spheroid at incidence exhibits a variety of complex three-dimensional turbulent shear flows such as;
stagnation flow, three-dimensional boundary layers under the influence of strong pressure
gradients, streamline curvature, cross-flow separation, formation and evolution of free
vortex sheet, streamline vortices, etc [120]. Figure 5-1 shows the contours of normalized
invariant of the deformation tensor [121] computed using PUMA_RSM at an axial station
$x/L = 0.77$ where $x$ is the axial coordinate and $L$ is the length of the prolate spheroid. The
invariant is defined as:
\[
D = \frac{S_{ij}S_{ij} - \Omega_{ij}\Omega_{ij}}{S_{ij}S_{ij} + \Omega_{ij}\Omega_{ij}}
\]
\[5.1\]
where $S_{ij}$ is the mean strain rate tensor and $\Omega_{ij}$ is the mean rotation rate tensor.

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right)
\]
\[5.2\]
\[
\Omega_{ij} = \frac{1}{2} \left( \frac{\partial \tilde{u}_i}{\partial x_j} - \frac{\partial \tilde{u}_j}{\partial x_i} \right)
\]
\[5.3\]

The value of this invariant varies between -1 and 1. It denotes pure strain when it is 1, pure shear when it is 0, and pure rotation when it is -1. All these extremes can be seen in Figure 5-1 which shows that the analyzed flow is highly three-dimensional and carries extra rates of strain and rotation; a very difficult case for turbulence models.
The following solutions for the prolate spheroid are performed at a Reynolds number of $\text{Re} = 6.5 \times 10^6$, Mach number of 0.1322 (45 fps), and at an incidence of 30 degrees. This condition is selected because of the available experimental data [122]. The turbulence intensity ($T_u$) at freestream is taken as 0.1 % according to the wind tunnel measurements [122] and the turbulent Reynolds number of the freestream is assumed to be 150. The numerical solution is obtained using an unstructured mesh of one million tetrahedral cells, a sectional view of which can be seen in Figure 5-2. The cells are clustered in the vicinity of the solid body in order to obtain sufficient boundary layer resolution. Here the $y^+$ values of the first cells are on the order of one. The details of the cells close to the solid surface are displayed in Figure 5-3.

Figure 5-1: Contours of normalized invariant of deformation tensor. 
($\text{Re} = 6.5 \times 10^6, \alpha = 30^\circ, M = 0.1322, x/L = 0.77$)
Figure 5-2: Sectional view of the mesh for 6:1 prolate spheroid (1 million cells)

Figure 5-3: Close-up view of the cells near the solid boundary (6:1 prolate spheroid) (1 million cells)
One of the most important features of this flow is the phenomenon known as cross-flow separation. Unlike two-dimensional separation, three-dimensional separation is rarely associated with the vanishing of the wall shear stress [123]. Here a sheet of fluid moves away from the surface and rolls into a vortex. This causes the surface skin friction lines on either side of the separation to converge asymptotically toward a separation line [124, 125]. At high angles of attack secondary separation might occur at a higher circumferential angle. Such behavior can be easily observed from Figure 5-4 where surface skin friction lines are displayed.

Figure 5-4: Surface skin friction lines
(Re = 6.5x10^6, α = 30°, T_u = 0.1%, Re_T = 150, M = 0.1322)
Cross-flow separation generates stream-wise vortices by causing flow reversal in the circumferential direction. Figures 5-5, 5-6 and 5-7 show circumferential velocity distributions and Figures 5-8, 5-7 and 5-8 show circumferential boundary layer profiles at different axial stations.

Figure 5-5: Circumferential Velocity Distribution (Re=6.5x10^6, α=30°, M=0.1322, Tu=0.1%, ReT=150, x/L=0.4839)
Figure 5-6: Circumferential Velocity Distribution (Re=6.5x10^6, α=30°, M=0.1322, Tu=0.1%, ReT=150, x/L=0.6571)

Figure 5-7: Circumferential Velocity Distribution (Re=6.5x10^6, α=30°, M=0.1322, Tu=0.1%, ReT=150, x/L=0.8279)
Figure 5-8: Circumferential Boundary Layer Profiles (Re=6.5x10^6, α=30°, M=0.1322, 
Tu=0.1%, Re_T=150, x/L=0.4839)

Figure 5-9: Circumferential Boundary Layer Profiles (Re=6.5x10^6, α=30°, M=0.1322, 
Tu=0.1%, Re_T=150, x/L=0.6571)
It is clear from the images that a sheet of fluid starts to move away from the spheroid and then rolls into a vortex. As we move toward the trailing edge, the reverse flow region grows and vortical flowfield moves away from the body.

Figures 5-11, 5-12 and 5-13 display the variation of turbulent kinetic energy normalized by the freestream kinetic energy at different axial stations. These contours show that moving of the circumferential flow away from the spheroid draws a sheet of very turbulent fluid from the boundary layer towards the core of the vortical flow. Such behavior was also observed in detailed wind tunnel measurements [123].
Figure 5-11: Variation of Turbulent Kinetic Energy (Re=6.5x10^{6}, \alpha=30^\circ, M=0.1322, Tu=0.1\%, Re_T=150, x/L=0.4839)

Figure 5-12: Variation of Turbulent Kinetic Energy (Re=6.5x10^{6}, \alpha=30^\circ, M=0.1322, Tu=0.1\%, Re_T=150, x/L=0.6571)
It is known that the intensity of the freestream turbulence in the flow affects the evolution of the turbulent flowfield [126, 127]. In order to analyze this event the above problem is solved with a new turbulence intensity of 1 % but the turbulent Reynolds number of freestream was kept unchanged. Circumferential velocity distributions at the same axial stations as the previous case are displayed in Figures 5-14, 5-15 and 5-16. Comparing these figures with Figures 5-5, 5-6 and 5-7, shows that the negative velocity region of the second case is bigger in all the axial stations. Although the turbulence intensity of the second case was ten times more than the first case, it yielded a bigger separated region. The differences between the solutions are also evident in the distribution of skin friction lines on the surface. Figure 5-17 shows the skin friction lines of the 1 % turbulence intensity case. The solution could not even predict the secondary separation line, meaning that the flow did not reattach after the primary separation.
Figure 5-14: Circumferential Velocity Distribution (Re=6.5x10^6, α=30°, M=0.1322, Tu=1%, Re_T=150, x/L=0.4839)

Figure 5-15: Circumferential Velocity Distribution (Re=6.5x10^6, α=30°, M=0.1322, Tu=1%, Re_T=150, x/L=0.6571)
Figure 5-16: Circumferential Velocity Distribution (Re=6.5x10^6, α=30°, M=0.1322, Tu=1%, ReT=150, x/L=0.8379)
These results were somewhat unexpected. The higher freestream turbulence of the second case was expected to delay or even prevent the primary separation. But the opposite of this is observed. The answer to this problem may lie in the definition of turbulent Reynolds number. The number is defined by Eq. 2.25. This equation clearly states that in order to increase turbulent intensity by ten times without changing the turbulent Reynolds number, one has to increase the turbulence dissipation rate by $10^4$ times. This drastically increases the rate of decay of freestream turbulence. Figures 5-18 and 5-19 show the turbulent kinetic energy contours normalized by the freestream value at the symmetry plane for both cases.
Figure 5-18: Turbulent Kinetic Energy Contours (Re=6.5x10^6, α=30°, M=0.1322, Tu=0.1%, ReT=150)

Figure 5-19: Turbulent Kinetic Energy Contours (Re=6.5x10^6, α=30°, M=0.1322, Tu=1%, ReT=150)
It is clear from these figures that in the second case turbulent kinetic energy decays much more rapidly and drops to a significantly lower fraction of its freestream value. For further analysis a new solution is performed with the same mean flow conditions. Turbulence intensity is taken as 1 % but turbulence dissipation rate value is kept the same as the first case. This causes the turbulent Reynolds number of this third case to be 1.5 million. Figure 5-20 shows the surface skin friction lines plotted for this third case. High turbulence clearly prevented cross-flow separation. This is also supported by the circumferential velocity plots displayed in Figures 5-21, 5-22 and 5-23 at the same axial stations as the other two cases. These images also show no sign of cross-flow separation or reverse flow.

