

Numerical Investigation of Transitional Flow Past Low Reynolds Number Airfoils and Wings

A thesis

Submitted in Partial Fulfillment of the Requirements for
the Award of the Degree of

Doctor of Philosophy

by

Dattatraya S Kulkarni



**DEPARTMENT OF MECHANICAL ENGINEERING
INDIAN INSTITUTE OF TECHNOLOGY GUWAHATI**

November 2024

DECLARATION

I hereby declare that the research thesis entitled “ **Numerical Investigation of Transitional Flow Past Low Reynolds Number Airfoils and Wings**” submitted by me to the **Indian Institute of Technology Guwahati** for the award of the degree of **Doctor of Philosophy** in Mechanical Engineering is a bonafide research work carried out by me under the supervision of **Prof. Vinayak Kulkarni**, Department of Mechanical Engineering, IIT Guwahati and **Dr. B. N. Rajani**, Chief Scientist, CTFD Division, CSIR-NAL, Bengaluru. Any part of this work has not been submitted for the award of any degree, diploma, associate-fellowship, fellowship or its equivalent to any university or institution.

Dattatraya S Kulkarni

Registration No: 166103024

Department of Mechanical Engineering,

Indian Institute of Technology Guwahati,

Guwahati-781039, Assam, India.

Date:

CERTIFICATE

This is to certify that the thesis entitled “ **Numerical Investigation of Transitional Flow Past Low Reynolds Number Airfoils and Wings**” submitted by Mr. Dattatraya S Kulkarni (Roll No. 166103024) has been carried out under our supervision and this work has not been submitted elsewhere for the award of any other degree or diploma.

Dr. Vinayak Kulkarni

Professor, Department of Mechanical Engineering,
Indian Institute of Technology Guwahati,
Guwahati-781039, Assam, India.

Dr. B. N. Rajani

Chief Scientist, CTFD Division,
CSIR-National Aerospace Laboratories,
Bengaluru-560017, Karnataka, India.

Date:

Acknowledgments

The journey of doctoral study is always a difficult and challenging task. I would never have successfully completed this thesis without the assistance of numerous people who I am indebted to. Their direction, advise, support and contribution have proved invariable along the way.

First and foremost, I would like to express my sincere and profound gratitude to my supervisors Prof. Vinayak Kulkarni, Department of Mechanical Engineering Indian Institute of Technology (IIT) Guwahati and Dr. B. N. Rajani, Chief Scientist, Computational and Theoretical Fluid Dynamics (CTFD) Division, CSIR-National Aerospace Laboratories (CSIR-NAL) Bengaluru for their excellent guidance throughout my research work. I appreciate their contributions of time, support, encouragement and active guidance to make my PhD experience productive and simulating. A very special gratitude goes to both of them since from motivation to successful completion of my research.

This research work would not be possible without the support of IIT Guwahati and CSIR-NAL, Bengaluru. I am grateful to the Director IIT Guwahati and Director CSIR-NAL for providing me the necessary environment and giving me all the facilities for carrying out this research. I express my sincere gratitude to present and past Head of the Mechanical Engineering Department, IIT Guwahati and CTFD Division, CSIR-NAL, Bengaluru for their support for my research and administrative activities.

My sincere and special thanks to the Doctoral Committee members, Prof. Amaresh Dalal, Prof. Ujjwal K. Saha and Prof. Raghavendra Gupta for their insightful discussions, valuable suggestions and encouragements during my research work. My Special gratitude to Prof. Niranjana Sahoo and Prof. Ganesh Natarajan for fruitful interaction with them during initial stage of research work.

I am extremely gratified to all the faculty and staff members of the Mechanical Engineering Department, academic and administrative Department and Guest house of IIT Guwahati for their support and help whenever needed during my stay at IIT Guwahati. My Special thanks to Dr. Soumya Ranjan Nanda and Mr. Anand for their affection and care during my stay in IIT Guwahati and for guiding me in the administrative procedures as a Co-Research scholar. I would like to thank Dr. Gireesh of VSSC, Dr. Deepak, Dr. Prabhat and Dr. Nur Alom who made my stay at IIT Guwahati as an enjoyable and memorable one. I would like to thank all the scientists of CTFD Division, CSIR-NAL for their helpful interactions. The timely help received from Mrs. D. Shobha and Mrs. L. Vijayalaxmi of the CTFD office also acknowledged. I am also thankful to Mrs. Premalatha for her patient reading of the manuscript and giving her valuable suggestions.

“The love of a family is life’s greatest blessings”. In this precious moment of my life, I would like to express my deep sense of gratitude to my parents, Mr. Surendrarao Kulkarni and Mrs. Indirabai for their love, blessings and constant encouragement throughout my life. I am extremely thankful to my loving wife Mrs. Shilpa Kulkarni for taking care of all the family affairs and allowing me to concentrate on my research work. Once again, I express my deepest gratitude to my better-half for her co-operation, inspiration and support during all ups and downs. My special thanks to sweet sons Master. Shripad Kulkarni and Master. Shubham Kulkarni who filled my tense and busy times with moments of pleasure. I would like to thank my sisters Surekha, Karuna, Janhavi, Anasuya and all my brother-in-laws for their love and best wishes. I am also very thankful to my in-laws Mr. Kishore Joshi, Mrs. Kavita Joshi and Mr. Kedar Joshi for their love and moral support during my research work.

I wish to express my deep gratitude to all my teachers, elders, friends and well wishers those who have helped and supported me in various ways during the tenure of my Ph.D. work at IIT Guwahati.

Dattatraya S Kulkarni

Abstract of the Thesis

Low Reynolds number (Re) aerodynamics is important for both natural and man-made flying objects. Over the last few decades there is a growing interest in designing Micro Air Vehicles (MAVs) which are used for both civil and military applications. The small length scale and low Reynolds number ($10^4 - 5 \times 10^5$) flight regime encountered in these classes of air vehicles lead to a complex flow phenomenon. This flow complexity is mainly due to viscous effects, laminar to turbulent transition, formation of the laminar separation bubble (LSB) with turbulent reattachment, which pose a major aerodynamic challenge to designers. The phenomenon of laminar to turbulent flow transition is one of the major area which is not fully understood by the Computational Fluid Dynamics (CFD) community. However, there are different ways in CFD to predict transitional flow namely (i) empirical e^N method based on linear stability theory approach (ii) Direct Numerical Simulation (iii) Large Eddy Simulation (iv) the statistical based RANS models. The present work mainly focuses on predicting the aerodynamics and transition characteristics of airfoils and wings used in MAV application and understanding the influence of different flow and geometry parameters on the performance of these airfoils and wings. In this study, the numerical simulations for Re ranging from 4×10^4 to 2×10^5 are carried out using the RANS based transition models viz. $k_T - k_L - \omega$ and $\gamma - Re_\theta$ SST available in the in-house multi-block incompressible flow solution code 3D-PURLES (**3D Pressure based Unsteady RANS LES**). The RANS based transition models are chosen for this study as they are computationally less expensive with their average flow quantities being reasonably accurate and best suited as a designer approach. This report discusses, the systematic numerical study carried out to investigate the effect of geometrical parameters like camber, thickness and planforms, grid parameters, flow parameters, transition models for low Re

Selig-Donovan (SD) airfoil series on the aerodynamic and transition characteristics for a wide range of angle of attack. The grid size, grid resolution, grid topology, farfield location and other flow parameter like spatial discretisation schemes, freestream turbulence intensity and inlet eddy viscosity ratio which control the accuracy of the flow solution code is fixed based on the sensitivity study carried out at $Re = 6 \times 10^4$ for SD7003 airfoil. The aerodynamic performance parameters (stall angle, lift coefficient and drag coefficient) and transition characteristics (transition onset, separation point, reattachment point and length of the LSB) are validated against the available measurement data. This comparative study indicated that the transition and aerodynamic features of the low Re SD7003 airfoil was predicted better by the $\gamma - Re_\theta$ SST transition model when compared to the $k_T - k_L - \omega$ transition model. The $\gamma - Re_\theta$ SST model because of its better performance was used for all the other simulations for different airfoils and wings carried out in this work. Based on the study carried out to understand the influence of the airfoil camber and thickness in the low Re regime it was inferred that increasing the airfoil camber significantly improved the performance of the airfoil whereas the increasing the thickness did not bring in any significant improvement in the performance of the airfoil. It was observed from this study that increasing the Re from 6×10^4 to 2×10^5 enhanced the aerodynamic performance whereas the transition features were not significantly effected. The numerical simulation past the non-dimensionalized SD7003 rectangular wing using the $\gamma - Re_\theta$ SST transition model is also discussed in detail. In this study, the effect of the different wing semi aspect ratio (sAR) of 0.25, 0.5 & 1 in the low Re regime at $Re = 6 \times 10^4$ is carried out to predict and understand its influence on the aerodynamic and transition characteristics. This study showed that wing with lowest semi aspect ratio of 0.25 is most suited for MAV application as its performance was better in terms of having lower drag, higher maximum lift coefficient, delayed stall angle, negative pitching moment slope and having a larger laminar region as compared to other two wing planforms with higher semi aspect ratio. It was further noted from this study that increasing the Re from 4×10^4 to 1.5×10^5 did not bring a considerable change in the aerodynamic and transition features of the SD7003 rectangular wing with 0.25 semi aspect ratio.

Contents

Acknowledgements	i
Abstract of the thesis	iii
Contents	v
List of Figures	ix
List of Tables	xiii
Nomenclature	xiv
1 Introduction	1
1.1 Background	1
1.2 Objectives of the Present Research Work	6
1.3 Organisation of the Thesis	6
2 Review of Literature	8
2.1 Background	8
2.2 Experimental	8
2.3 Computational	14
2.4 Summary	21
3 Numerical Grid Generation Procedure	22
3.1 Background	22

3.2	Differential-algebraic hybrid method for 2D plane	23
3.3	Governing equations	23
3.4	Bicubic spline fitting for coarse to fine interpolation	25
3.5	Modeling of boundary geometry	25
3.6	Single block to multiblock conversion	29
3.7	3D Grid Generation	30
3.8	Summary	31
4	Mathematical Modelling of Flow Physics	32
4.1	Background	32
4.2	Governing Equations	33
4.3	Transition Modelling	34
4.3.1	Laminar kinetic energy based $k_T - k_L - \omega$ transition model	34
4.3.2	Correlation based $\gamma - Re_\theta$ SST transition model	39
4.4	Finite Volume Method for Flow Solution	46
4.4.1	Control volume and variable arrangement	47
4.4.2	Geometrical parameters	48
4.4.3	Flux balance equation	48
4.4.4	Spatial discretisation	49
4.4.5	Temporal discretisation	51
4.4.6	Finite volume equation	52
4.4.7	Computation of pressure field	53
4.4.8	Boundary conditions	54
4.4.9	Multiblock computation for implicit solvers	56
4.4.10	Parallelisation of the flow solution algorithm	58

4.4.11	Computation scheme	59
4.4.12	Sequence of the multiblock flow solution algorithm	60
4.4.13	Convergence criterion	61
4.5	Summary	62
5	Validation and Sensitivity Study	63
5.1	Background	63
5.2	Computational Details	64
5.2.1	Estimation of Transition Parameters	65
5.3	Validation Study	67
5.4	Sensitivity Study for SD7003 Airfoil	69
5.4.1	Influence of grid quality on flow solution accuracy	69
5.4.2	Influence of flow solution control parameters on solution accuracy	77
5.5	Summary	80
6	Flow Analysis of SD7003 Airfoil	82
6.1	Background	82
6.2	Performance of the Transition Model	82
6.3	Influence of Reynolds Number on the Airfoil Performance	92
6.4	Summary	97
7	Flow Analysis of SD Airfoil Series at Low Re	99
7.1	Background	99
7.2	Influence of Varying Camber on Flow Characteristics	99
7.2.1	Aerodynamic characteristics	99
7.2.2	Transition characteristics	110

7.3	Influence of Varying Thickness on Flow Characteristics	118
7.3.1	Aerodynamic characteristics	118
7.3.2	Transition characteristics	124
7.4	Summary	129
8	Flow Past Low Aspect Ratio Wings	130
8.1	Background	130
8.2	Computational Details	130
8.3	Effect of Wing Aspect Ratio on the Flow Characteristics	132
8.4	Effect of Reynolds Number on the Flow Characteristics	147
8.5	Summary	154
9	Conclusions and Scope for Future Work	155
9.1	Conclusions	155
9.2	Scope for Future Work	159
	Bibliography	160
	Appendices	173
A	Study of Freestream Turbulence Intensity Influence at Low Re	173
A.1	Background	173
A.2	Effect of Tu on Aerodynamic Characteristics	173
A.3	Effect of Tu on Transition Characteristics	179
A.4	Summary	185
	Publications	186