Figure 5-20: Surface skin friction lines (Re=6.5x10^6, α=30°, T_u=1%, Re_T=1.5 million, M=0.1322)
Figure 5-21: Circumferential Velocity Distribution (Re=6.5x10^6, α=30°, M=0.1322, T_0=1%, Re_T =1.5million, x/L=0.4839)

Figure 5-22: Circumferential Velocity Distribution (Re=6.5x10^6, α=30°, M=0.1322, T_0=1%, Re_T =1.5million, x/L=0.6571)
These results revealed very interesting conclusions. It was known that the intensity of freestream turbulence affects the evolution of a turbulent flow. But here it was also shown that besides turbulence intensity, the value of the turbulent Reynolds number (or turbulence length scale) can drastically affect the evolution of the flow. In the third case the solutions could not predict cross-flow separation, which is one of the most important features of the current flow [122]. Although the amount of turbulence present in a wind tunnel is usually measured and given, no information can be found about turbulent Reynolds number, turbulence length scale or turbulence dissipation rate. Therefore, a reasonable number for these quantities could only be assigned through some experience. The value of 150 assigned for turbulent Reynolds number in this dissertation is generally considered to be a number above which the original (high Reynolds number)
versions of the Reynolds Stress Model works well. Hence it is used in this study. In addition to this, the solutions performed using this value qualitatively gave correct results. In order to check the validity of the solutions quantitatively, results are compared with wind tunnel measurements [122].

5.3.1 Circumferential Velocity Distribution

Comparisons include circumferential velocity predictions at different axial stations \((x/L = 0.4839, 0.6571, 0.8279)\) and at different circumferential angles \((\varphi = 30, 60, 90, 120, \text{and } 150 \text{ degrees})\). Note that, \(\varphi = 0\) corresponds to the bottom of the spheroid, and \(\varphi = 180\) corresponds to the top.

5.3.1.1 Circumferential Velocity Distribution at \(x/L = 0.4839\)

Figures 5-24 to 5-28 display the variation of circumferential velocity versus distance from the wall. The parameter \(h\) in the figures stands for the normal distance from the solid surface. Predictions are in very good agreement with measurements with some discrepancies observed at \(\varphi=30^\circ\) and \(\varphi=150^\circ\). The former is close to the bottom of the body where the flow is attached. Discrepancies may be due to insufficient grid resolution in the circumferential direction because on this part of the body the flow is under a favorable pressure gradient. The latter is close to the top of the spheroid where the flow is separated. Although the lower peak is captured well, the upper peak is slightly underestimated. This may be due to premature dissipation of the vortical flowfield due to
grid coarsening. There may also be some errors in the measurements. Unfortunately there were not any error bars available for the wind tunnel data. Therefore, the extent of measurement errors is not known.

![Circumferential Velocity Diagram](image)

**Figure 5-24**: Circumferential Velocity, $x/L = 0.4839$, $\varphi = 30^\circ$ (Re=6.5x10$^6$, $\alpha=30^\circ$, $M=0.1322$)
Figure 5-25: Circumferential Velocity, \( x/L = 0.4839, \varphi = 60^\circ \) (\( Re=6.5\times10^6, \alpha=30^\circ, M=0.1322 \))

Figure 5-26: Circumferential Velocity, \( x/L = 0.4839, \varphi = 90^\circ \) (\( Re=6.5\times10^6, \alpha=30^\circ, M=0.1322 \))
Figure 5-27: Circumferential Velocity, \( x/L = 0.4839, \phi = 120^\circ \) (Re=6.5x10^6, \( \alpha = 30^\circ \), M=0.1322)

Figure 5-28: Circumferential Velocity, \( x/L = 0.4839, \phi = 150^\circ \) (Re=6.5x10^6, \( \alpha = 30^\circ \), M=0.1322)
5.3.1.2 Circumferential Velocity Distribution at $x/L = 0.6571$

The variation of circumferential velocity in the normal direction for $x/L = 0.6571$ are plotted in Figures 5-29 to 5-33. Comparisons are very similar to those at $x/L = 0.4839$. Again discrepancies between predictions and measurements are observed at $\phi = 30^\circ$ and $\phi = 150^\circ$ while there is a good agreement at other locations. The difference between the solutions with $Tu = 0.1 \%$ and $1 \%$ are more clear at $\phi = 120^\circ$ (Figure 5-32). $Tu = 0.1 \%$ gave better results.

Figure 5-29: Circumferential Velocity, $x/L = 0.6571$, $\phi = 30^\circ$ (Re=6.5x10$^6$, $\alpha=30^\circ$, $M=0.1322$)
Figure 5-30: Circumferential Velocity, \(x/L = 0.6571, \phi = 60^\circ\) (\(Re=6.5\times10^6, \alpha=30^\circ, M=0.1322\))

Figure 5-31: Circumferential Velocity, \(x/L = 0.6571, \phi = 90^\circ\) (\(Re=6.5\times10^6, \alpha=30^\circ, M=0.1322\))
Figure 5-32: Circumferential Velocity, \( x/L = 0.6571, \varphi = 120^\circ \) (Re=6.5\times10^6, \( \alpha=30^\circ \), M=0.1322)

Figure 5-33: Circumferential Velocity, \( x/L = 0.6571, \varphi = 150^\circ \) (Re=6.5\times10^6, \( \alpha=30^\circ \), M=0.1322)
5.3.1.3 Circumferential Velocity Distribution at x/L = 0.8279

Circumferential velocity plots for x/L = 0.8279 are displayed in Figures 5-34 to 5-38. This station is closer to the end of the spheroid where separation is stronger, but the results are still similar to those at the other stations. Unlike the first two stations negative velocity can be observed at $\varphi = 120^\circ$ and predictions follow the measurements very closely. Again $T_u = 0.1 \%$ case performed better than $T_u = 1 \%$ case. Discrepancies between predictions and measurements at $\varphi = 150^\circ$ are more drastic here.

Figure 5-34: Circumferential Velocity, $x/L = 0.8279$, $\varphi = 30^\circ$ (Re=$6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$)
Figure 5-35: Circumferential Velocity, $x/L = 0.8279$, $\varphi = 60^\circ$ (Re=6.5x10^6, $\alpha=30^\circ$, $M=0.1322$)

Figure 5-36: Circumferential Velocity, $x/L = 0.8279$, $\varphi = 90^\circ$ (Re=6.5x10^6, $\alpha=30^\circ$, $M=0.1322$)
Figure 5-37: Circumferential Velocity, \( x/L = 0.8279, \varphi = 120^\circ \) (Re=6.5x10^6, \( \alpha = 30^\circ \), M=0.1322)

Figure 5-38: Circumferential Velocity, \( x/L = 0.8279, \varphi = 150^\circ \) (Re=6.5x10^6, \( \alpha = 30^\circ \), M=0.1322)
Velocity comparisons at three axial stations revealed similar behavior. At all stations predictions were in good agreement with measurements except at $\phi = 30^\circ$ and $\phi = 150^\circ$. The former is at the windward side of the spheroid where the flow is attached and under very high favorable pressure gradient. Thus the flow accelerates very fast in the circumferential direction. The mesh employed for the solutions is sufficiently fine in the normal direction but relatively coarse in the circumferential direction and probably not sufficient to resolve the high pressure gradient in this region. The latter location is on the leeside of the body where the flow is separated and highly vortical. It is known that vortical flow simulations are very susceptible to numerical dissipation. Premature dissipation of vortices has been observed in places where the computational mesh is not fine enough to resolve high velocity gradients in the vicinity of the vortex core [119]. Therefore, discrepancies at $\phi = 150^\circ$ are mainly due to insufficient grid resolution due to which vortices could not be properly captured.

In order to test the ability of the code in predicting the cross-flow separation location a new solution was obtained to predict the separation locations along the length of the prolate spheroid. These new solutions are performed at a Reynolds number of $7.2 \times 10^6$, a Mach number of 0.1322 and an angle of attack of $30^\circ$. Solutions are compared with the relevant wind tunnel measurements [128]. Figure 5-39 shows surface skin friction lines. A single separation line is predicted by the solution which agrees with the experiment [128]. Separation locations along the length of the prolate spheroid are displayed in Figure 5-40.
Figure 5-39: Surface skin friction lines (Re=7.2×10^6, α=30°, T_u=0.1%, Re_T=150, M=0.1322)

Figure 5-40: Axial Position of Separation Locations (Re=7.2×10^6, α=30°, T_u=0.1%, Re_T=150, M=0.1322)
Although computations were performed on a relatively coarse mesh, predictions are in good agreement with the measurements. This shows that the RSM employed for this dissertation is very good in capturing three-dimensional cross-flow separation, which is a very hard case for many turbulence models. Some discrepancies are observed in the forward locations. At these regions separation is very weak so it is very hard to predict with a coarse mesh or observe in a wind tunnel.