List of Figures

1.1	Characteristics of laminar separation bubble (Courtesy: Xia <i>et al.</i> (2021))	2
3.1	2D grid for different topology for SD7003 airfoil	28
3.2	2-block SD7003 airfoil grid for different topologies	30
3.3	Grid generated for SD7003 rectangular wing	31
4.1	Typical control volume and cell face projection area	47
4.2	Schematic sketch of the different spatial discretisation schemes	51
4.3	Boundary conditions for a two-block computation of flow past an airfoil	56
4.4	Schematic arrangement of domain decomposition	58
4.5	Parallelisation scheme	59
5.1	Boundary conditions used for the airfoil flow simulations	65
5.2	Determination of transition onset based on surface pressure distribution	66
5.3	Determination of transition onset based on normalized Reynolds shear stress	66
5.4	Determination of length of laminar separation bubble (LSB)	67
5.5	Schematic diagrams showing farfield distance (R_{out}) for SD7003 airfoil	70
5.6	Leading edge and trailing edge grid resolution (ΔS) for SD7003 airfoil	72
5.7	Influence of grid near-wall distance (ΔY_{nw}) for SD7003 airfoil	74
5.8	Schematic diagrams and zoomed view near leading and trailing edge	76

6.1	Effect of transition models on the variation of C_p and C_f at different α	87
6.2	Comparison of normalized Reynolds shear stress contours at $\alpha = 4^\circ$	88
6.3	Normalized Reynolds shear stress profiles at $\alpha = 4^\circ$	88
6.4	Comparison of normalized Reynolds shear stress contours at $\alpha = 8^\circ$	89
6.5	Normalized Reynolds shear stress profiles at $\alpha = 8^\circ$	89
6.6	Variation of transition onset and length of LSB for SD7003 airfoil	90
6.7	Effect of transition models on surface streamlines	91
6.8	Comparison of aerodynamic coefficients with angle of attack	91
6.9	Reynolds shear stress contours and streamlines at different Re for $\alpha = 4^\circ$	94
6.10	Effect of Reynolds number on transition characteristics for $\alpha = 4^\circ$	95
6.11	Effect of Reynolds number on aerodynamic characteristics for $\alpha = 4^\circ$	95
6.12	Effect of Reynolds number on transition characteristics	96
6.13	Effect of Reynolds number on aerodynamic characteristics	96
7.1	SD airfoil geometries having different camber	101
7.2	Validation of the aerodynamic performance of SD7090 airfoil at lower Re	102
7.3	Validation of the aerodynamic performance of SD8040 airfoil at lower Re	103
7.4	Validation of the aerodynamic performance of SD7032 airfoil at lower Re	104
7.5	Validation of the aerodynamic performance of SD7090 airfoil at higher Re	105
7.6	Validation of the aerodynamic performance of SD8040 airfoil at higher Re	106
7.7	Validation of the aerodynamic performance of SD7032 airfoil at higher Re	107
7.8	Influence of airfoil camber on aerodynamic performance at $Re = 6 \times 10^4$	108
7.9	Influence of airfoil camber on aerodynamic performance at $Re = 2 \times 10^5$	109
7.10	Influence of camber on aerodynamic parameters at different Re	110
7.11	Influence of camber on Reynolds shear stress at $Re = 6 \times 10^4$	113

7.12	Influence of camber on surface streamlines at $Re = 6 \times 10^4$	114
7.13	Influence of camber on Reynolds shear stress at $Re = 2 \times 10^5$	115
7.14	Influence of camber on surface streamlines at $Re = 2 \times 10^5$	116
7.15	Influence of airfoil camber on transition characteristics at $Re = 6 \times 10^4$	117
7.16	Influence of airfoil camber on transition characteristics at $Re = 2 \times 10^5$	117
7.17	SD airfoil geometries having different thickness	119
7.18	Validation of the aerodynamic performance of SD5060 airfoil at lower Re	120
7.19	Validation of the aerodynamic performance of SD5060 airfoil at higher Re	121
7.20	Influence of airfoil thickness on aerodynamic performance at $Re = 6 \times 10^4$	122
7.21	Influence of airfoil thickness on aerodynamic performance at $Re = 2 \times 10^5$	123
7.22	Influence of thickness on aerodynamic parameters at different Re	124
7.23	Influence of thickness on Reynolds shear stress at $Re = 6 \times 10^4$	126
7.24	Influence of thickness on surface streamlines at $Re = 6 \times 10^4$	126
7.25	Influence of thickness on Reynolds shear stress at $Re = 2 \times 10^5$	127
7.26	Influence of thickness on surface streamlines at $Re = 2 \times 10^5$	127
7.27	Influence of airfoil thickness on transition characteristics at $Re = 6 \times 10^4$	128
7.28	Influence of airfoil thickness on transition characteristics at $Re = 2 \times 10^5$	128
8.1	H-H Grid showing the symmetry plane and wing surface	131
8.2	Boundary conditions used in the present simulations	132
8.3	Reynolds shear stress contours for different wing spans at $\alpha = 0^\circ$	136
8.4	Reynolds shear stress contours for different wing spans at $\alpha = 4^\circ$	137
8.5	Reynolds shear stress contours for different wing spans at $\alpha = 8^\circ$	138
8.6	Reynolds shear stress contours for different wing spans at $\alpha = 16^\circ$	139
8.7	Effect of wing-span on the transition onset location at $Re = 6 \times 10^4$	140

8.8	Volume streamlines for different wing span at $Re = 6 \times 10^4$ and $\alpha = 0^\circ$	140
8.9	Volume streamlines for different wing span at $Re = 6 \times 10^4$ and $\alpha = 4^\circ$	140
8.10	Volume streamlines for different wing span at $Re = 6 \times 10^4$ and $\alpha = 8^\circ$	141
8.11	Volume streamlines for different wing span at $Re = 6 \times 10^4$ and $\alpha = 16^\circ$	141
8.12	Effect of wing-span on the aerodynamic coefficient	142
8.13	Comparison of C_p and C_f for the wing root and tip sections at $\alpha = 0^\circ$	143
8.14	Comparison of C_p and C_f for the wing root and tip sections at $\alpha = 4^\circ$	144
8.15	Comparison of C_p and C_f for the wing root and tip sections at $\alpha = 8^\circ$	145
8.16	Comparison of C_p and C_f for the wing root and tip sections at $\alpha = 16^\circ$	146
8.17	Effect of Reynolds number on the aerodynamic coefficient	149
8.18	Reynolds shear stress contours at different Re for $sAR = 0.25$ and $\alpha = 8^\circ$	150
8.19	Reynolds shear stress contours at different Re for $sAR = 0.25$ and $\alpha = 16^\circ$	151
8.20	Reynolds shear stress contours at different Re for $sAR = 0.25$ and $\alpha = 32^\circ$	152
8.21	Effect of Re on tip vortex formation at different α for wing $sAR = 0.25$	153
A.1	Influence of Tu on aerodynamic characteristics at $Re = 6 \times 10^4$	175
A.2	Influence of Tu on aerodynamic characteristics at $Re = 1.5 \times 10^5$	176
A.3	Influence of Tu on C_p at different angle of attack for $Re = 6 \times 10^4$	177
A.4	Influence of Tu on C_p at different angle of attack for $Re = 1.5 \times 10^5$	178
A.5	Influence of Tu on transition onset location at different Reynolds number	180
A.6	Influence of Tu on LSB features at $Re = 6 \times 10^4$	181
A.7	Influence of Tu on LSB features at $Re = 1.5 \times 10^5$	182
A.8	Surface streamlines around SD7003 airfoil for $Tu = 0.1\%$ at $Re = 6 \times 10^4$	183
A.9	Surface streamlines around SD7003 airfoil for $Tu = 0.1\%$ at $Re = 1.5 \times 10^5$	184

List of Tables

4.1	$k_T - k_L - \omega$ model constants	39
4.2	Model constants for γ and $\tilde{Re}_{\theta t}$ equation	44
5.1	Comparison of flow characteristics for $\alpha = 4^\circ$	68
5.2	Comparison of flow characteristics for $\alpha = 8^\circ$	69
5.3	Influence of farfield distance on flow characteristics	71
5.4	Influence of airfoil leading & trailing edge resolution on flow characteristics	71
5.5	Influence of airfoil near-wall resolution on flow characteristics	73
5.6	Influence of grid topology on flow characteristics	75
5.7	Influence of spatial discretisation scheme on flow characteristics	78
5.8	Influence of freestream turbulence intensity on flow characteristics	79
5.9	Influence of inlet eddy viscosity ratio on flow characteristics	80
6.1	Comparison of separation, reattachment and transition onset location at $\alpha = 4^\circ$. .	86
6.2	Comparison of separation, reattachment and transition onset location at $\alpha = 8^\circ$. .	88

Nomenclature

A_{nb}	Coefficients of the quasilinear equation
C	Chord of the airfoil (m)
C_L	Coefficient of lift
C_{Lmax}	Maximum coefficient of lift
C_D	Coefficient of drag
C_M	Coefficient of moment about quarter chord from the LE of the airfoil
C_p	Non-dimensional pressure coefficient
C_f	Non-dimensional skin friction coefficient
C_w	Mass flux at the west face of the CV (Kg/m^2s)
C_μ	Turbulent viscosity coefficient
CC	Convergence criterion
D_T	Anisotropic near wall dissipation for k_T
D_L	Anisotropic near wall dissipation for k_L
d	Wall distance (m)
F_{length}	Dimensionless function to control the length of transition
F_{onset}	Dimensionless function to control the transition onset location
f_{INT}	Intermittency damping function
$f_{NAT,crit}$	Function incorporates freestream turbulence effects on natural transition
f_W	Inviscid near-wall damping function
f_{SS}	Shear-sheltering damping function
f_ν	Viscous near-wall damping function

f_ω	Boundary layer wake term damping function in ω equation
$f_{\tau,l}$	Time-scalar damping function
I^C	Convective flux (Kg/m^2s)
I^{DC}	Cross diffusive flux across the face m of the CV (Kg/m^2s)
I^{DN}	Normal diffusive flux (Kg/m^2s)
I_m	Total (diffusive + convective) flux (Kg/m^2s)
J	Jacobian of transformation matrix
k	Kinetic energy (m^2/s^2)
k_L	Laminar kinetic energy (m^2/s^2)
k_T	Turbulent kinetic energy (m^2/s^2)
$k_{T,l}$	Effective “large-scale” turbulent kinetic energy (m^2/s^2)
$k_{T,s}$	Effective “small-scale” turbulent kinetic energy (m^2/s^2)
k_{TOT}	Total fluctuation kinetic energy, $k_T + k_L$ (m^2/s^2)
L/D	Lift to drag ratio
$(L/D)_{max}$	Maximum lift to drag ratio
L_{LSB}	Length of laminar separation bubble
P_{kL}	Production of laminar kinetic energy by mean strain rate
P_{kT}	Production of turbulent kinetic energy by mean strain rate
Re	Flow Reynolds number based on airfoil chord
Re_T	Effective turbulence Reynolds number
Re_Ω	Vorticity Reynolds number
Re_T	Turbulent Reynolds number
Re_y	Wall-distance-based turbulent Reynolds number
Re_θ	Momentum thickness Reynolds number
$Re_{\theta c}$	Momentum thickness Re where the intermittency starts to increase
$Re_{\theta t}$	Transition onset momentum-thickness Reynolds number
$\tilde{Re}_{\theta t}$	Local transition onset momentum-thickness Reynolds number

R_{BP}	Bypass transition production term
R_{NAT}	Natural transition production term
R_{out}	Far field radius (m)
$R_{\phi_P}^n$	Residual after n iterations for flow variable ϕ
$RNORM$	A reference value used for normalization
S	Magnitude of mean strain rate tensor ($1/s$)
SU and SP	Linearised source terms
Tu	Freestream turbulence intensity
t	Non-dimensional time
U_∞	Free-stream velocity (m/s)
X_S/C	Separation point
X_R/C	Reattachment point
X_T/C	Transition onset location
y^+	Non-dimensional wall normal distance

Greek Symbols

α	Angle of attack ($^{\circ}$)
α_v	Under-relaxation parameter for velocity field
α_P	Under-relaxation parameter for pressure field
α_T	Effective diffusivity for turbulence dependent variables
β_{BP}	Bypass transition threshold function
β_{TS}	Tollmien-Schlichting threshold function
β_{NAT}	Natural transition threshold function
γ	Intermittency
Δt	Time step size (s)
ΔV	Cell volume (m^3)
δ	Boundary layer thickness (m)
θ	Momentum thickness (m)
λ_{DS}	Blending function used for spatial discretisation
λ_{eff}	Effective (wall-limited) turbulence length scale
λ_T	Turbulence length scale
λ_{θ}	Pressure gradient parameter
μ	Fluid viscosity (Ns/m^2)
μ_l	Laminar viscosity (Ns/m^2)
μ_t	Turbulent or Eddy viscosity (Ns/m^2)
$\mu_{T,s}$	Small-scale turbulent viscosity contribution
$\mu_{T,l}$	Large-scale turbulent viscosity contribution
ρ	Density (kg/m^3)
τ	shear stress (N/m^2)

ϕ	Flow variable
ϕ_P	Field value at node centre
ϕ_{BP}	Model bypass transition parameter
ϕ_{NAT}	Model natural transition parameter
Ω	Absolute value of vorticity ($1/s$)
ω	Specific dissipation rate of turbulence ($1/s$)



Subscripts/Superscripts

i, j, k	Tensor directions, varying from 1 to 3
1, 2, 3	Tensor directions, varying from 1 to 3
sep	Separation
$reattach$	Reattachment
eff	Effective
$wake$	Relative to the wake
0	Initial or previous iteration
C	Convective contribution of the flux
DC	Cross diffusion contribution of the flux
DN	Normal diffusion contribution of the flux
DS	Desired scheme
E, W, N, S, T, B	Neighbouring CV centers - east, west, north, south, top and bottom
e, w, n, s, t, b	Faces of the CV - east, west, north, south, top and bottom
N	Number of grid nodes
nb	Neighbouring nodes
n	Time instant or iteration n
$n + 1$	Time instant $n + 1$
$n - 1$	Time instant $n - 1$
P	Centre of the CV
ϕ	Flow variable
$+$	Variable expressed in wall units