The final set of computations with the current coarse mesh was performed to check the method’s ability to capture the three-dimensionality and anisotropy of turbulence. Townsend’s structural parameter relates Reynolds shear stresses to turbulent kinetic energy. It is defined in Eq. 5.4. The value of this quantity is zero in isotropic turbulence and found to be 0.15 in two-dimensional boundary layers [125].

\[
A1 = \frac{\sqrt{(\tau_{xy})^2 + (\tau_{xy})^2}}{2 \overline{\rho} k}
\]  

5.4

For this case computations are performed at a Reynolds number of 4.0x10^6 with a freestream Mach number of 0.1322 and angle of attack of 10°. Predictions are compared with wind tunnel measurements [125]. Figures 5-41, 5-42 show the variation of Townsend’s structural parameter in the normal direction from the solid surface at two circumferential locations. In the figures, \( \delta \) is defined to be the local boundary layer thickness. Predictions are in good agreement with the measurements especially at \( \phi = 120° \). Some discrepancies are observed at \( \phi = 150° \). But one has to keep in mind that Reynolds stresses used to compute Townsend’s parameter are not aligned with the body axis system but aligned with a “local freestream velocity coordinate system” [125]. Here the origin of the system is on the surface of the body, with the x-axis aligned with the
local freestream velocity at the edge of the boundary layer, the y-axis points radially outward from the surface, and the z-axis completing the right-hand rule. It is evident that the origin and the orientation of this axis system changes at every location on the body. Any small discrepancy in the boundary layer thickness or the direction of the freestream velocity at the edge of the boundary layer will result in a different coordinate system. Since turbulence is anisotropic near solid walls, any change in orientation will result in a different Reynolds stress tensor, hence lead to some discrepancies between the predictions and measurements.

Figure 5-41: Townsend’s Structural Parameter (Re=4.0x10^6, α=10°, T_u=0.03%, Re_T=150, M=0.1322, x/L=0.7, φ=120°)

---

---
5.3.2 Grid Refinement

The flow over a prolate spheroid at high incidence angle experiences strong pressure gradients in the circumferential directions. Strong favorable pressure gradients on the windward side cause rapid flow acceleration while strong adverse pressure gradients on the lee side lead to cross-flow separation. Therefore, the computational mesh used for solutions must also have sufficient resolution in the circumferential direction in order to capture the physics of this complex flow. Circumferential velocity comparisons (Figures 5-24 to 5-38) showed that there are noticeable discrepancies between the predictions and measurements at the separated regions (Figures 5-28, 5-33 and 5-38), which was thought to occur due to insufficient grid resolution in the circumferential
direction. Hence, a grid refinement study is performed by increasing the number of nodes in the circumferential and axial directions. Grid resolution in the normal direction is kept unchanged. The resulting mesh has 4.7 million tetrahedral cells with 51308 surface elements on the prolate spheroid while the coarse mesh had one million tetrahedral cells with 7064 surface elements. A sectional view of the fine mesh is displayed in Figure 5-43. A new solution is performed using this mesh at a Reynolds number of 6.5x10^6, a Mach number of 0.1322 and an angle of attack of 30°. Freestream turbulence quantities are obtained using a turbulence intensity of 0.1%, a turbulent Reynolds number of 150 and assuming the turbulence is isotropic. Surface skin friction lines obtained for this case are displayed in Figure 5-44.

Figure 5-43: Sectional view of the fine mesh for 6:1 prolate spheroid (4.7 million cells)
Figure 5-44 shows sharper separation lines when compared to the coarse mesh solution (Figure 5-4). Between the primary and the secondary separation lines, skin friction lines seem to diverge from each other. Just as the convergence of skin friction lines means separation, their divergence indicates reattachment. This is also visualized better in the fine mesh solution. Circumferential locations of the separation points also seem to be different for both cases. Table 5-1 shows the circumferential locations of primary and secondary separation points at \( x/L = 0.738 \) obtained by fine and coarse meshes and the corresponding wind tunnel measurements [122]. Coarse mesh solutions overpredicted the primary separation location by nearly 15 degrees but could capture the secondary location with a discrepancy of roughly three degrees. Grid refinement improved the results considerably. The solution obtained with the fine mesh predicted the primary separation location with an error of two degrees and the secondary separation location almost at the same point with the measurements.

Figure 5-44: Surface skin friction lines (Re=6.5x10^6, \( \alpha=30^\circ \), Tu=0.1\%, Re_T=150, M=0.1322)
Figures 5-45, 5-46 and 5-47 display the circumferential velocity distributions at different axial stations. Compared to the coarse mesh results (Figures 5-5, 5-6, 5-7) the reverse flow regions are considerably bigger indicating that the fine mesh conserved the vortical flowfield generated after cross-flow separation better than the coarse mesh. In addition to this, flow reattachments (non-negative flow regions close to the wall) are clearer in fine mesh solutions. Details of cross-flow separation, flow reversal and reattachment can be seen from Figures 5-48, 5-49, and 5-50 where circumferential boundary layer profiles are displayed for different axial stations. Profiles are plotted with ten degree increments in the circumferential direction such that first profile is at $\phi=100^\circ$ while last profile is at $\phi=170^\circ$.

### Table 5-1: Primary and secondary separation locations $x/L = 03.738$

<table>
<thead>
<tr>
<th>Circumferential Location of Primary Separation (degrees)</th>
<th>RSM (Coarse Mesh)</th>
<th>RSM (Fine Mesh)</th>
<th>Measurement</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$\sim 123$</td>
<td>$\sim 106$</td>
<td>$\sim 108$</td>
</tr>
<tr>
<td>Circumferential Location of Secondary Separation (degrees)</td>
<td>$\sim 159$</td>
<td>$\sim 156$</td>
<td>$\sim 156$</td>
</tr>
</tbody>
</table>
Figure 5-45: Circumferential Velocity Distribution (Re=6.5x10^6, α=30°, M=0.1322, Tu=0.1%, ReT=150, x/L=0.4839)

Figure 5-46: Circumferential Velocity Distribution (Re=6.5x10^6, α=30°, M=0.1322, Tu=0.1%, ReT=150, x/L=0.6571)
At \( x/L = 0.4839 \) (Figure 5-48) the profile at \( \phi = 120^\circ \) seems attached while the profile at \( \phi = 130^\circ \) is completely separated. A small reattachment region can be seen close to the wall at \( \phi = 140^\circ \) but it quickly separates again. As one moves downstream toward the end of the prolate spheroid the primary separation line moves toward the windward side while the secondary separation line moves toward the top of the body. The reattachment region between the separation lines also grows. This situation is clearly displayed in Figure 5-49 \((x/L = 0.6571)\). Unlike at \( x/L = 0.4839 \), the profile at \( \phi = 120^\circ \) is separated and the reattached region can be seen at \( \phi = 130^\circ \) and \( \phi = 140^\circ \). The drastic growth of boundary layer thickness after the separation is also evident from these figures. The same trend is also observed at \( x/L = 0.8279 \) (Figure 5-50).
Figure 5-48: Circumferential Boundary Layer Profiles (Re=6.5\times 10^6, \alpha=30^\circ, M=0.1322, 
T_u=0.1\%, \text{Re}_T=150, x/L=0.4839)

Figure 5-49: Circumferential Boundary Layer Profiles (Re=6.5\times 10^6, \alpha=30^\circ, M=0.1322, 
T_u=0.1\%, \text{Re}_T=150, x/L=0.6571)
Results presented have showed that grid refinement improved the solution qualitatively. In order to see the quantitative improvements, results are compared with wind tunnel measurements [122] in Figures 5-51 to 5-65. For completeness, the coarse mesh results are also included in the plots. Comparisons include circumferential velocity predictions at different axial stations ($x/L = 0.4839, 0.6571, 0.8279$) and at different circumferential angles ($\varphi = 30, 60, 90, 120$ and $150$ degrees). Again $\varphi = 0$ corresponds to the bottom of the spheroid, and $\varphi = 180$ corresponds to the top.