Abbreviations

AR	Aspect Ratio (=Square of the wing span/Wing area)
CDS	Central Difference Scheme
CFD	Computational Fluid Dynamics
CPU	Central Processing Unit
CV	Control Volume
DNS	Direct Numerical Simulation
ERCOFTAC	European Research Community on Flow, Turbulence and Combustion
HLPA	Hybrid Linear Parabolic Approximation
ILES	Implicit-Large Eddy Simulation
LAR	Low Aspect Ratio
LE	Leading Edge
LES	Large Eddy Simulation
LNB	Low speed wind tunnel of the technical University of Braunschweig
LSB	Laminar separation bubble
LST	Linear stability theory
LTPT	Low Turbulence Pressure Tunnel
MAV	Micro Air Vehicle
NA	Not Available
NACA	National Advisory Committee for Aeronautics
NAL	National Aerospace Laboratories
NASA	National Aeronautics and Space Administration
NAV	Nano Air Vehicle
PIV	Particle Image Velocimetry
QUICK	Quadratic Upwind Interpolation for Convection Kinematics

RANS	Reynolds Averaged Navier-Stokes
sAR	Semi Aspect Ratio
SD	Selig-Donovan
SIMPLE	Semi Implicit Method for Pressure-Linked Equation
SIP	Strongly Implicit Procedure
SGS	Sub-Grid Scale
SST	Shear Stress Transport
TE	Trailing Edge
UAV	Unmanned Air Vehicle
UIUC	University of Illinois at Urbana Champaign
URANS	Unsteady Reynolds Averaged Navier-Stokes
VAHL	Vehicle Aero Hydrodynamics Laboratory
WUB	Water tunnel of the technical University of Braunschweig
2D	Two Dimensional
3D	Three Dimensional
3D-PURLES	3D-Pressure based Unsteady Reynolds averaged Navier-Stokes LES

Chapter 1

Introduction

1.1 Background

Low Reynolds number aerodynamics are important for both natural and man-made flying objects. The complexity and the fascinating aerodynamic features encountered during the flight of small birds, bats, insects has generated enormous research interest among biologists and aerospace engineering community. Over the last few decades there is a growing interest in designing aircrafts that are as small as possible, among them are the micro air vehicles (MAVs) which have received special attention for both civil and military applications. MAVs are unmanned autonomous flying machines with small linear dimension and operating at low speed leading to a low Reynolds number flight regime ($10^4 < Re < 5 \times 10^5$). Aerodynamic design of these vehicles reported so far, have employed different kinds of efficient lift generation systems viz., fixed wing (Mueller, 2001; Grameyer and Keennon, 2001), flapping wing (Mueller, 2001; Shyy *et al.*, 1999; Jones *et al.*, 2004), flexible wing (Shyy *et al.*, 1999) and rotary wing (Hein, 2005) and or their combinations. The fixed-wing MAVs are commonly used because they are simple and easy to implement and they usually fly at the upper end of the low Reynolds number regime (Selig *et al.*, 1995; Mueller, 1999; Torres *et al.*, 2004). The stability of these vehicle is another important factor due to its dimensions and weight. The present work is focused on MAVs with fixed wing as it is known to be more stable compared to the flexible wing or flapping wing. These MAVs also need to have the ability to loiter

for a long duration with efficient maneuver capability both in open and confined space. The small length scale and low Reynolds number flight regime pose challenges related to geometry handling and aerodynamic flow analysis required in the design of MAVs. The aerodynamic challenge being that the flow is transitional and dominated by large viscous effects resulting in higher viscous drag and lower maximum lift. Further the flow is susceptible to early separation with the formation of a laminar separation bubble (LSB). This early separation often causes the flow to become unsteady at relatively lower angles of attack. The flow feature involved in the formation of the transitional separation bubble has been explained in detail by Horton (1968). More recently Xia *et al.* (2021) has provided an indepth insight of the evolution of LSB and flow structure encountered by low Re airfoils at low Reynolds number as shown in Fig. 1.1. The formation of LSB and its effect on aerodynamic performance in terms of increase in the drag and decrease in the lift coefficient have been studied by various researchers both experimentally (Mueller and Batill, 1982; O'Meara and Mueller, 1987; McGhee *et al.*, 1988; Laitone, 1997; Selig and McGranahan, 2004; Ol *et al.*, 2005; Radespiel *et al.*, 2006; Burgmann *et al.*, 2006; Hu and Yang, 2008; Swift, 2009) and numerically (Shyy *et al.*, 1999; Hoarau *et al.*, 2003; Yuan *et al.*, 2005; Windte *et al.*, 2006; Lian and Shyy, 2007; Sahin *et al.*, 2008; Galbraith and Visbal, 2008; Uranga *et al.*, 2009; Karasu *et al.*, 2013; Kondo *et al.*, 2013; Kulkarni *et al.*, 2018). Another important feature at low Reynolds number especially for flow over airfoils is that the separated laminar boundary layer undergoes laminar to turbulent transition inside the separated bubble with a turbulent reattachment.

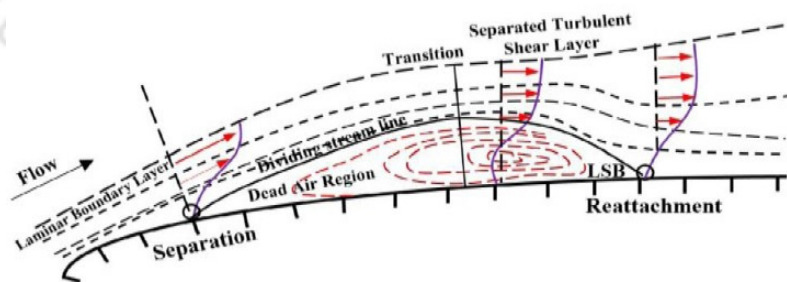


Figure 1.1: Characteristics of laminar separation bubble (Courtesy: Xia *et al.* (2021))

Numerical prediction of the complex phenomenon of the laminar to turbulent transition flow is one of the most challenging task encountered by the CFD community. The challenge faced to

develop a reliable numerical method to predict flow transition is mainly due to the lack of proper understanding of the physics related to the growth of the instabilities causing the laminar flow to become turbulent and having very little knowledge about the different factors like Reynolds number, turbulent intensity, pressure gradient etc. affecting the growth of these instabilities. The flow transition numerically can be tackled primarily by four different approaches *viz.* (i) Linear stability theory (LST) (ii) Direct numerical simulation (DNS) (iii) Large eddy simulation (LES) and (iv) Reynolds averaged Navier-Stokes (RANS) based models. The e^N method which is based on linear stability theory and boundary layer theory was one of the commonly used approach and adapted by designer to model transition and is used in the popular airfoil design code XFOIL (Derla, 2002). This approach though simple cannot be directly adapted to the general CFD code as its empirical nature makes it problem dependent. However few investigators (Radespiel *et al.*, 2006; Lian and Shyy, 2006; Yuan *et al.*, 2005) with the motivation to develop a simple and fast methods have coupled the e^N based approach with the variety of turbulence models available in the CFD codes and have successfully used to predict laminar to turbulent transition over low Re airfoils operating at low Reynolds number. The major drawback of this approach being the difficulty of extending it to three-dimensional problems (Arnal *et al.*, 1996). The DNS and LES are highly accurate as the transition and turbulence are obtained directly with no or minimal modelling hence falls in the other end of the fidelity spectrum. Though these approaches are gaining importance due to the advancement of high performance computing power, high accuracy schemes and acceleration schemes are still limited to simple research problems due to its demand in high computational resources and time. Few researchers (Yuan *et al.*, 2005; Galbraith and Visbal, 2008; Uranga *et al.*, 2009; Castonguay *et al.*, 2010; CartondeWiert and Hillewaert, 2012; Boom and Zingg, 2013) inspite of high computation cost have used LES and DNS to simulate flow past airfoil at low speeds in order to obtain the detail characteristics of these flows. The need of these methods arise as alternative to experiments as it is difficult to obtain accurate measurement data at these low speeds. These methods are compute intense and hence can not be used in the design cycle. The RANS based models which can predict the mean flow quantities reasonably accurate with lesser computational resources and faster turn around time are considered as the most viable tool to be used in the design cycle. These RANS models are basically applicable to fully turbulent flow however in the last few decades a

lot of development has taken place in order to include the effect of transition. The most popular approach to handle transition in the RANS framework is to solve additional transport equations which includes the effect of transition. These approaches to model transition may be broadly classified into phenomenological models and correlation based models. In the phenomenological model class, Walters and Leylek (2002) proposed the $k_T - k_L - \omega$ model which was revised by Walters and Cokljat (2008) and successfully applied to engineering problems. This eddy viscosity based model can address laminar, transition and fully turbulent flow within the RANS framework by solving three transport equations namely the turbulent kinetic energy (k), the laminar kinetic energy (k_L) and specific dissipation (ω). This model is essentially a single point technique as it combines the correlation methods used for transition prediction and eliminates the requirement of the intermittency parameter. Furst *et al.* (2013) have successfully used this single point transition model for predicting different transition process namely natural transition, bypass transition and transition due to incoming wake. Menter *et al.* (2006) developed a correlation based transition model using local variables which has been validated for many applications and is widely used in many commercial CFD codes. This eddy-viscosity based $\gamma - Re_\theta$ SST transition model which also referred as Local Correlation-based Transition model solves for additional two transport equations, one for intermittency (γ) and another for momentum thickness based Reynolds number (Re_θ) for finding the transition onset along with original SST turbulence model transport equations. The three closure correlation functions used in this model were not disclosed initially but were published later by Langtry and Menter (2009) and Malan *et al.* (2009) simultaneously. These transition models have demonstrated their ability to predict flow transition for wide range of applications with acceptable accuracy using significantly lesser computational resources and time.

In addition to the aerodynamic challenges, the MAV wings are posed with the geometrical complexity. The key geometric feature of this class of vehicle is the thin cambered wing section with low aspect ratio (LAR). The performance of these low aspect ratio wings in the low Re flow regime are greatly influenced by the wing section thickness and camber (Mueller, 2001; Kellogg and Bowman, 2004; Kulkarni *et al.*, 2023). To address the geometrical complexity, research have been carried out mainly via experiments to design airfoil series namely Eppler, Selig, Selig & Donovan *etc.* in order to handle the different flow features (*viz.* highly viscous flows, laminar flow separation, flow

reattachment and laminar to turbulent flow transition) confronted in low Re flows regime. These airfoils in general are termed as low Re airfoils in literature. The experimental studies (Winslow *et al.*, 2018) have reported that the aerodynamic performance of the low Re airfoil series are mainly influenced by the airfoil camber and thickness. In this direction, Selig and Donovan focused their research in creating low Re SD airfoil series that could provide higher lift and lower drag particularly at low speed and high angles of attack. The low Re SD7003 airfoil having 8.5% thickness with 1.4% camber was chosen for the present study as it is commonly used for low Reynolds number aerodynamic applications like MAVs. The small dimension of these vehicles demand wing with low aspect ratio ($AR \leq 2$) and the wing section (airfoil) having small thickness and significant camber in order to have a better aerodynamic performance at low Reynolds number. These thin wings with low aspect ratio may pose difficulties in generating a good quality grid so as to obtain accurate flow solution. Several researchers have carried out experiments (Mueller, 1999; Pelletier and Mueller, 2000; Torres *et al.*, 2004; Reid and Kozak, 2006; Zuo and Wang, 2010) and numerical study (Null and Shkarayev, 2005; Chen and Qin, 2013; Kulkarni *et al.*, 2021) in order to understand the effects of wingspan and wing section camber on aerodynamic characteristics at low Reynolds number. It is shown from experiments (Mueller, 1999; Torres *et al.*, 2004) that wing planforms which are rectangular, elliptical, circular or its variants are ideal for MAV application since they offer more lifting area. Torres *et al.* (2004) further have experimentally shown that the wing aspect ratio is a more important parameter than planform shape that influences the aerodynamic characteristics at low Reynolds number. In view of this, the present work aims to numerically predict the influence of the aspect ratio on the aerodynamic and transition characteristics of the wing having a rectangular planform.

In the present work, RANS based $\gamma - Re_\theta$ SST transition model available in the in-house incompressible flow solution code 3D-PURLES (Kulkarni *et al.*, 2019) is used to simulate flow past low Re Selig-Donovan (SD) series airfoils in the low Re regime for which measurement data are available for both transition and aerodynamic characteristics. 3D-PULRES is also used simulate transitional flow past SD7003 rectangular wing with different low aspect ratios for Re ranging from 4×10^4 to 2×10^5 in order study the effect of wing aspect ratio and Reynolds number. The computed lift coefficient, drag coefficient, transition onset location and length of the laminar separation bubble

are compared with the available measurement and others LES/DNS computational data.

1.2 Objectives of the Present Research Work

Flow simulation of low Reynolds number flows over MAV wings poses a major challenge to CFD community to predict the separated flow transition experienced by these flows. The motivation of the present work is to understand and numerically predict the flow characteristics of low Reynolds number airfoils and low aspect ratio wings used in the MAV configurations. The numerical prediction have been carried out using the in-house flow code 3D-PURLES (3D-Pressure based Unsteady Reynolds averaged Navier-Stokes LES). In the present work, transitional flow simulations past low Re airfoils and wings are carried out at low Reynolds number in order to investigate the following

- The aerodynamic and transition characteristics for the low Re SD7003 airfoil predicted using the two transition models and also assess the relative performance of the transition models.
- The influence of Reynolds number on aerodynamic and transition features for the SD7003 airfoil.
- The effect of airfoil camber and thickness on the flow characteristics at different low Reynolds number.
- The flow characteristics of low aspect ratio wings and effect of wing aspect ratio & Reynolds number.

1.3 Organisation of the Thesis

The present research work mainly focuses on the understanding the complex flow past low Re airfoils and low aspect ratio wings operating at low Reynolds number through the numerical simulation of the Navier-Stokes equations. In order to predict the transition and aerodynamic characteristics of the airfoils and wings, the transition models available in the in-house flow solution code 3D-PURLES were used.

Chapter 1 provides a brief introduction of the flow characteristics encountered at low Reynolds number and different approaches to numerically simulate these flows. The scope of the present research work is also provided.

The detailed literature review carried out for low Re flows past airfoils and wings covering both the numerical simulation work and experimental work has been covered in Chapter 2.

Chapter 3 covers the numerical grid generation procedure used in the present work.

The detailed mathematical modelling of the governing Navier-Stokes equations and transition modelling used for this research work is discussed in Chapter 4.

The results obtained for the low Re airfoils and low aspect ratio wings are presented in the subsequent chapters.

Chapter 5 covers validation and the detailed study carried out for the low Re SD7003 at $Re = 6 \times 10^4$ in order to assess the dependency of the different grid and flow control parameters on the flow solution accuracy. The flow past SD7003 airfoil where the the performance of the transition models and the influence of Reynolds number on the performance of this airfoil is discussed in Chapter 6.

In Chapter 7, the effect of different airfoil geometrical parameters such as camber and thickness for SD airfoil series on the aerodynamic and transition characteristics are presented. Chapter 8 presents the study carried out on the flow past SD7003 rectangular wing with three different semi-aspect ratios at $Re = 6 \times 10^4$ using the $\gamma - Re_\theta$ SST transition model and influence of Reynolds number in the low Re regime on the performance of the SD7003 rectangular wing with 0.25 semi aspect ratio.

The last chapter provides concluding remarks on the present work and future plans.

Chapter 2

Review of Literature

2.1 Background

This chapter mainly gives brief description of literature survey carried out on effect of low Reynolds number flows on the aerodynamic and transitional characteristics of wing sections/wings. Numerous researchers have performed their research both experimentally and numerically to study the various parameters influencing the low Reynolds number flows especially applicable to MAV/UAV kind of applications. The findings reported in open literature by various researchers are discussed in the following subsections.

2.2 Experimental

A comprehensive study on the physical aspects on low Re flows have been carried out by Carmichael (1981) experimentally. Through their experiments they have shown that the characteristic behaviour of airfoils at relatively low Re ($10^4 < Re < 10^5$) is the existence of a laminar separation bubble (LSB).

Mueller and Batill (1982) conducted both visualization and force measurements in a low-turbulence subsonic wind tunnel at the University of Notre Dame to understand the cause for the deterioration of the aerodynamic performance at low Re . Their experiments focused on measuring the lift and

drag forces using an external strain gauge balance on the 18% thick symmetric $NACA66_3 - 018$ airfoil at Re ranging between 4×10^4 and 4×10^5 . Through smoke visualization experiments they first observed the formation of leading edge separation bubble for $Re = 4 \times 10^4$ at 8° angle of attack which induces transition and results in sudden increase in lift. They also observed the unusual negative lift at lower angles of attack between 0° and 4° due to the formation of laminar separation on both suction and pressure side.

O'Meara and Mueller (1987) experimentally analyzed the length and height of the LSB with respect to angle of attack and Reynolds number 5×10^4 to 2×10^5 . Their experiments showed that increasing the Reynolds number resulted in the reduction in the bubble length and increase in the bubble height. On the other hand, increasing the angle of attack from 10° to 12° increased both height and length of the LSB. In their conclusion they suggested that one can improve the aerodynamic performance by tripping the boundary layer.

McGhee *et al.* (1988) conducted wind-tunnel experiments in the Langley Low-Turbulence Pressure Tunnel (LTPT) in order to determine the performance characteristics of the Eppler 387 airfoil at Re ranging from 6×10^4 to 4.6×10^5 . Oil flow visualization was used to determine laminar separation and turbulent reattachment locations and they also measured the pressure distribution. Cole and Mueller (1990) performed similar on the Eppler 387 airfoil in low-speed wind tunnel at University of Notre Dame. The pressure distributions obtained by them is similar to that of McGhee *et al.* (1988).

Schmitz (1989) conducted experiments for Re ranging between 2×10^4 and 2×10^5 . he measured the forces generated by three airfoil shapes *viz.* a thin flat plate, a thin cambered plate, and a thick cambered airfoil. His studies showed that the thick cambered airfoil has a critical Re where the performance changes drastically. He also observed that when Re is above the critical Re , the flow undergoes transition to turbulence somewhere on the airfoil and results in higher lift-to-drag ratio.

Laitone (1997) through his experiments in low turbulence wind tunnel and suggested that a good airfoil for relatively low Reynolds number flow ($Re < 7 \times 10^4$) is a thin plate with 5% circular arc camber. He showed that this airfoil has a better L/D ratio at lower Reynolds numbers when compared to NACA0012 and a reversed NACA0012 airfoils. Further in his experiments he observed

the following (i) Thin, cambered airfoil geometry produced a higher total lift for all angles of attack and (ii) sharpening the leading edge resulted in largest lift curve slope. Lowson (1999) confirms the findings of Laitone (1997) through his experiments in open circuit wind tunnel at University of Bristol.

Grundy *et al.* (2000) conducted experiments and measured the forces of the Eppler 61 airfoil (5% thick, 5% camber) for open circuit wind tunnel at University of Bristol with turbulence levels of 0.1% - 0.3 %. They showed that airfoil performance at this Re range depends on acoustic excitation resonating at a different frequency, He also stated that different wind tunnels have measured different aerodynamic characteristics in this range of Re attributed the difference to the resonant frequencies of the wind tunnels.

Selig and McGranahan (2004) conducted experiments in subsonic wind tunnel in the University of Illinois at Urbana Champaign in order to determine the aerodynamic characteristics for six different airfoils at Reynolds numbers of 10^5 , 2×10^5 , 3.5×10^5 , 5×10^5 which included Eppler 387 airfoil. They reported the oil flow visualization and the aerodynamic performance data (lift, moment and drag polar) matches reasonably well with low turbulence pressure tunnel measurements of NASA Langley.

Ol *et al.* (2005) conducted series of experiments and compared LSB formation over an SD7003 airfoil in three different facilities (water tow tank, wind tunnel, and water tunnel). Their results from the different tunnels showed qualitative similarity in the separation bubble shape, velocity field and Reynolds stress distribution but differed in the separation bubble location.

Radespiel *et al.* (2006) conducted experiments as well as unsteady RANS (Reynolds Averaged Navier-Stokes) simulations for stationary and plunging SD7003 airfoil at low Reynolds number (6×10^4). The experimental data consist of high-resolution, Particle Image Velocimetry (PIV) measurements on the upper side of the SD7003 airfoil carried out in the low turbulence wind tunnel (LNB) and the water tunnel (WUB) of the Technical University at Braunschweig. In the numerical simulation the transition was predicted using linear stability analysis coupled to the Menter's baseline $k - \omega$ model. For a stationary airfoil, they compared the computed laminar separation bubble and turbulent shear stress distribution which was used to determine the transition onset with their

experiments and obtained reasonably good agreement.

Burgmann *et al.* (2006) used scanning PIV measurement technique to investigate the spanwise structure and dynamics of the roll-up of vortices within the separation bubble over an SD7003 airfoil at Reynolds numbers between 2×10^4 to 6×10^4 . They discussed in detail about the formation and abrupt bursting of non-regular vortex-rolls at the downstream of the LSB.

Bastedo and Mueller (1986) reported a detailed discussion on the effect of tip vortices on spanwise pressure distributions and formation of the LSB. Their 2D and 3D measurements showed that the aerodynamic performance enhanced with Reynolds number whereas the reduction in aspect ratio deteriorated the performance due to tip vortex. They also reported an existence of LSB which covered the major portion of the wing span.

Pelletier and Mueller (2000) conducted a series of experiments to study the performance for both 2D airfoil as well as 3D rectangular planform wings. They used a wind tunnel with a turbulence level of 0.05% and measured forces for a thin flat-plate wing (1.3% thick, 0% camber) and a thin cambered-plate wing (1.3% thick, 4% camber) at Re ranging between 6×10^4 and 2×10^5 . They used end plates to measure approximate 2D conditions, and removed one of the end plates to measure approximate 3D conditions. Their results indicated the cambered plate had better aerodynamic characteristics. They also showed that (i) decreasing the Reynolds number deteriorated the wing performance indicated by a reduced Lift-to-Drag (L/D) ratio. (ii) turbulent intensity of the wind tunnel and the trailing-edge geometry have insignificant effect on the force coefficients (C_L , C_D , and C_M).

Anyojiu *et al.* (2003) investigated experimentally the aerodynamic characteristics of wings with $AR = 4$ and having three different planforms viz. rectangular, elliptical and delta at Reynolds number ranging from 5×10^3 to 6×10^5 . These experiments were carried out in the low density, planetary atmospheric wind tunnel at JAXA/ISAS at University of Tokyo. They evaluated the effect of Reynolds number on the aerodynamic performance for the three different wing planforms. They observed that for the rectangular and the elliptical wings the lift curve was linear upto certain angles of attack (approx 9°) and was almost independent of Reynolds number. They further noticed that for all Re , the stall angle of the delta wing had a delayed stall angle which is about twice as obtained

by the the rectangular and elliptical wings. Their experiments also indicated that the $(L/D)_{max}$ has a tendency to decrease with the Reynolds number for all the three planforms with the elliptical wing having the lowest L/D .

Torres and Mueller (2004) conducted wind-tunnel tests to experimentally determine the aerodynamic characteristics of low aspect ratio wings at Reynolds numbers below 2×10^5 . They investigated the effect of wing planforms (Rectangular, Zimmerman, Inverse Zimmerman and Elliptical) and aspect ratio (0.5 – 2) on the force coefficients for different angles of attack ranging between 0° - 40° . They observed that the aerodynamic characteristics of low-aspect-ratio wings at low Re are significantly different from that observed at high Re . Their measurements indicated nonlinearity in the lift curve, especially for aspect ratios below 1.25. They reported that wing with inverse Zimmerman planform was found to be more efficient than other planforms for aspect ratios below 1.0 at high angles of attack. Further their experiments indicated that the elliptical planform had better aerodynamic performance for wings with higher AR especially at low to moderate angles of attack. They concluded that the AR played a prominent role in determining the aerodynamic characteristics of the wing rather than the planform shapes.

Sathaye *et al.* (2004) investigated a NACA0012 wing with an aspect ratio of unity in the Reynolds number range of 3×10^4 to 9×10^4 . They observed a dramatic increase in induced drag coefficient for Reynolds numbers below 5×10^4 . They also conducted experiment for a 3% thin flat plate with a sharpened leading edge which indicated that the maximum lift per unit span is at the mid-span location.

Ananda *et al.* (2012) conducted wind tunnel results of wings of low-to-moderate aspect ratio ($2 \leq AR \leq 5$) at low Reynolds numbers 6×10^4 to 1.6×10^5 . Tests were conducted in the low-turbulence wind tunnel in the Subsonic Aerodynamics Research Laboratory at the University of Illinois at Urbana-Champaign (UIUC) at a turbulence intensity of less than 0.1%. Low Reynolds number tests were performed on a wing having an aspect ratio of 4 and using Wortmann $FX63-137$ airfoil. They showed a existence of a critical Reynolds number of 9×10^4 for which a jump in the aerodynamic characteristics was observed. They also conducted experiments for both rectangular and tapered flat plate wings having low aspect ratio at Reynolds number between 6×10^4 to 1.6×10^5

and reported that for flat plates there is no critical Reynolds number for this range of Reynolds number. Ananda *et al.* (2015) also conducted experiments for Re ranging between 5×10^4 to 1.5×10^5 to understand the low Reynolds number aerodynamics of ten flat plate wings having different aspect ratios ($2 \leq AR \leq 5$) and taper ratio ($0 \leq TR \leq 1$) in the 5×10^4 to 1.5×10^5 Reynolds number range. They used oil flow visualization technique to characterize the flow for both straight and tapered wings. The measured aerodynamic coefficients (lift, drag and moment) using three component force balance showed that all wings exhibited sensitivity to changes in aspect ratio and Reynolds number.