![Figure 5-50: Circumferential Boundary Layer Profiles (Re=6.5x10^6, $\alpha=30^\circ$, M=0.1322, $T_u=0.1\%$, Re_T=150, x/L=0.8279)](image)
5.3.2.1 Circumferential Velocity Distribution at $x/L = 0.4839$

Figures 5-51 to 5-55 display the variation of circumferential velocity in the normal direction to the spheroid at different circumferential angles. At $\varphi = 30^\circ$ the fine mesh did not improve the solution much and there are still some discrepancies between the predictions and measurements. At $\varphi = 60^\circ$, $90^\circ$ and $120^\circ$ the coarse mesh results were already in good agreement with the wind tunnel data and grid refinement slightly improved them. But the greatest improvement occurred at $\varphi = 150^\circ$. Predictions closely matched the measurements especially close to the body. There are still some discrepancies in the vicinity of the positive peak but the improvement is notable.

Figure 5-51: Circumferential Velocity, $x/L = 0.4839$, $\varphi = 30^\circ$ ($Re=6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$)
Figure 5-52: Circumferential Velocity, $x/L = 0.4839$, $\varphi = 60^\circ$ (Re=$6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$)

Figure 5-53: Circumferential Velocity, $x/L = 0.4839$, $\varphi = 90^\circ$ (Re=$6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$)
Figure 5-54: Circumferential Velocity, $x/L = 0.4839$, $\varphi = 120^\circ$ (Re=6.5x10$^6$, $\alpha=30^\circ$, $M=0.1322$)

Figure 5-55: Circumferential Velocity, $x/L = 0.4839$, $\varphi = 150^\circ$ (Re=6.5x10$^6$, $\alpha=30^\circ$, $M=0.1322$)
5.3.2.2 Circumferential Velocity Distribution at $x/L = 0.6571$

The variation of circumferential velocity in the normal direction for $x/L = 0.6571$ are plotted in Figures 5-56 to 5-60. Comparisons are very similar to those at $x/L = 0.4839$. The negative velocity region can be observed at $\varphi = 120^\circ$ for the fine mesh solution. This phenomenon was not captured by the coarse mesh. Improvement at $\varphi = 150^\circ$ is even more notable when compared to $x/L = 0.4839$. Predictions follow measurements very closely at every point.

Figure 5-56: Circumferential Velocity, $x/L = 0.6571$, $\varphi = 30^\circ$ (Re=6.5x10$^6$, $\alpha=30^\circ$, $M=0.1322$)
Figure 5-57: Circumferential Velocity, \( x/L = 0.6571, \varphi = 60^\circ \) (\( Re=6.5\times10^6, \alpha=30^\circ, M=0.1322 \))

Figure 5-58: Circumferential Velocity, \( x/L = 0.6571, \varphi = 90^\circ \) (\( Re=6.5\times10^6, \alpha=30^\circ, M=0.1322 \))
Figure 5-59: Circumferential Velocity, $x/L = 0.6571$, $\varphi = 120^\circ$ (Re=$6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$)

Figure 5-60: Circumferential Velocity, $x/L = 0.6571$, $\varphi = 150^\circ$ (Re=$6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$)
5.3.2.3 Circumferential Velocity Distribution at $x/L = 0.8279$

The circumferential velocity plots for $x/L = 0.8279$ are displayed in Figure 5-61 to Figure 5-65. As in the previous cases the biggest improvement is at $\phi = 150^\circ$. At $\phi = 120^\circ$ the fine mesh results matched measurements in the near-wall region better than the coarse mesh results.

Figure 5-61: Circumferential Velocity, $x/L = 0.8279$, $\phi = 30^\circ$ (Re=6.5x10$^6$, $\alpha=30^\circ$, M=0.1322)
Figure 5-62: Circumferential Velocity, $x/L = 0.8279$, $\varphi = 60^\circ$ ($Re=6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$)

Figure 5-63: Circumferential Velocity, $x/L = 0.8279$, $\varphi = 90^\circ$ ($Re=6.5\times10^6$, $\alpha=30^\circ$, $M=0.1322$)
Figure 5-64: Circumferential Velocity, $x/L = 0.8279$, $\varphi = 120^\circ$ (Re=$6.5 \times 10^6$, $\alpha=30^\circ$, M=0.1322)

Figure 5-65: Circumferential Velocity, $x/L = 0.8279$, $\varphi = 150^\circ$ (Re=$6.5 \times 10^6$, $\alpha=30^\circ$, M=0.1322)
Finally, figure 5-66 displays the circumferential pressure distribution at $x/L = 0.6062$. Predictions are compared with the corresponding wind tunnel measurements [122]. Pressure distribution can be used to indicate flow separation but it is not a good indicator of separation location [124] because pressure is strongly dependent on the entire flowfield rather than a local phenomenon such as separation. According to Figure 5-66 the coarse mesh solution matched the measurements up to 80 degrees but showed disagreements after this location. Mesh refinement improved the predictions considerably. The fine mesh solution is in very good agreement with experiment; it successfully captured the flat region which is supposed to occur in the separated region. But there are discrepancies between 140° and 160°. The sudden drop in pressure after the flat region is caused by streamwise vortices and the rise in pressure after this point is an indicator of secondary separation. According to measurements there is a huge pressure gradient occurring over a very small distance. Numerical prediction could not capture this properly mainly due to grid spacing requirements. Although the grid used was sufficiently fine to capture the flow physics, additional refinement may be necessary in the vicinity of this region.
Overall, the grid refinement considerably improved the solutions except at $\phi = 30^\circ$. Predictions failed to follow measurements closely at this location although they successfully did even at separated regions where the flowfield is highly vortical and very complex. It is possible that at $\phi = 30^\circ$ the wind tunnel data contain measurement errors, which may explain the slight disagreement between computations and measurements. Combined with a sufficiently fine mesh, the Reynolds Stress Model used in this dissertation proved to be very powerful when handling complex turbulent flow conditions such as the above cases.

In order to see the effect of mesh refinement on the turbulence structure, the variation of turbulent kinetic energy normalized by the freestream kinetic energy is plotted at different axial stations in Figures 5-67, 5-68 and 5-69. It is known from wind tunnel measurements [123] that cross-flow separation moves a sheet of high turbulence.
flow from the boundary layer toward the vortex core. To be able to visualize this effect secondary streamlines are added to the figures. It is clear from the figures that right after the separation a mass highly turbulent fluid is drawn away from the body toward the vortex core. After that, the turbulent fluid also moves with the streamwise vortex.

Figure 5-67: Variation of Turbulent Kinetic Energy (Re=6.5x10^6, α=30°, M=0.1322, T_0=0.1%, Re_T=150, x/L=0.4839)
Figure 5-68: Variation of Turbulent Kinetic Energy (Re=6.5x10^6, \( \alpha = 30^\circ \), M=0.1322, \( T_u = 0.1\% \), Re_T=150, x/L=0.6571)

Figure 5-69: Variation of Turbulent Kinetic Energy (Re=6.5x10^6, \( \alpha = 30^\circ \), M=0.1322, \( T_u = 0.1\% \), Re_T=150, x/L=0.8279)
5.4 Turbulent Flow Simulations around a Sphere

A sphere, like a prolate spheroid, is very simple in geometry but can induce complex three-dimensional flow fields. Therefore, the current RSM method was also used to perform a turbulent flow solution around a sphere. An unstructured mesh with 3.8 million tetrahedral cells was generated for the computations. The cells are clustered in the vicinity of the solid boundary for sufficient boundary layer resolution. Here the non-dimensional distance $y^+$ of the first cells is on the order of one similar to the grids constructed for prolate spheroid. The sectional view of the computational mesh generated for the problem and a close-up view of the cells near the solid boundary are displayed in Figures 5-70 and 5-71.

Figure 5-70: Sectional view of the mesh for a sphere (3.8 million cells)
For the sphere solution, the freestream Reynolds number and freestream Mach number are taken to be $1.14 \times 10^6$ and 0.1763, respectively. The freestream turbulence is assumed to be isotropic with an intensity of 0.45%. The turbulent Reynolds number at freestream is assumed to be 150. Figure 5-72 shows the circumferential pressure distribution and comparison with experimental data [129]. The results are in very good agreement with the measurements with slight discrepancies between 90 and 120 degrees. Such discrepancies were also observed by the LES solutions in [118].

Figure 5-71: Close-up view of the cells near the solid boundary (sphere) (3.8 million cells)
As in the prolate spheroid case, flow separation is analyzed by plotting skin friction lines. In addition to this, the circumferential skin friction distribution is also computed and plotted. Figure 5-73 shows the skin friction lines along with the velocity contours and Figure 5-74 shows the circumferential skin friction distribution and comparison with the measurements [129].

![Figure 5-72: Circumferential Pressure Distribution (Re=1.14x10^6, M=0.1763, Tu=0.45%, Re_T=150)](image)
It is clear from Figure 5-73 that the skin friction lines stop at a circumferential angle of approximately 120 degrees, which indicates the separation location. Boundary layer thickening beyond this point is also evident from the velocity distribution contours. The predicted separation location is very close to the experimental value of 120° [129]. This is also supported by Figure 5-74 where the computed skin friction coefficient is in very good agreement with the experiment [129] with some discrepancies in the vicinity of 90 degrees. These differences are due to laminar-turbulent transition which is not modeled by the current numerical method.
Taneda [130], through his flow visualizations, suggested a mechanism for vortex shedding in the wake of a sphere. He observed that a vortex sheet separating from the sphere forms a pair of strong streamwise vortices. Such a behavior is observed in Figure 5-75 which shows a time averaged iso-vorticity surface. This structure was also observed in the LES results by Jindal et al [118].
In order to analyze the turbulent structure of the flow, components of the turbulence anisotropy tensor \( a_{ij} \) (Eq. 2.39) are plotted and displayed. Figures 5-76, 5-77 and 5-78 display \( a_{xx} \), \( a_{zz} \), and \( a_{xz} \) respectively. In isotropic turbulence, these quantities take the value of zero. The large deviations from zero mean non-isotropy is present in the turbulence structure. It is clear from these figures that the flow is highly anisotropic. The figures clearly support the notion of employing anisotropic turbulence models, such as RSM, for accurate prediction of three-dimensional separated flow. Traditional algebraic, one or two-equation model turbulence models could not predict these solutions.

Figure 5-75: Iso-vorticity surface in the wake of a sphere (\( \text{Re}=1.14\times10^6 \), \( M=0.1763 \), \( T_u=0.45\% \), \( \text{Re}_f=150 \))
Figure 5-76: $a_{xx}$ Contours (Re=1.14x10^6, M=0.1763, Tu=0.45%, Re_f=150)

Figure 5-77: $a_{zz}$ Contours (Re=1.14x10^6, M=0.1763, Tu=0.45%, Re_f=150)
Figure 5-78: $a_{\infty}$ Contours (Re=1.14x10$^6$, M=0.1763, $T_u$=0.45%, Re$_T$=150)
5.5 Turbulent Flow Simulations around a Bell 214ST Fuselage

In the previous sections RSM has been tested for flows over simple geometries such as a prolate spheroid and a sphere, and it was shown that the model can handle three-dimensional separated flows where most of the eddy viscosity models would be insufficient. In this section RSM is employed for simulation of turbulent flow around a Bell 214ST fuselage. Solutions around a Bell 214ST fuselage are obtained using an unstructured mesh composed of 2.9 million tetrahedral cells. The $y^+$ values at the wall-adjacent cells were on the order of 40. The sectional views of the computational mesh generated for the problem are displayed in Figure 5-79 Figure 5-80.

---

Figure 5-79: Sectional view of the computational mesh (Bell 214ST) (2.9 million cells)
In order to include the rotors in the simulation, the main and tail rotors of the helicopter are modeled using actuator disks where a rotor is represented as a thin disk across which pressure undergoes a discontinuity while other variables remain continuous [18]. Figure 5-81 shows the actuator disks along with the fuselage. The pressure jump across the disk can be computed using a momentum theory or a blade element theory. Blade element theory, being superior to momentum theory due to its more accurate model of the disk loads, is considered to be the best disk model and has been successfully employed for the predictions of steady and unsteady thrust response of the FANTAIL™ of RAH66 Comanche helicopter in hover, forward flight and sideward flight [27, 34]. But it requires the knowledge of blade and airfoil parameters. Without the knowledge of such
parameters, momentum theory with a linear loading can be considered as a first approximation [131].

In the present work, momentum theory with a linear load distribution is employed for the modeling of main and tail rotors. The main rotor thrust is equated to the maximum weight of the helicopter and tail rotor thrust is approximated using a simple momentum analysis [18].

Figure 5-81: Actuator disks for rotor modeling

Results include high Reynolds number turbulent flow over an isolated Bell 214ST fuselage for three flight conditions and helicopter simulations with rotors modeled using linear momentum theory. The freestream Reynolds number and freestream Mach number
are set to $1.5\times10^6$ per ft and 0.2322, respectively. The three isolated fuselage cases are low angle of attack cruise, high angle of attack, and high yaw angle. Helicopter simulations with rotors are performed only for low angle of attack cruise condition. Numerical solutions are obtained after the L2 norm of the relative residual dropped four orders of magnitude.

### 5.5.1 Low Angle of Attack Cruise Condition

For this numerical solution the angle of attack is set to -2.28 degrees. Figure 5-82 shows the surface pressure distribution and Figure 5-83 shows the comparison of dorsal centerline pressure distribution with the experimental data [132]. The predictions are in good agreement with the measurements.

![Figure 5-82: Surface Pressure Distribution ($\alpha = -2.28^\circ$, $\psi = 0^\circ$)](image-url)
In Figure 5-83 $x$ is the streamwise distance from the nose of the aircraft and $\psi$ is the yaw angle.

In order to analyze the turbulent structure of the flow, normalized $\tau_{xz}$ and $\tau_{yz}$ distributions are plotted on the corresponding $x$-$z$ and $y$-$z$ planes, and displayed in Figures 5-84 and 5-85. Normalization is performed by dividing the quantity by mean density and turbulent kinetic energy. Isotropic turbulence is achieved when the diagonal terms of the Reynolds stress tensor are equal to each other and the off-diagonal terms are equal to zero. Thus in isotropic turbulence normalized $\tau_{xz}$ and $\tau_{yz}$ become zero. The inhomogeneity and anisotropy of turbulence is evident from these figures where the contour quantities are non zero and change spatially.
Figure 5-84: Normalized $\tau_{xz}$ contours ($\alpha = -2.28^\circ$, $\psi = 0^\circ$)

Figure 5-85: Normalized $\tau_{yz}$ contours ($\alpha = -2.28^\circ$, $\psi = 0^\circ$)
5.5.2 High Angle of Attack Condition

For this case the angle of attack is set to 17.4 degrees. Figure 5-86 shows the surface pressure distribution and Figure 5-87 shows the comparison of dorsal centerline pressure distribution with the experimental data [132]. The predictions are in good agreement with the experiment. The expansion around the canopy region is much stronger than the previous case but RSM gives good agreement even where the expansions are quite abrupt.