Aranda *et al.* (2016) carried out a series of experimental studies to find the precise dynamic response of low-aspect-ratio NACA0012 airfoil (aspect ratio, $AR = 2$) by means of a digital force sensor at low to moderate Reynolds numbers. Experimental tests were performed in the low speed wind tunnel Vehicle Aero-Hydrodynamics Laboratory (VAHL) at Málaga University. They performed for freestream velocity (U_∞) ranging from $4m/s$ to $30m/s$ and turbulence intensity (Tu) ranging from 1.1 % to 2.3 %. Their experimental results showed that wingtip vortex and the low aspect ratio were the main cause for the reduction in lift force when compared to NACA0012 airfoil. The reduction in the lift force was approximately 40 % for all the Reynolds numbers tested. In addition, it was observed that the 3D wing produced an induced drag which was six times greater than the 2D airfoil. They also reported that the stall angle increased with the Reynolds number.

Zilli *et al.* (2017) experimentally investigated the effects freestream turbulence intensity on the formation of separation bubble and transition mechanisms of the flow over an SD7003 airfoil. Surface pressure and skin friction measurements were used to determine how sensitive the separation bubble and transition process. All experiments were performed in the low speed wind tunnel at the University of Toronto Institute for Aerospace Studies. The test section in the wind tunnel is 5 m long, 1.2 m wide and 0.8 m high. The tunnel is capable of operating at a maximum freestream velocity of approximately 40 m/s. The freestream turbulence intensity at this velocity is 0.08%. Three turbulence-generating grids were used to produce the turbulence intensities required for the experiments.

Istavan *et al.* (2018) conducted a series of experiments to investigate the effects of freestream turbulence intensity on laminar separation bubbles forming over the suction side of an airfoil for angles of attack between 0° and 20° , chord Reynolds numbers between 100,000 and 200,000, and freestream turbulence intensities between 0.09% and 2.03%. Experiments were performed in a wind-tunnel facility. All experiments were performed in the closed-loop wind tunnel located in the Fluid Mechanics Research Laboratory at the University of Waterloo. The test section of the tunnel is 0.61 m wide by 0.61 m high and 2.4 m long. The flow is conditioned by aluminum honeycomb and a set of five screens upstream of a 9:1 contraction, resulting in a freestream turbulence intensity of 0.1% over the investigated Reynolds number range. The freestream turbulence intensity was elevated above the baseline level by placing turbulence generating grids just upstream of the test section.

Istavan and Yarusevych (2018) conducted the wind tunnel tests to examine the effects of freestream turbulence intensity on transition in a laminar separation bubble formed over the suction side of a NACA0018 aerofoil. All experiments were performed at an angle of attack of 4° , for chord Reynolds numbers of 80,000 and 125,000, and for freestream turbulence intensities between 0.06% and 1.99%. Both low and high-speed two-component PIV measurements were performed to characterize the transitional flows.

Aarunvinthan and Pillai (2019) carried out Wind tunnel tests to examine the influence of turbulence intensity over the aerodynamic characteristics of an airfoil at different Reynolds number and at various turbulence intensities. Based on the detailed experimental investigation, attempts were made to elucidate the aerodynamic hysteresis phenomenon of airfoils. Comparison of the aerodynamic forces along with the pressure distributions were also discussed in detail.

2.3 Computational

Various computational approaches have been explored for the prediction of low Reynolds number aerodynamics which includes Unsteady Reynolds Averaged Navier-Stokes (URANS) simulations, Large Eddy Simulations (LES) and Direct Numerical Simulations (DNS).

Lin and Pauley (1996) investigated the unsteady laminar boundary layer separation with externally imposed pressure gradient for Eppler 387 airfoil at low Reynolds numbers using two-dimensional DNS. In this study they found that for a mild pressure gradient a closed steady separation bubble. Whereas When the pressure gradient was strong they observed periodic vortex shedding due to Kelvin-Helmholtz instabilities in the shear layer.

Hoarau *et al.* (2003) investigated numerically the three-dimensional transitional flow around a NACA0012 airfoil at Reynolds numbers from 8×10^2 to 10^4 . Their DNS results showed the existence of three-dimensional undulated large-scale vortices row with a regular spanwise wavelength which is very similar to that of bluff body wakes.

Deng *et al.* (2007) used DNS to predict flow separation and transition around a NACA0012 airfoil at an angle of attack 4° and a Reynolds number of 10^5 . In this study they investigated in detail flow separation, formation of detached shear layer, Kelvin-Helmholtz instability, vortex shedding, interaction of non-linear waves, breakdown and reattachment.

Sahin *et al.* (2008) carried out unsteady calculations of low Reynolds number flows over Eppler-387 airfoil for both two and three dimensions using DNS. They calculated instantaneous and time averaged pressure and aerodynamic coefficients and compared with the experimental data. Their simulations demonstrated a significant difference between two and three dimensional pressure coefficient distributions over the airfoil surface. This difference they have mainly attributed to the presence of three dimensional instabilities making the flow eventually three dimensional for the 2D geometries. Through these numerical simulations they have provided a detailed information about the laminar separation bubble which is not directly possible with the wind tunnel experiment due to relatively large level of freestream turbulence fluctuations at the inflow. Rodriguez *et al.* (2013) also carried out the DNS for NACA0012 airfoil at two different angles of attack 9.25° and 12° at relatively low Reynolds number ($Re = 5 \times 10^4$) to address the mechanisms of transition at the separated shear layer.

Yuan *et al.* (2005) numerically investigated SD7003 airfoil at 6×10^4 for 4° and 8° angles of attack to predict the location of separation and reattachment of the laminar separation bubble and the transition locations using LES and URANS simulations of $k - \omega$ turbulence model coupled to linear

stability approach. They have concluded that the LES computations have given deeper insights to the flow physics, whereas the coupling of RANS models with transition prediction capability serves as good practical engineering tool for aerodynamic design and analysis.

Yuan *et al.* (2006) carried out preliminary parametric study using LES on the SD7003 airfoil at 6×10^4 in which they investigated the effect of grid resolution and sub-grid scale (SGS) models. They could not capture the 3D effects accurately because of the coarse grid resolution in the spanwise direction. These preliminary LES calculations have revealed some important flow characteristics such as leading edge laminar separation and vortex shedding from the primary laminar separation bubble on this low Reynolds number airfoil.

Galbraith and Visbal (2008) and Uranga *et al.* (2009) used Implicit LES to predict unsteady flow simulation over SD7003 airfoil at low Reynolds number. The results clearly demonstrated the critical role of transition on the unsteady flows and significant three dimensional effects on SD7003 infinite wing even at low to moderate Reynolds numbers.

Gordnier and Visbal (2009) applied Implicit LES for transitional flow over SADDAM airfoil for $Re = 4 \times 10^4$ at 4° . Zhou and Wang (2010) analysed for this case at 6×10^4 with a spectral difference method on unstructured hexahedral mesh.

Castonguay *et al.* (2010) simulated transitional flow over SD7003 airfoils under low Reynolds number at 10^4 and 6×10^4 for an angle of attack of 4° using an Implicit LES coupled to high order spectral difference method for spatial discretisation and computed separation, transition and reattachment locations.

Catalano and Tognaccini (2011) numerically analysed SD7003 airfoil for angles of attack varying from 0° to 12° using two approaches *viz.* LES and URANS. The URANS simulations were carried out using $k - \omega$ low Re SST model by fixing the transition location at a prescribed location based on the experimental trip location. Results of the RANS simulations are in good agreement with the LES data and available measurement data in terms of length of LSB, lift and drag coefficient.

CartondeWiert and Hillewaert (2012) numerically investigated the flow past SD7003 airfoil for $Re = 6 \times 10^4$ at $\alpha = 4^\circ$ using DNS as well as Implicit LES coupled to the high order discontinuous

Galarkin method. They found that the Implicit LES results were in close agreement with the DNS results for all the flow quantities compared in terms of velocity, vorticity and skin-friction.

Kondo *et al.* (2013) performed Implicit LES computations over a owl-like wing to analyse the flow field and aerodynamic performance at a chord Reynolds number of 2.3×10^4 and at an angle of attack ranging from 0° to 9° . Their instantaneous results at low angles of attack showed separation, transition, and reattachment on the pressure side and the time averaged results (time and span) showed the formation of LSB on pressure side which had little influence on the variation on lift coefficient with angle of attack. On the other hand, at high angles of attack their instantaneous results showed a reversed phenomenon and the time averaged LSB on the suction side had a significant effect on the variation of lift coefficient. Their studies showed that possibility for improving aerodynamic performance by choosing an appropriate airfoil surface geometry.

Shyy *et al.* (1999) investigated the issues related to the design of low Reynolds number airfoils, such as the thickness, camber, and surface profiles. They performed the XFOIL analysis for NACA0012, CLARK-Y, S1223 and modified S1223 (UF) airfoils having different camber and thickness for Reynolds number ranging from 7.5×10^4 to 2.0×10^6 at various angles of attack and studied the relative performance by comparing the corresponding aerodynamic coefficients. They concluded that for this range of Re , the cambered S1223 airfoil showed a better performance compared to the symmetric NACA0012 and CLARK-Y airfoils. Whereas the cambered UF airfoil with higher thickness showed an improved aerodynamic characteristics over the S1223 airfoil when the Reynolds number is reduced.

Singh (2000), performed computations using XFOIL over several airfoils at $Re = 8 \times 10^4$. His results showed that the thin Selig cambered airfoil had a better aerodynamic coefficients when compared to thicker and less cambered airfoils (Wortmann and NACA) airfoils.

Kellogg and Bowman (2004) carried out a parametric computational study on the thickness of airfoils for the MAVs at three different Reynolds numbers (6×10^4 , 10^5 and 1.5×10^5) and found that decreasing the Reynolds number also decreased the optimal thickness with respect to L/D . Through his simulations he concluded that an airfoil to be used for low Reynolds number flows should be relatively thin.

Elimelch *et al.* (2005), conducted both experiments and computations to study the turbulence characteristics over NACA0009 and Eppler 61 airfoils. It was found that a very fine mesh could capture the turbulence quantities quite well when compared to the flow visualization. They further noticed that for these airfoils, the transition from laminar to turbulent occurs in the Reynolds number between 2×10^4 to 6×10^4 at low angles of attack.

Tang (2006) used RANS simulations for SD7003 airfoil to predict low Reynolds number airfoil aerodynamics. In his simulations initially the transition onset location point was predicted numerically using the laminar computation at the point where the tangential velocity adjacent to the solid surface reverses its direction for the second time after the laminar separation. Later RANS computations are performed with zero production term for the selected turbulence model upto this transition point and fully turbulent computation beyond this transition point. He studied the robustness of this methodology to predict the LSB and transition onset location for SD7003 airfoil using Spalart-Allamaras and few two equation baseline turbulence models and reported that the results obtained by Spalart-Allamaras are closer to the measurement data as compared to the two equation models.

Windte *et al.* (2006) performed RANS simulations on the SD7003 airfoil using Menter's base line two layer $k - \omega$ model at different angles of attack and computed the length of LSB as well as aerodynamic coefficients. Reasonably good agreement is obtained with measurement data for transition onset location and with XFOIL data for force coefficients.

Lian and Shyy (2007) carried out numerical investigations at $Re = 6 \times 10^4$ for rigid and flexible SD7003 airfoil using $k - \omega$ model coupled to e^N transition model and obtained a reasonably good predictions for the transition location, aerodynamic coefficients and flow structures.

Rival *et al.* (2010) used $\gamma - Re_\theta$ model for SD7003 airfoil for stall onset angle of attack 8° at $Re = 6 \times 10^4$ and found that predicted lift and drag coefficients by the $\gamma - Re_\theta$ SST model are marginally better than standard $k - \omega$ SST model as compared to measurement data. The separation point and transition onset location are closer to the measurement data while the reattachment point is overpredicted.

Counsil and Boulama (2011) numerically investigated flow over NACA0012 airfoil at three low Reynolds numbers *viz.* 5×10^4 , 10^5 and 2.5×10^5 at three angles of attack *viz.* 0° , 4° and 8° using

Menter's $\gamma - Re_\theta$ transition model. These results were compared with Menter's fully turbulent shear stress transport (SST) model. They concluded that $\gamma - Re_\theta$ transition model was able to capture the LSB, transition phenomena and aerodynamic characteristics with better accuracy.

Karasu *et al.* (2013) carried out Fluent computations to predict LSB over a NACA2415 airfoil at low Reynolds numbers and at various angles of attack using $k - \omega$ SST and $k_T - k_L - \omega$ models. Computations were carried out at pre-stall ($4^\circ, 8^\circ$), at stall (12°) and at post-stall (15°) angles of attack for the Reynolds numbers $Re = 3 \times 10^5$ & $Re = 5 \times 10^5$. They also observed that as the angle of attack increases the laminar separation bubbles moves towards the leading edge and the length of the separation bubble decreases. They concluded that the transition models are more successful than the $k - \omega$ SST turbulence model for prediction of laminar separation bubble at low Reynolds number flows.

Jian and Ke-Qin (2004) carried out a numerical study of a low aspect ratio elliptic wing with an Eppler 174 airfoil cross section at a Reynolds number of 10^4 . They observed that at an angle of attack larger than 11° , flow around the wing becomes bilateral asymmetric due to the interaction between the secondary separated vortex and the tip vortices. Further their results showed that beyond 33° a large separated stationary vortex above the wing was observed.

Cosyn and Vierendeels (2006) numerically investigated aerodynamic forces acting on rectangular low aspect ratio wings (AR between 0.5 and 2) at $Re = 10^5$ using CFD and a strip method and compared results with the experimental data. The CFD simulations were carried out using the Fluent with Spalart Allamaras turbulence model whereas the effect of low Reynolds number effect and flow separations were studied using a strip method which uses 2D experimental lift and drag data to compute corresponding lift and drag for 3D. Based on the these studies they derived two methods to predict the lift and drag of LAR wings, One for profiled wings with low to moderate Reynolds number effect whereas the other for flat plate wings with large flow separation.

Taira and Colonius (2009) performed numerical simulations of three dimensional separated flows around LAR flat plate wings at low Reynolds numbers using an immersed boundary method and results were validated with the oil tow-tank measurements. Simulations were performed for $Re = 300$ and 500 with different aspect ratios, angles of attack and planforms. They studied both

the initial transient as well as the long time behaviour of the flow. Their results highlighted the unsteady phenomenon observed at these Reynolds numbers. They showed that the aspect ratio and angle of attack were found to have a large influence on the stability of the wake profile and the force experienced by the body. Their results also showed that Wake behind non-rectangular wings were observed to be different from the rectangular planforms in terms of the vortex shedding pattern and the the formation of tip vortices. However, the overall trend for the lift and drag histories were found to be similar to those of the rectangular case. Three dimensional separated flows obtained by them for low aspect ratio plates were found to be significantly different from the analogous two dimensional flows. They also observed that the tip effects in three dimensional flows stabilized the flow and exhibited nonlinear interaction of the shedding vortices.

Hamburg *et al.* (2011) conducted a detailed computational investigation using $k - \omega$ turbulence model to study the effects of aspect ratio (1.0, 2.0, 5.0 6.0) and taper ratio (0.2 to 1.0) on wings with the NACA0009 airfoil section. Based on the above study, they proposed a preliminary model for estimation of induced drag for low Re flows based on aspect ratio and taper ratio.

Noughabi and Tadjfar (2013) studied the influence of crosswind on flow over the inverse Zimmerman wings with AR and Re ranging between 1 and 2 and 6×10^4 and 10^5 respectively. Numerical simulations were carried out using a three dimensional finite volume method solver with Menter's shear stress transport turbulence model. They considered crosswind's angles from 0° to 40° and angle of attack from 0° to 12° and they found that lift and drag coefficient generally decreasing when the angle of the crosswind is increased.

Shetty *et al.* (2013) carried out URANS simulations coupled to standard $k - \epsilon$ turbulence model over two wing planforms viz. modified inverse Zimmerman (Black Kite) and a modified cropped delta wing (Golden Hawk) to analyze their aerodynamic characteristics at root chord based Reynolds number 2.4×10^5 and 1.7×10^5 respectively. They performed computations for Black Kite wing using standard $k - \epsilon$ turbulence model whereas for Golden Hawk wing they used three different turbulence ($k - \epsilon$, SA, $k - \omega$ SST) models and the computed aerodynamic coefficients reasonably matches with NAL measurement data for both the wing planforms. They also captured the wake flow and the tip vortex for both the wings.

Demir *et al.* (2016) investigated flow over NACA4412 airfoil both numerically and experimentally at different Reynolds numbers. The experiments were carried out in a low speed, suction type wind tunnel located at the Department of Energy Systems Engineering, University of Erciyes. Turbulence intensity of the tunnel is 0.35% and the operating range of tunnel speed are $1 - 45\text{m/s}$. The experiments were conducted at various angles of attack and different Reynolds numbers (2.5×10^4 and 5×10^4) and aspect ratios ($AR = 1$ and $AR = 3$). Smoke wire and oil flow visualization methods were used to visualize the surface flow patterns. The numerical simulations were carried out using FLUENT with SST transition model. They compared the numerical results with experimental results for the laminar separation bubble and the effect of tip vortices. They reported that tip vortices that occurred on the airfoil tip affected the flow over the airfoil and additionally, leading edge vortices combined with tip vortices results in 3D vortices. which in turn decreases the bubble.

2.4 Summary

The detailed literature survey has been carried out from 1982 to 2022 (i) To understand the principle and characteristics of low Reynolds number flow regime (ii) Experiment work carried out for low Re airfoils and low aspect ratio wings at different tunnels – Smoke flow, Oil flow, PIV (iii) Numerical simulation work carried out for low Re flow regimes – Empirical, CFD using Finite volume, Finite Element, Finite Difference with turbulence simulated using RANS, LES or DNS (iv) State-of-art of the different eddy viscosity turbulence models and their application to low Re airfoils in particular.

Chapter 3

Numerical Grid Generation Procedure

3.1 Background

Generation of smooth, body-fitted grids with approximate orthogonality at the boundaries is practically the first step towards the accurate numerical solution of fluid flow equations for arbitrary configuration using either finite volume or finite element or any other methods. The location and kind of physical boundaries, interpreted mathematically as the boundary conditions of the governing differential equations of flow determine the spatial distribution of the flow variables in the domain of interest. Therefore the distribution of grid points on domain boundaries, usually specified by the user, should accurately reflect the geometrical boundaries of the domain. The grid should also have sufficient number of points distributed appropriately to detect the major characteristics of the flow. Other desirable features of a good grid generation methodology are the smoothness and the boundary-orthogonality of the grid generated, an easy and direct control of grid-spacing and grid skewness at any desired location and finally an efficient and fast numerical algorithm. A poorly constructed grid may yield inaccurate results and may bring down the convergence rate of the numerical solution of the flow equations also to a prohibitively low level. The grids used for the present study are generated using the in-house grid generation code and details of this code are described in the following sections.

3.2 Differential-algebraic hybrid method for 2D plane

The present in-house grid generation algorithm (Fathima *et al.*, 1994) developed at the CTFD Division, NAL involves the solution of the elliptic type differential equations at a coarse level, followed by simple algebraic interpolation from the coarse to a finer level. The coarse grid is primarily generated by solving a system of inverted Poisson equations for a given point distribution at all four boundaries of a two dimensional computation domain. The control functions in the equations are automatically adjusted in an iterative procedure to achieve boundary-orthogonality and need no ad-hoc adjustment of the problem-dependent parameters. Finally, when the desired concentration of grid points is specified at the fine level on one boundary along each direction, the fine level field grid coordinates is obtained by fitting bicubic spline functions passing through the coarse level grid nodes. This Hybrid approach proposed first by Zhu (1990) makes a compromise between the simple algebraic and the expensive differential approach and guarantees smooth grid of desired fineness and approximate boundary-orthogonality for a very reasonable computation cost.

3.3 Governing equations

In the differential equation based grid generation methods, the coordinates of the grid points (x, y) in the physical space are computed by solving the following inhomogeneous Poisson equations (Pal *et al.*, 1995) in a rectangular computation domain (ξ, η) .

$$\xi_{xx} + \xi_{yy} = P(\xi, \eta) \quad (3.1)$$

$$\eta_{xx} + \eta_{yy} = Q(\xi, \eta) \quad (3.2)$$

where, P and Q are the source terms to control the spacing and orientation of the grid in the field. However the equations in the above form, where (x, y) are independent variables, are somewhat inconvenient to handle since in a grid generation problem a rectangular mesh is chosen on the plane (ξ, η) and it is required to calculate the x and y coordinates of the corresponding point by mapping

it back to the physical space. Therefore the above equations are to be transformed to a system where (ξ, η) are the independent variables. This transformation yields the following elliptic system of non-linear equations.

$$\alpha x_{\xi\xi} - 2\beta x_{\xi\eta} + \gamma x_{\eta\eta} = -J^2(Px_{\xi} + Qx_{\eta}) \quad (3.3)$$

$$\alpha y_{\xi\xi} - 2\beta y_{\xi\eta} + \gamma y_{\eta\eta} = -J^2(Py_{\xi} + Qy_{\eta}) \quad (3.4)$$

where,

$$\alpha = x_{\eta}^2 + y_{\eta}^2 \quad \beta = x_{\xi}x_{\eta} + y_{\xi}y_{\eta} \quad \gamma = x_{\xi}^2 + y_{\xi}^2$$

and J denotes Jacobian of the transformation

$$J = x_{\xi}y_{\eta} - x_{\eta}y_{\xi}$$

Eqns. 3.3 and 3.4 are solved numerically for x and y on a uniform rectangular grid consisting of unit squares along ξ and η directions. The source terms P and Q are computed iteratively by the method suggested by Thomas and Middlecoff (1979) which can effect and automatic control the interior grid points. The governing equations are modified in the following form:

$$\alpha(x_{\xi\xi} + px_{\xi}) - 2\beta x_{\xi\eta} + \gamma(x_{\eta\eta} + qx_{\eta}) = 0 \quad (3.5)$$

$$\alpha(y_{\xi\xi} + py_{\xi}) - 2\beta y_{\xi\eta} + \gamma(y_{\eta\eta} + qy_{\eta}) = 0 \quad (3.6)$$

where the control functions p and q are related to P and Q of the original equations in the following manner:

$$P = p(\xi, \eta)(\xi_x^2 + \xi_y^2) \quad Q = q(\xi, \eta)(\eta_x^2 + \eta_y^2)$$

The control functions p and q are evaluated along the boundaries satisfying the boundary-orthogonality condition and the desired grid spacing, discussed in the next section. Eqns. 3.5 and 3.6 are solved numerically using an iterative procedure over a uniform rectangular grid in the computational domain to generate a body-fitted boundary-orthogonal grid on the physical plane.

The boundary conditions in the flow solver becomes straightforward and the solution accuracy is also enhanced if the grids are nearly orthogonal at least on the boundaries of the computational domain. The non-orthogonal curvilinear grids are therefore made to follow the approximate boundary-orthogonality to eliminate the steep gradients arising out of the cross-derivative terms. To implement the local orthogonality at boundaries, several approaches based on the original procedure due to Thompson *et al.* (1974) are reported in literature.

The non-linear and strongly coupled inverted Poisson equations (Eqns. 3.5 and 3.6) are solved numerically using simple finite difference method where the differential operator of the equations are discretised by appropriate central difference operators and the control functions p and q are evaluated like source terms at each grid node.

3.4 Bicubic spline fitting for coarse to fine interpolation

To obtain the fine grid with desired concentration of grid lines by fitting bicubic splines through the smooth coarse grid lines generated by the Poisson solver. The bicubic spline fitting consists of the repeated use of univariate cubic spline along two directions in the transformed space. The values of ξ and η at the coarse grid points and converted to fine grid is referred in NAL PD by Fathima *et al.* (1994)

3.5 Modeling of boundary geometry

As discussed in the previous section, the Poisson solver as well as the bicubic spline function used for coarse to fine interpolation requires specification of the coordinates of the points on the boundary. The Poisson solver treats the grid generation problem as a Dirichlet kind of boundary value problem for a system of elliptic equation and hence the coordinates must be specified at all four boundaries of a computational domain in two-dimensional space. However for algebraic interpolation from coarse to a finer level of the grid, the boundary points at fine level need to be specified on one computation boundary only along each of the two directions. The value of ξ corresponding to

the $\eta = \eta_{min}$ boundary and the values of η along $\xi = \xi_{min}$ boundary are evaluated from the coarse level information on the same boundary and the specified boundary coordinates (x, y) at fine level. Since the grid network at fine level also is assumed to be represented on the $\xi - \eta$ plane as cartesian mesh only, the values of ξ and η evaluated on ξ_{min} and η_{min} boundaries remain unaltered on the corresponding opposite boundaries as well. This section describes a systematic procedure of prescribing the coordinates of boundary points for a physical problem according to a desired intensity and location of stretching.

For any grid generation problem, the domain on computational plane is always defined by four edges of a square *i.e.*, bounded by the minimum and maximum values of ξ and η . The number of boundaries considered is always four, irrespective of the physical problem. But on the corresponding physical space $(x - y)$, each of these four computational boundaries may consist of more than one segments, separated by salient points which should necessarily be a grid node at both coarse and fine level. The segment may however be linear or non-linear depending on the geometry considered. For a straight segment, the specification of the two end points is good enough for its definition. But for a curved segment however, coordinates of more than two discrete points on the curve need to be specified for its definition. Once a segment is defined numerically by the end points or a set of discrete points on the segment, the next important task is how to lay out points on the same segment according to a specified stretching pattern.

The coordinates of a set of discrete points on a curved segment are not enough to compute the arc length for the purpose of stretching or to compute back the coordinates of the points to be marked on the segment following a stretching rule. A curved segment is therefore redefined by a cubic spline approximation *i.e.*, passing a piecewise cubic polynomial through the given data points as nodes for the spline. Once the analytical expression is derived for the spline, the arc-length at every nodal point, measured from the starting point of the spline and the total arc-length may be accurately calculated. The desired points may then be marked easily on the curved segment following a geometrical progression rule of stretching described in the next section.