Figure 5-86: Surface Pressure Distribution ($\alpha = 17.4^\circ$, $\psi = 0^\circ$)
Normalized $\tau_{xz}$ and $\tau_{yz}$ distributions on the corresponding $x$-$z$ and $y$-$z$ planes are displayed in Figures 5-88 and 5-89. The effect of flow angle of attack on the Reynolds stresses can be seen. In order to check the effectiveness of RSM, Reynolds stresses are recomputed using the Boussinesq approximation with an eddy viscosity computed using Eq. 2.24, 2.25 and 2.26. Figure 5-90 shows the normalized $\tau_{xz}$ computed with this method. Comparison of figures 12 and 14, clearly shows that Reynolds stresses and mean strain rates are grossly misaligned. This suggests that turbulence models based on the Boussinesq approximation might perform poorly for this flow and warrants the use of RSM.
Figure 5-88: Normalized $\tau_{xz}$ contours ($\alpha = 17.4^\circ, \psi = 0^\circ$)

Figure 5-89: Normalized $\tau_{yz}$ contours ($\alpha = 17.4^\circ, \psi = 0^\circ$)
5.5.3 High Yaw Angle Condition

For this case the yaw angle is set to 16.4 degrees and angle of attack is set to -1.6 degrees. The surface pressure distribution and a comparison of dorsal centerline pressure distribution with the experimental data [132] are displayed in Figures 5-91 and 5-92. A non-symmetric pressure distribution is especially evident on the elevator. Pressure predictions are in good agreement with experiment except in the tailboom region. Here the predicted pressure is higher than the measurements. Such a behavior was also observed in Navier-Stokes simulations by Narramore and Brand [8]. The discrepancy in the tail boom region might be due to the crossflow coming through the engine exhaust which was not included in our computational simulation.
Figure 5-91: Surface Pressure Distribution ($\alpha = -1.6^\circ$, $\psi = 16.4^\circ$)

Figure 5-92: Dorsal Centerline Pressure Distribution ($\alpha = 1.6^\circ$, $\psi = 16.4^\circ$)
Normalized $\tau_{xz}$ and $\tau_{yz}$ distributions for this case are displayed in Figures 5-93 and 5-94. The effect of side flow on the Reynolds stresses can be observed.

Figure 5-93: Normalized $\tau_{xz}$ contours ($\alpha = -1.6^\circ$, $\psi = 16.4^\circ$)

Figure 5-94: Normalized $\tau_{yz}$ contours ($\alpha = -1.6^\circ$, $\psi = 16.4^\circ$)
5.5.4 Helicopter Simulation with Rotors

In this case main and tail rotors, modeled using momentum theory with linear load distribution, are included for the low angle of attack cruise condition case. Here the main rotor thrust is taken as 17,500 lbs. and tail rotor thrust is approximated to be 1104 lbs after a simple trim analysis using momentum theory. In order to see the effect of the rotors on the flowfield, the pressure distributions at $x/L = 0.32$ and $x/L = 0.95$ are plotted with and without rotors in Figures 5-95, 5-96, 5-97 and 5-98, respectively. In the figure captions $T$ stands for main rotor thrust and $T_{tr}$ stands for tail rotor thrust. The effect of the main rotor and also the tail rotor can be observed from these figures. The contours of normalized $w$ velocity (mean velocity component in z-direction) on the main rotor plane are shown in Figure 5-99. For comparison purposes solution without the rotors is also included in Figure 5-100.

Figure 5-95: Pressure Distribution at $x/L = 0.32$. ($\alpha=-2.28^\circ$, $\psi=0^\circ$, $T=17500$ lbs, $T_{tr}=1104$ lbs)
Figure 5-96: Pressure Distribution at $x/L = 0.32$. ($\alpha = -2.28^\circ$, $\psi = 0^\circ$, without rotors)

Figure 5-97: Pressure Distribution at $x/L = 0.95$. ($\alpha = -2.28^\circ$, $\psi = 0^\circ$, $T=17500$ lbs, $T_r=1104$ lbs)
Figure 5-98: Pressure Distribution at $x/L = 0.95$. ($\alpha = -2.28^\circ$, $\psi = 0^\circ$, without rotors)

Figure 5-99: Normalized $w$ distribution at the main rotor plane ($\alpha = -2.28^\circ$, $\psi = 0^\circ$, $T = 17500$ lbs, $T_r = 1104$ lbs)
The induced velocity due to the pressure jump across the main rotor disk is observed in these figures. One can also see adjacent positive and negative $w$ values in the vicinity of the rotor perimeter in Figure 5-99. This clearly implies the tip vortex formation. This vortical flowfield is then convected downstream. It is clear that the presence of the rotors not only induces additional flow gradients but also generates a vortical flowfield which travels downstream. This induced flow will change the turbulence structure. In order to analyze the effects of rotors on turbulence, normalized $\tau_{xz}$ and $\tau_{yz}$ distributions are plotted in Figures 5-101 and 5-102. Solutions without rotors were previously displayed in Figures 5-84 and 5-85. Comparing Figures 5-84 and 5-101 the effect of the main rotor on the turbulence structure can be observed. Here the effects
are mainly due to additional flow gradients because tip vortices are weaker in this plane. The effects of tip vortices can be analyzed by comparing Figure 5-85 and Figure 5-102. It is evident that induced vortical flow increases the turbulence inhomogeneity and anisotropy. In order to see the effect of the tail rotor and the convecting tip vortices, the normalized $\tau_{yz}$ distributions are plotted at $x/L = 0.95$ with and without rotors in Figures 5-103 and 5-104. It is clear from these figures that the tail rotor has a small effect on the turbulence structure. But other than that one can easily observe some contours away from the helicopter in Figure 5-103 which are not seen in Figure 5-104. This shows the turbulence induced by the downstream traveling tip vortices.

Figure 5-101: Normalized $\tau_{xz}$ contours ($\alpha=-2.28^\circ$, $\psi=0^\circ$, $T=17500$ lbs, $T_{tr}=1104$ lbs)
Figure 5-102: Normalized $\tau_{yz}$ contours at $x/L = 0.32$ ($\alpha$=-2.28°, $\psi$=0°, $T$=17500 lbs, $T_{tr}$=1104 lbs)

Figure 5-103: Normalized $\tau_{yz}$ contours at $x/L = 0.95$ ($\alpha$=-2.28°, $\psi$=0°, $T$=17500 lbs, $T_{tr}$=1104 lbs)
Drag predictions are performed for the cases studied above and the results are compared with the wind tunnel data [132], Bell simulations of [8] around their most complex configuration including the elevator and vertical tail, and LES predictions of [15]. In the present work, drag is computed by integrating the pressure and surface skin friction over the fuselage surface. Since the mesh was not sufficient to resolve the entire boundary layer, surface skin friction is computed using Spalding’s law of the wall [133] during the post processing. In the predictions a correction to the force in the longitudinal direction was made to take into account the faired-over inlet faces in the computation,
which were open in the wind tunnel model. The dynamic head pressure was integrated over the inlet area, and subtracted from the longitudinal force component. The drag predictions in [8] contain pressure drag only while the predictions of [15] contain both pressure drag and total drag. Here total drag was computed by evaluating the velocity deficit in the wake of the helicopter. Table 5-2 shows a list of results for drag from wind tunnel measurements, Bell simulations, LES predictions and present solutions. For completeness drag predicted for the helicopter configuration with rotors is also displayed. The results showed reasonable agreement with the wind tunnel data. For the low angle of attack cruise case the RSM solution predicted a drag force with a 17.6 % error which is still better than the pressure drag prediction of [8], which employed Baldwin-Lomax turbulence model, and much better than the LES predictions of [15] which uses a very coarse mesh to reduce the CPU time. A similar trend is also observed for the high angle of attack case. The present technique predicted a total drag with an error of 9.7 %. This is less than the error of the Bell simulation, which will increase once the viscous drag is added. For the high yaw angle case the total drag is underpredicted with a 14.4 % discrepancy, which is still a reasonable error considering the differences between the wind tunnel and the computation geometries. From Table 5-2 one can also see that the presence of the main and tail rotors increase the pressure drag by 3 % and total drag by 2.8 %. This is an expected result because the flow condition is a high speed forward flight and the rotor wakes barely interacts with the fuselage.

Overall, RSM predicted fuselage drag with reasonable accuracy although the computational mesh was not sufficient to resolve the boundary layer. Considering some
of the methods used in the HELIFUSE program miscalculated drag by nearly 100% error (see Figure 1-9), one can better understand the effectiveness of RSM.