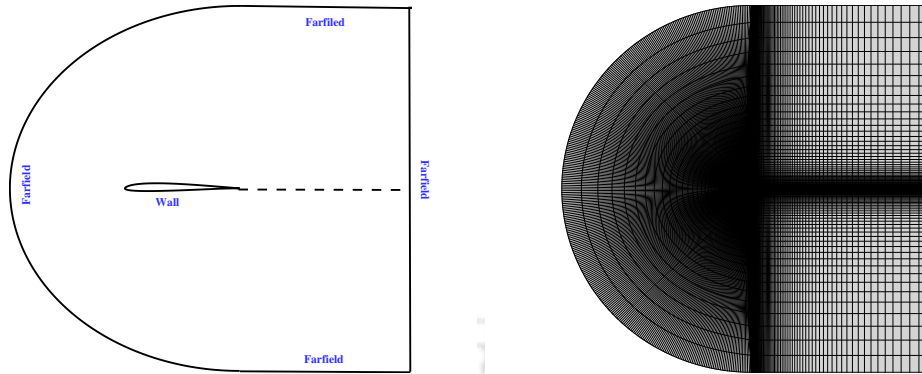
The stretching functions used in the present work are purely one-dimensional *i.e.*, the grid size increases or decreases only along the segment length (L), irrespective of whether the segment is

straight or curved. Points are marked on a given segment in such a way that the grid size *i.e.*, the length between consecutive points, measured along the segment-arc follows a geometric progression rule whose common ratio decides the extent of stretching. The points may be stretched either at the beginning or at the end or near the both ends of the segment, the number of grid points (N) to cover the segment including the ends, the common ratio (R) of the geometric progression and a location index to indicate whether stretching is intended at the start or end or both ends of the segment. If S be the minimum grid size, measured along the segment-arc, the magnitude of S according to the geometric progression rule is :

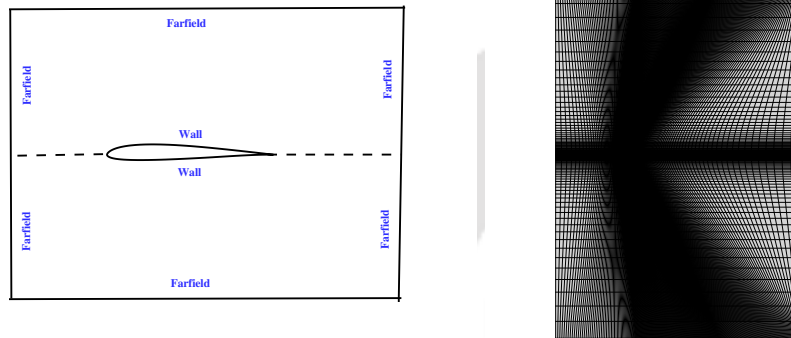
$$S = \frac{L(R - 1)}{(R^{N-1} - 1)} \quad (3.7)$$

In case of stretching at both ends, the number of nodes must be odd and the arc is divided into two equal halves and each portion is stretched near the corresponding edge using similar formulae.

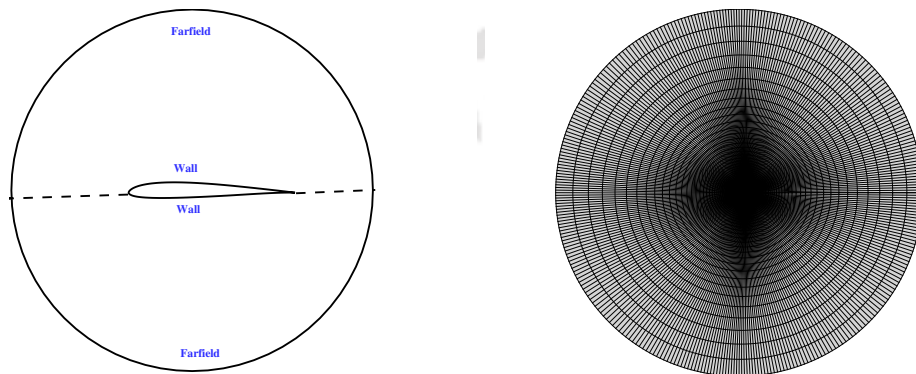
The C-grid, H-grid and O-grid topology grids generated using this code for SD7003 airfoil are given in Fig. 3.1.



(a) C-grid, Grid size: 527×101



(b) H-grid, Grid size: 638×202



(c) O-grid, Grid size: 319×101

Figure 3.1: 2D grid for different topology for SD7003 airfoil

3.6 Single block to multiblock conversion

The dotted line shown in Fig. 3.1 is not any physical boundary like wall, symmetry, farfield, inlet, outlet etc. but it is a part of the internal flow domain and needs a special treatment. In the present simulations using the in-house flow solver 3D-PURLES this flow boundary is handled by splitting the flow domain into two halves along this line and this line is treated as the block interface (cut boundary). The details of handling the cut boundary is given in Section 4.4.8. The single block grid generated for the airfoil for different grid topologies are split into halves appropriately as shown in Fig. 3.2. The C-grid is divided into the airfoil and wake block whereas for the O-grid and H-grid topologies the grid is divided into upper surface and lower surface of the airfoil.

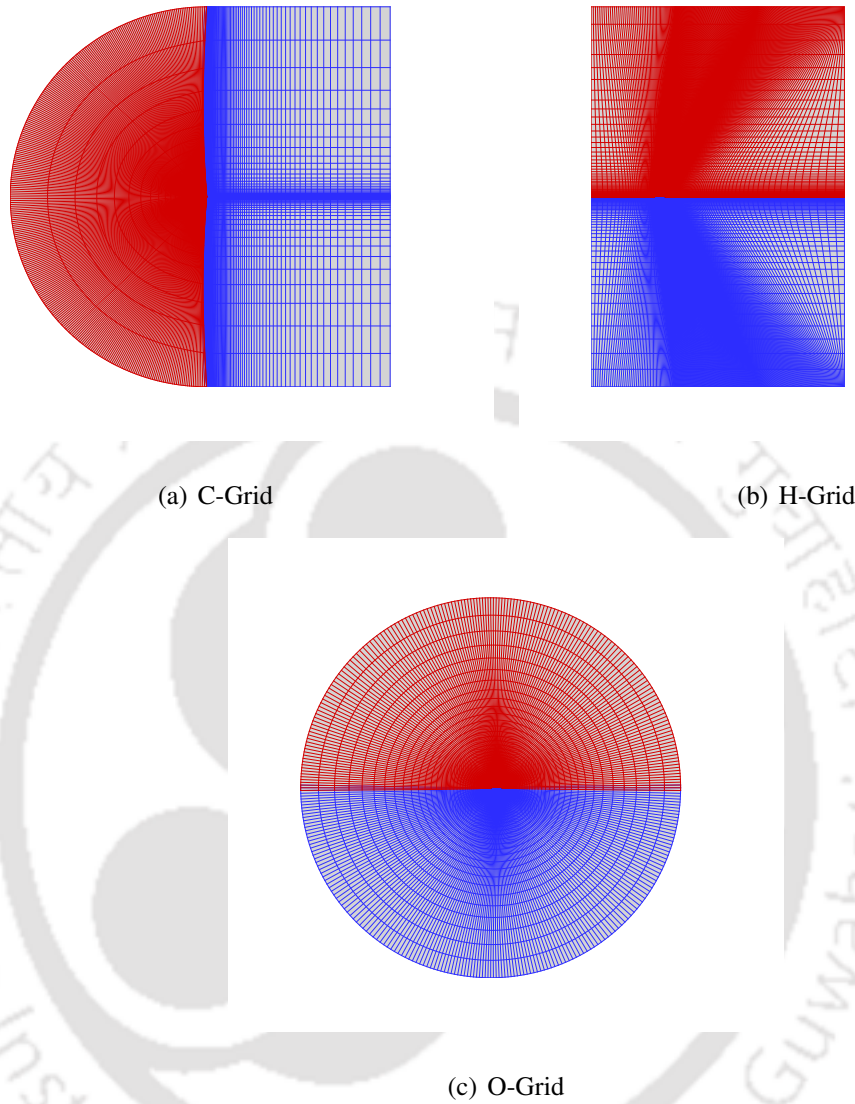


Figure 3.2: 2-block SD7003 airfoil grid for different topologies

3.7 3D Grid Generation

3D grid is generated using a quasi 3D-procedure. The boundary orthogonal grids on each 2D plane are stacked either longitudinally on parallel planes or circumferentially (in the case of axisymmetric bodies) depending on the geometrical similarity. In the present study, the grid for the SD7003 rectangular wing is generated by stacking the 2D airfoil H-grid along the wing span to obtain a H-H grid topology as shown in Fig. 3.3.

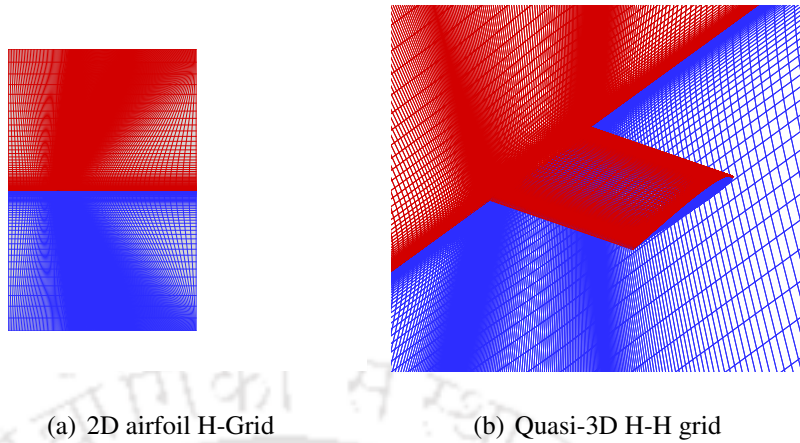


Figure 3.3: Grid generated for SD7003 rectangular wing

3.8 Summary

In this chapter, the 2D and quasi-3D grid generation procedure used for the present work is discussed in detail.

Chapter 4

Mathematical Modelling of Flow Physics

4.1 Background

The analysis of transitional flow past low Re Selig-Donovan (SD) series airfoils and SD7003 rectangular wings in the low Reynolds number flow regime are carried out using in-house flow solution code 3D-PURLES. In this flow solution code, the Navier-Stokes (NS) equations are appropriately modelled to handle the geometrical complexities like arbitrary shaped boundaries as well as the physical complexities. This code uses the pressure-based finite volume algorithm for non-orthogonal coordinates with Cartesian velocities as dependent variables as discussed in following sections. The code uses variety of turbulence models to solve the fully turbulent flows and it also has the Large Eddy Simulation capability. The LES though can handle transitional flow is computational expensive and not suitable as designer tool. As a part of my research work, the RANS based transition models namely $k_T - k_L - \omega$ and $\gamma - Re_\theta$ SST models were implemented in 3D-PURLES to give a quick and reasonably accurate prediction of the transitional flows and the details of the implementation are discussed in section 4.3.

4.2 Governing Equations

The unsteady Reynolds averaged Navier-Stokes (URANS) equations for incompressible turbulent flow in non-orthogonal curvilinear coordinates with Cartesian velocities as dependent variables used in 3D-PURLES flow code is as follows:

Momentum transport equation for the Cartesian velocity component $\langle U_i \rangle$:

$$\frac{\partial (\rho \langle U_k \rangle)}{\partial t} + \frac{1}{J} \frac{\partial}{\partial x_j} \left[\rho \langle U_i \rangle \langle U_k \rangle \beta_i^j - \frac{\mu}{J} \left(\frac{\partial \langle U_k \rangle}{\partial x_i} B_i^j + \frac{\partial \langle U_i \rangle}{\partial x_m} \beta_i^j \beta_k^m \right) \right] + \frac{1}{J} \frac{\partial}{\partial x_j} [\langle P \rangle \beta_k^j + \rho \langle u_i u_k \rangle \beta_i^j] = S_{U_k} \quad (4.1)$$

where, $\langle P \rangle$ is the phase averaged pressure and $\langle U_i \rangle$ are the phase averaged velocity components. J is the Jacobian of the transformation matrix. β_i^j and B_i^j are the relevant metric coefficients related to the geometrical transformation. μ is the fluid viscosity and ρ is the fluid density. u_i are the fluctuating velocity components and S_{U_k} is any other body force other than pressure gradient. These momentum equations are further supplemented by the mass conservation (continuity) equation.

Mass conservation equation

$$\frac{\partial}{\partial x_j} (\rho \langle U_i \rangle \beta_i^j) = 0 \quad (4.2)$$

The unknown phase averaged turbulent (or) Reynolds stress term $-\rho \langle u_i u_k \rangle$ appearing in Eq. 4.1 is expressed based on the eddy viscosity hypothesis as follows :

$$-\rho \langle u_i u_k \rangle = \mu_t \left(\frac{\partial \langle U_i \rangle}{\partial x_k} + \frac{\partial \langle U_k \rangle}{\partial x_i} \right) - \frac{1}{3} \rho \delta_{ik} \langle u_m u_m \rangle \quad (4.3)$$

where, δ_{ik} is the Kronecker delta. and m is the summing index over $m = 1, 2, 3$. The term $\frac{1}{3} \rho \delta_{ik} \langle u_m u_m \rangle$ ensures that the sum of the normal stresses is $2k$ as per definition of the turbulence kinetic energy. ($k = \frac{1}{2} \langle u_i u_i \rangle$). The eddy viscosity μ_t is assumed to be an isotropic scalar whose value

depends on the local state of turbulence which determined either using the turbulence or transition models. The 3D-PURLES flow code has five different eddy viscosity based turbulence models *viz.*, variants of $k - \epsilon$ model (Launder and Spalding, 1974; Chien, 1982), $k - \omega$ model (Wilcox, 1994), SST model (Menter, 1994), SA model (Spalart and Allamaras, 1992) and $v^2 - f$ model (Durbin, 1995), In this code two transition models *viz.*, laminar kinetic energy based $k_T - k_L - \omega$ transition model (Walters and Cokljat, 2008) and local correlation based $\gamma - Re_\theta$ transition model (Langtry and Menter, 2009) have been recently incorporated and validated (Kulkarni *et al.*, 2019). As the focus of present work is mainly on transitional flows, the simulations have been carried using the transition models which are described in the following sub section.