<table>
<thead>
<tr>
<th></th>
<th>α (degrees)</th>
<th>ψ (degrees)</th>
<th>C_D</th>
<th>% Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>wind tunnel data</td>
<td>-2.28</td>
<td>0</td>
<td>0.078</td>
<td>N/A</td>
</tr>
<tr>
<td>(total drag)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bell Simulations</td>
<td>-2.28</td>
<td>0</td>
<td>0.093</td>
<td>18.9</td>
</tr>
<tr>
<td>(pressure drag)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>RSM simulation</td>
<td>-2.28</td>
<td>0</td>
<td>0.074</td>
<td>5.22</td>
</tr>
<tr>
<td>(pressure drag)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>RSM simulation</td>
<td>-2.28</td>
<td>0</td>
<td>0.092</td>
<td>17.6</td>
</tr>
<tr>
<td>(total drag)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>LES prediction</td>
<td>-2.28</td>
<td>0</td>
<td>0.082</td>
<td>5.3</td>
</tr>
<tr>
<td>(pressure drag)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>LES prediction</td>
<td>-2.28</td>
<td>0</td>
<td>0.106</td>
<td>35.4</td>
</tr>
<tr>
<td>(total drag)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>RSM simulation</td>
<td>-2.28</td>
<td>0</td>
<td>0.076</td>
<td>N/A</td>
</tr>
<tr>
<td>(with rotors)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>RSM simulation</td>
<td>-2.28</td>
<td>0</td>
<td>0.094</td>
<td>N/A</td>
</tr>
<tr>
<td>(with rotors)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>wind tunnel data</td>
<td>17.4</td>
<td>0</td>
<td>0.111</td>
<td>N/A</td>
</tr>
<tr>
<td>(total drag)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bell Simulations</td>
<td>17.4</td>
<td>0</td>
<td>0.122</td>
<td>9.8</td>
</tr>
<tr>
<td>(pressure drag)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>RSM simulation</td>
<td>17.4</td>
<td>0</td>
<td>0.102</td>
<td>8.4</td>
</tr>
<tr>
<td>(pressure drag)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>RSM simulation</td>
<td>17.4</td>
<td>0</td>
<td>0.121</td>
<td>9.7</td>
</tr>
<tr>
<td>(total drag)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>wind tunnel data</td>
<td>-1.6</td>
<td>16.4</td>
<td>0.256</td>
<td>N/A</td>
</tr>
<tr>
<td>(total drag)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>RSM simulation</td>
<td>-1.6</td>
<td>16.4</td>
<td>0.200</td>
<td>22.1</td>
</tr>
<tr>
<td>(pressure drag)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>RSM simulation</td>
<td>-1.6</td>
<td>16.4</td>
<td>0.219</td>
<td>14.4</td>
</tr>
<tr>
<td>(total drag)</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
5.5.6 Code Performance

Simulations were performed using an unstructured mesh of 2.9 million tetrahedral cells and required 8.77 GB of memory. Each solution took 6 days on 30 COCOA3 processors. In Ref. [15] LES solutions performed for the same geometry took 60 days on sixteen 800 MHz Pentium III processors for a physical time of 1.25 seconds. But a much coarser mesh with 530,000 cells was used for the solutions. The present RSM solution on this coarse mesh took one day on 30 COCOA3 processors (2.4 GHz). Hence it can be concluded that RSM (3-D problem) is nearly ten times cheaper than LES (4-D problem).

5.6 Parallel Performance

This section of the thesis contains the performance analysis of the parallel algorithm used for computations. Parallel processing is a very powerful tool used in computational fluid dynamics for reducing required CPU time and memory. The approach consists of decomposing a computational domain into subdomains and distributing the computational job over more than one processor to be performed simultaneously. The essential element of parallel processing is data transfer between processes, which is necessary for continuity of flow variables across inner subdomain boundaries and should be performed accurately in order to obtain correct results. The approach is very powerful but the user has to take care of load balancing in order to prevent some processors from idling while the others still work to finish their part. In addition to this, one has to carefully select the number of processors to be employed such that the algorithm should not spend most of the time for data communications.
In order to analyze the performance of the parallel algorithm used in this dissertation sample timing runs are performed for a turbulent flow over a 6:1 prolate spheroid with the coarse (1 million tetrahedral cells) and the fine meshes (4.7 million tetrahedral cells) described in the previous sections. Timing runs are performed on parallel machines COCOA3 and MUFASA [105] using 10, 20, 30 and 40 processors. In this dissertation message passing operations were performed in a synchronous manner [68]. Therefore, the results do not depend on number of processors. Ideally, doubling the number of processors would drop the required CPU time to half of its original value. But as the number of processors increases the code spends more time for data communication. The ratio of communication time to computation time increases, hence the performance of the parallel algorithm degrades. Therefore, scalability [68] becomes very crucial for large parallel systems. For comparison purposes computations performed with ten processors is taken as the base point and speedups were computed relative to this case. Table 5-3 shows relative speedup versus number of processors data for COCOA3. The real speedup shown in this table is computed by comparing the CPU times spent by different number of processors (20, 30, and 40) to that spent by 10 processors. Here the same solution was obtained for each case.

Table 5-3 indicates that real performance departs from the ideal performance as the number of processors increases. For the coarse mesh case increasing the number of processors by four decreased the CPU time 3.33 times. The situation is much better for the fine mesh solution. Here real performance followed ideal performance very closely. Performance comparisons can also be visualized from Figure 5-105.
Relative speedup versus number of processors data for MUFASA is displayed in Table 5-4. For this case performance of the coarse mesh solution is much better than that on COCOA3. Note that, MUFASA has a faster network connection compared to COCOA3. Here increasing the number of processors four times decreased the CPU time 3.6 times. Fine mesh computations could not be performed on MUFASA with 10 processors. This is mainly due to the insufficient memory problem; memory on each processor was not enough to handle the problem. Therefore, this analysis has been left out of this dissertation. Comparison for MUFASA can also be visualized from Figures 5-106.

<table>
<thead>
<tr>
<th>Number of Processors</th>
<th>Relative Speedup (real)</th>
<th>Relative Speedup (ideal)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td>20</td>
<td>1.81</td>
<td>2.0</td>
</tr>
<tr>
<td>30</td>
<td>2.5</td>
<td>3.0</td>
</tr>
<tr>
<td>40</td>
<td>3.33</td>
<td>4.0</td>
</tr>
</tbody>
</table>

Table 5-3: Relative speedup with number of processors (COCOA3)

<table>
<thead>
<tr>
<th>Number of Processors</th>
<th>Relative Speedup (real)</th>
<th>Relative Speedup (ideal)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td>20</td>
<td>1.8</td>
<td>2.0</td>
</tr>
<tr>
<td>30</td>
<td>2.935</td>
<td>3.0</td>
</tr>
<tr>
<td>40</td>
<td>3.798</td>
<td>4.0</td>
</tr>
</tbody>
</table>
Table 5-4: Relative speedup with number of processors (MUFASA)

<table>
<thead>
<tr>
<th>Number of Processors</th>
<th>Relative Speedup (real)</th>
<th>Relative Speedup (ideal)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>1.0</td>
<td>1.0</td>
</tr>
<tr>
<td>20</td>
<td>1.5</td>
<td>2.0</td>
</tr>
<tr>
<td>30</td>
<td>2.25</td>
<td>3.0</td>
</tr>
<tr>
<td>40</td>
<td>3.6</td>
<td>4.0</td>
</tr>
</tbody>
</table>

Figure 5-105: Relative Speedup vs. Number of Processors (COCOA3)
Figure 5-106: Relative Speedup vs. Number of Processors (MUFASA, coarse mesh)
Chapter 6

Summary and Conclusions

Numerical simulations of flowfields around the RAH-66 Comanche helicopter were performed for various flight conditions with and without the FANTAIL™ operating. In the solutions, the FANTAIL™ was modeled using an actuator disk and blade element methods. The first method included the application of momentum theory with uniform loading. The desired fan thrust was achieved by applying a uniform pressure jump across the rotor disk. The second method consisted of coupling blade element theory with computational fluid dynamics, in which the fan thrust was computed as a function of collective pitch setting and local velocity field. Momentum theory with uniform loading provided a preliminary knowledge of the effects of the FANTAIL™ on the overall flowfield. Blade element theory allowed us to obtain a relationship between the fan thrust and collective pitch settings for different flight conditions. These relations were very important for the design and analysis of the flight control system of the aircraft.

Through coupling blade element theory with computational fluid dynamics, the flow through the FANTAIL™ of the Comanche helicopter was simulated at different blade pitch settings and different flight conditions; hover, forward flight and sideward flight. Effects of blade pitch settings, blade twist angle, and forward and sideward speed on the FANTAIL™ were analyzed and simulated in detail. Some of the flowfields studied were practically impossible to handle using conventional momentum theories.
Steady state thrust predictions showed good agreement with the wind tunnel data. The differences were most likely due to geometric differences and use of an inviscid solver, and linear lift curve for blades but the agreement was very good. It was also shown that the ratio of total device thrust (fan + shroud) to fan thrust is about 1.83 and is very nearly independent of collective pitch. This value is very close to the ideal augmentation factor of 2.0. These solutions can be improved by using representative blade lift curves. In general, however, these were very encouraging results and show that CFD can be used to evaluate the static FANTAIL™ control effectiveness.

In the unsteady solutions the pitch angle was changed by 5 degrees from some equilibrium point at a rate of 144 degrees per second and the transient response of fan thrust, and antitorque (yawing) moment were obtained. These relations were important in understanding the directional control sensitivity of the helicopter.

Preliminary unsteady simulations for hover (pitch angle decreasing from 20° to 15°) showed that, there is a nearly 180% overshoot in the unsteady thrust response. This overshoot became more severe at low pitch settings. A 1067% overshoot was observed when the blade pitch angle is increased from 0 to 5 degrees while only 93% overshoot was observed when it is increased from 35 to 40 degrees. In addition to this, convergence to steady state was also more than two times slower for the low pitch angle case. Transient response of yawing moment was very similar to that of thrust in nature. In hover, as expected, the dominant component for the antitorque moment was the fan. The shroud also generated a significant amount of yawing moment, nearly as much as the fan. The difference between the moments generated by fan and shroud, which also grows as
the pitch setting increases, may be a result of the linear lift curve slope assumption made in the blade element theory.

Unlike hover, an oscillatory thrust response with a much smaller overshoot (44%) is observed in forward flight. The main reason for this response can be said to be the separated and highly vortical flowfield occurring in the duct at zero pitch setting, which was the nominal operating condition of the fan in forward flight. Transient response of yawing moment was also quite different than that of thrust. This is because other fuselage components also generated considerable amounts of yawing moments and they also were affected by the pitch settings. Introduction of the forward speed clearly made the vertical tail the dominant element for the antitorque moment. The shroud also created a high amount of yawing moment. The change in collective pitch settings effectively increased the moments generated by the shroud and vertical tail. In fact the effective increase in vertical tail and shroud components of the antitorque moment were much larger than the fan component.

In the second part of the dissertation high Reynolds number turbulent flows around a 6:1 prolate spheroid, a sphere, and a BELL 214ST fuselage were simulated using RSM. The model consists of a numerical solution of the Favre-Reynolds averaged Navier-Stokes equations coupled with the transport equations for Reynolds stresses and the turbulent dissipation rate.

First solutions for the prolate spheroid were performed at a Reynolds number of 6.5x10^6, Mach number of 0.1322, and at an incidence of 30 degrees. Computations were performed using a relatively coarse mesh consisting of one million tetrahedral cells. Initially, the turbulence intensity and turbulent Reynolds number were taken as 0.1% and
The computations could capture the basic flow features such as primary and secondary cross-flow separations.

The effects of freestream turbulence intensity on the flowfield were analyzed by increasing the turbulence intensity to 1% while keeping turbulent Reynolds number the same. Although turbulence intensity was ten times higher it could not delay separation and it was shown that besides turbulence intensity, the value of the turbulent Reynolds number (or turbulence dissipation rate) can drastically affect the evolution of the flow. This was also supported by the third set of solutions where turbulence intensity is increased to 1% while keeping the turbulence dissipation rate same.

Circumferential velocity predictions correlated well with the measurements except at the separated flow regions. This was mainly due to the inability of the coarse mesh in resolving highly vortical flow after separation. Substantial grid refinement greatly improved this deficiency. Circumferential velocity predictions with the fine mesh were in good agreement with measurements even when there was strong separation. Grid refinement also improved pressure and separation location predictions. Refining the mesh to 4.7 million cells decreased a 15° discrepancy to 2° for primary separation and a 3° discrepancy to nearly 0° for secondary separation.

Overall results proved the ability of the current RSM for complex turbulent flows. Even on a coarse mesh the code could capture separation locations and turbulence anisotropy with good accuracy.

The sphere solution was performed at a Reynolds number of 1.14x10^6, and a Mach number of 0.1763. Surface pressure and skin friction predictions agreed well with wind tunnel measurements even in the separated regions. Slight discrepancies were
observed between the computed and measured skin friction coefficients, which were mainly due to the laminar-turbulent transition which was not modeled in the current code. Analysis of the normalized turbulent stresses revealed that the turbulent structure of the flow was highly anisotropic. This supported the necessity of employing anisotropic turbulence models for three-dimensional separated flows.

Numerical solutions for the Bell 214ST fuselage were performed at a Reynolds number of $1.5\times10^6$ per foot and a Mach number of 0.2322. Results for flow over isolated fuselage at three flight conditions and a helicopter configuration with rotors modeled using momentum theory with linear loading were presented. The results were also compared with wind tunnel data and other numerical simulations. Pressure correlations for low angle of attack cruise and high angle of attack conditions showed very good agreement with experiment while some discrepancies were observed on the tail boom region for the high yaw angle case. This may be due to faired-over inlet faces which were open in the wind tunnel model. Normalized Reynolds stress contours plotted for each flow condition clearly showed the anisotropy and inhomogeneity of the current turbulent flow. The effects of flow condition on turbulence were also observed. In an attempt to see the effectiveness of RSM, a Reynolds stress (computed using Boussinesq approximation) is compared with a predicted Reynolds stress. It was concluded that the Reynolds stress tensor does not align with the mean strain rate tensor and this supported the usage of a second moment closure for the predictions.

Full helicopter simulations were performed by modeling the main and the tail rotor as actuator disks. The pressure jumps across the disks were computed using momentum theory with linear loading distribution. The results were compared with the
isolated fuselage results obtained for the same flow conditions. It was observed that the presence of the rotors induces additional flow gradients and tip vortices which definitely change the turbulence structure by increasing the inhomogeneity and anisotropy.

Drag predictions were performed and compared with wind tunnel data and other numerical simulations. Spalding’s law of the wall is used for the surface skin friction predictions. The predictions show reasonable agreement with the wind tunnel data. The discrepancy was 17.6% for the low angle of attack case, while it was 9.7% and 22.1% for the high angle of attack and high yaw angle cases, respectively. However, RSM performed better than the Bell simulations (which employed an algebraic turbulence model) and LES solutions (which employed a much coarser mesh to reduce the required CPU time).

To see its effectiveness, the performance of the present RSM was compared to that of LES. It was concluded that RSM, which represents a three-dimensional problem, is nearly ten times faster than LES, which is a dynamic model thus represents a four-dimensional problem. RSM was also proved to be very effective in computing time averaged quantities.

Finally, the parallel efficiency of the RSM code is analyzed by making sample timing runs on the parallel machines COCOA3 and MUFASA using 10, 20, 30 and 40 processors. Relative speedup versus number of processors data obtained for both machines were tabulated and plotted. For comparisons, computations performed with ten processors were taken as the base point and speedups were computed relative to this case. Timing runs consisted of RSM solutions around a 6:1 prolate spheroid using a coarse (one million cells) and a fine (4.7 million cells) mesh. On COCOA3 the coarse mesh
solutions turned less efficient than the fine mesh case. Increasing the number of processors four times decreased the required CPU time 3.33 times. For the fine mesh solution, real performance followed the ideal performance very closely.

Coarse mesh solutions on MUFASA were more scalable compared to those on COCOA3. This is mainly due to the faster network connection of MUFASA. But, for the fine mesh MUFASA could not perform solutions on 10 processors because of insufficient memory. Therefore, this case had been left out.
Bibliography


25. Pahlke, K., Sides, J., Costes, M., “Towards the CFD Computation of the Complete Helicopter: First Results Obtained by French and German Research


105. http://www.cse.psu.edu/mufasa

106. http://gears.aset.psu.edu/hpc/systems/lionxm

107. http://gears.aset.psu.edu/hpc/systems/lionxl


VITA

Emre Alpman

Emre Alpman was born in 1977 in Ankara, Turkey. He received his B.S. degree from the Aeronautical Engineering Department of the Middle East Technical University, Ankara, Turkey, in 1999 with the first rank. In September 1999, he enrolled in the Graduate Program of the same department and received his M.S. degree in July 2001. In January 2002, he entered the Pennsylvania State University in pursuit of a Ph.D. degree in Aerospace Engineering and continued his graduate studies with a graduate research assistantship in the Aerospace Engineering Department.