4.3 Transition Modelling

The details of the formulation of the transition model transport equations which were implemented in the 3D-PURLES as a part of my current research works is given in the following sub sections. The implemented transition models were initailly validated for the standard ERCOFTAC flat plate test cases (Kulkarni *et al.*, 2019) before using for my research problem.

4.3.1 Laminar kinetic energy based $k_T - k_L - \omega$ transition model

The laminar kinetic energy based $k_T - k_L - \omega$ transition model was proposed by Walters and Cokljat (2008). This model solves three transport equations namely the turbulent kinetic energy (k_T), laminar kinetic energy (k_L) and specific dissipation (ω) as given below. The damping function and constants required to close these equations are taken from Furst *et al.* (2013).

k_T - equation

$$\frac{\partial(\rho k_T)}{\partial t} + \frac{1}{J} \frac{\partial}{\partial x_j} \left[\rho \langle U_i \rangle k_T \beta_i^j - \frac{(\mu + \alpha_T / \sigma_k)}{J} \left(\frac{\partial(k_T)}{\partial x_m} \beta_i^j \beta_i^m \right) \right] = P_{k_T} + R_{BP} + R_{NAT} - \rho \omega k_T - D_T \quad (4.4)$$

k_L - equation

$$\frac{\partial(\rho k_L)}{\partial t} + \frac{1}{J} \frac{\partial}{\partial x_j} \left[\rho \langle U_i \rangle k_L \beta_i^j - \frac{(\mu)}{J} \left(\frac{\partial(k_L)}{\partial x_m} \beta_i^j \beta_i^m \right) \right] = P_{kL} - R_{BP} - R_{NAT} - D_L \quad (4.5)$$

ω - equation

$$\begin{aligned} \frac{\partial(\rho \omega)}{\partial t} + \frac{1}{J} \frac{\partial}{\partial x_j} \left[\rho \langle U_i \rangle \omega \beta_i^j - \frac{(\mu + \mu_t / \sigma_\omega)}{J} \left(\frac{\partial \omega}{\partial x_m} \beta_i^j \beta_i^m \right) \right] = C_{\omega 1} \frac{\omega}{k_T} P_{kT} + \\ \left(\frac{C_{\omega R}}{f_W} - 1 \right) \frac{\omega}{k_T} (R_{BP} + R_{NAT}) - C_{\omega 2} \rho \omega^2 + C_{\omega 3} \rho f_\omega \alpha_T f_W^2 \frac{\sqrt{k_T}}{d^3} \end{aligned} \quad (4.6)$$

The total fluctuation kinetic energy, $k_{TOT} = k_T + k_L$.

The production of turbulent kinetic energy is defined as

$$P_{kT} = \mu_{T,s} S^2 \quad (4.7)$$

where $S = \sqrt{2S_{ik}S_{ik}}$ is the absolute value of strain rate.

The small scale eddy-viscosity is defined as

$$\mu_{T,s} = \rho f_\nu f_{INT} C_\mu \sqrt{k_{T,s}} \lambda_{eff} \quad (4.8)$$

where $k_{T,s}$ is the effective small scale turbulence defined as

$$k_{T,s} = f_{ss} f_W k_T \quad (4.9)$$

where k_T is the transported turbulent kinetic energy and f_{ss} and f_W are the damping functions related to the shear-sheltering effect and kinematic wall effect defined as follows

$$f_{ss} = \exp \left[- \left(\frac{C_{ss} \nu \Omega}{k_T} \right)^2 \right] \quad (4.10)$$

$$f_W = \left(\frac{\lambda_{eff}}{\lambda_T} \right)^{2/3} \quad (4.11)$$

$$\lambda_{eff} = \min(C_\lambda d, \lambda_T) \quad (4.12)$$

$$\lambda_T = \frac{\sqrt{k_T}}{\omega} \quad (4.13)$$

where λ_{eff} is the effective turbulence length scale, d is the wall distance and $\Omega = \sqrt{2\Omega_{ik}\Omega_{ik}}$ is the absolute value of vorticity.

$$f_\nu = 1 - \exp\left(-\frac{\sqrt{Re_T}}{A_\nu}\right) \quad (4.14)$$

$$Re_T = \frac{f_W^2 k_T}{\nu \omega} \quad (4.15)$$

f_ν is the viscous damping function which takes care of the near wall effect and is a function of effective turbulence Reynolds number (Re_T)

The turbulent viscosity coefficient C_μ is defined as

$$C_\mu = \frac{1}{A_0 + A_s \left(\frac{S}{\omega}\right)} \quad (4.16)$$

$$f_{INT} = \min\left(\frac{k_T}{C_{INT} k_{TOT}}, 1\right) \quad (4.17)$$

f_{INT} is the intermittency damping function which controls the turbulence production.

The production of laminar kinetic energy is defined as

$$P_{kL} = \mu_{T,l} S^2 \quad (4.18)$$

where

$$\mu_{T,l} = \min\left[\rho f_{\tau,l} C_{11} \left(\frac{\Omega \lambda_{eff}^2}{\nu}\right) \sqrt{k_{T,l}} \lambda_{eff} + \beta_{TS} C_{12} Re_\Omega d^2 \Omega, \frac{0.5\rho(k_L + k_{T,l})}{S}\right] \quad (4.19)$$

The limit ensures the satisfaction of the realizability constraint on the contribution of the total Reynolds stress. The first and second term in the above equation is related to the development of Klebanoff modes and self-excited modes respectively.

$$f_{\tau,l} = 1 - \exp \left[-C_{\tau,l} \frac{k_{T,l}}{\lambda_{eff}^2 \Omega^2} \right] \quad (4.20)$$

$$k_{T,l} = k_T - k_{T,s} \quad (4.21)$$

where $k_{T,l}$ is the large scale turbulence, k_T is the transported turbulent kinetic energy and $k_{T,s}$ is the effective small scale turbulence.

$$\beta_{TS} = 1 - \exp \left(-\frac{\max (Re_{\Omega} - C_{TS,crit}, 0)^2}{A_{TS}} \right) \quad (4.22)$$

$$Re_{\Omega} = \frac{d^2 \Omega}{\nu} \quad (4.23)$$

The anisotropic near-wall dissipation terms appearing in the k_T and k_L transport equations (Eqs. 4.4 and 4.5) are defined as follows

$$D_T = \frac{1}{J^2} \mu \frac{\partial \sqrt{k_T} \beta_i^j}{\partial x_j} \frac{\partial \sqrt{k_T} \beta_i^j}{\partial x_j} \quad (4.24)$$

$$D_L = \frac{1}{J^2} \mu \frac{\partial \sqrt{k_L} \beta_i^j}{\partial x_j} \frac{\partial \sqrt{k_L} \beta_i^j}{\partial x_j} \quad (4.25)$$

The effective diffusivity α_T in the k_T and ω equations is defined as

$$\alpha_T = f_{\nu} C_{\mu,std} \sqrt{k_{T,s}} \lambda_{eff} \quad (4.26)$$

The kinematic damping function, f_{ω} of the boundary layer production term appearing in ω equation (Eq. 4.6) is defined as

$$f_{\omega} = 1 - \exp \left[-0.41 \left(\frac{\lambda_{eff}}{\lambda_T} \right)^4 \right] \quad (4.27)$$

The terms R_{BP} and R_{NAT} which appear with opposite sign in the k_T to k_L transport equations are related to the laminar-to-turbulent transition process and are defined as

$$R_{BP} = \rho C_R \beta_{BP} k_L \omega / f_{\omega} \quad (4.28)$$

$$R_{NAT} = \rho C_{R,NAT} \beta_{NAT} k_L \Omega \quad (4.29)$$

The initiation of bypass and natural transition is determined by the following threshold functions β_{BP} and β_{NAT} .

$$\beta_{BP} = 1 - \exp \left(-\frac{\phi_{BP}}{A_{BP}} \right) \quad (4.30)$$

$$\phi_{BP} = \max \left[\left(\frac{k_T}{\nu \Omega} - C_{BP,crit} \right), 0 \right] \quad (4.31)$$

$$\beta_{NAT} = 1 - \exp \left(-\frac{\phi_{NAT}}{A_{NAT}} \right) \quad (4.32)$$

$$\phi_{NAT} = \max \left[(Re_{\Omega} - C_{NAT,crit} / f_{NAT,crit}), 0 \right] \quad (4.33)$$

$$f_{NAT,crit} = 1 - \exp \left(-C_{NC} \frac{\sqrt{k_L} d}{\nu} \right) \quad (4.34)$$

The damping function $f_{NAT,crit}$ ensures the proper process of the natural transition. The turbulent viscosity used in the momentum equation is the sum of the small-scale and large-scale contributions and is defined as

$$\mu_T = \mu_{T,s} + \mu_{T,l} \quad (4.35)$$

The coefficient $C_{\omega 2}$ in ω transport equation (Eq. 4.6) is given by the following equation.

$$C_{\omega 2} = 0.92 f_W^2$$

The model constants are given in Table 4.1.

Table 4.1: $k_T - k_L - \omega$ model constants

$A_0 = 4.04$	$C_{INT} = 0.75$	$C_{\omega 1} = 0.44$
$A_s = 2.12$	$C_{TS,crit} = 1000$	$C_{\omega 3} = 0.3$
$A_\nu = 6.75$	$C_{R,NAT} = 0.02$	$C_{\omega R} = 1.5$
$A_{BP} = 0.6$	$C_{11} = 3.4 \times 10^{-6}$	$C_\lambda = 2.495$
$A_{NAT} = 200$	$C_{12} = 1.0 \times 10^{-10}$	$C_{\mu,std} = 0.09$
$A_{TS} = 200$	$C_R = 0.12$	$\sigma_k = 1$
$C_{BP,crit} = 1.2$	$C_{SS} = 1.5$	$\sigma_\omega = 1.17$
$C_{NC} = 0.1$	$C_{\tau,1} = 4360$	$C_{NAT,crit} = 1250$

The boundary conditions at solid wall are $k_T = k_L = 0$ and $\frac{\partial \omega}{\partial \eta} = 0$. At the inlet $k_L = 0$ whereas k_T and ω are prescribed based on the inlet flow parameters of the problem.

4.3.2 Correlation based $\gamma - Re_\theta$ SST transition model

The correlation based transition model was proposed by Menter *et al.* (2006) which is based on his original SST model, however the complete details of this model was given later by Langtry and Menter (2009) and Malan *et al.* (2009). This transition model solves transport equations for intermittency and the transition onset criteria in terms of momentum thickness Reynolds number along with k and ω transport equations of the SST turbulence model. In this model, the intermittency (γ) equation triggers the transition onset phenomenon and $\tilde{Re}_{\theta t}$ equation avoids the use of non-local variables. The transport equations are described below

γ - equation

$$\frac{\partial(\rho\gamma)}{\partial t} + \frac{1}{J} \frac{\partial}{\partial x_j} \left[\rho \langle U_i \rangle \gamma \beta_i^j - \frac{(\mu + \mu_t/\sigma_f)}{J} \left(\frac{\partial \gamma}{\partial x_m} \beta_i^j \beta_i^m \right) \right] = P_\gamma - E_\gamma \quad (4.36)$$

The source term P_γ is defined as

$$P_\gamma = F_{length} C_{a1} \rho S [\gamma F_{onset}]^{0.5} (1 - C_{e1} \gamma)$$

where $S = \sqrt{2S_{ik}S_{ik}}$ is the absolute value of strain rate .

The dimensionless functions, F_{onset} and F_{length} control the intermittency equation in the boundary layer.

F_{onset} controls the transition onset location using the following function

$$F_{onset} = \max(F_{onset2} - F_{onset3}, 0) \quad (4.37)$$

$$F_{onset2} = \min(\max(F_{onset1}, F_{onset1}^4), 2.0) \quad (4.38)$$

$$F_{onset3} = \max\left(1 - \left(\frac{R_T}{2.5}\right)^3, 0\right) \quad (4.39)$$

$$F_{onset1} = \frac{Re_\nu}{2.193 Re_{\theta c}} \quad (4.40)$$

$$R_T = \frac{\rho k}{\mu \omega} \quad (4.41)$$

$$Re_\nu = \frac{\rho y^2 S}{\mu} \quad (4.42)$$

The destruction (relaminarization), E_γ term is defined as follows

$$E_\gamma = C_{a2} \rho \Omega \gamma F_{turb} (C_{e2} \gamma - 1)$$

F_{turb} controls the destruction/re-laminarization of the boundary layer is given by

$$F_{turb} = \exp[-(R_T/4)^4] \quad (4.43)$$

The correlation for F_{length} which controls the length of the transition region and Re_{θ_c} which is the critical momentum thickness Reynolds number where the intermittency first starts to grow were not published initially, but in 2009 these correlations were simultaneously given by Langtry and Menter (2009) and Malan *et al.* (2009). These correlations were basically arrived based on physical intuitions and numerical experiments. In the present formulation which was implemented in 3D-PURLES flow code, the F_{length} of Malan is retained and Re_{θ_c} is modified based on our validation study carried out for the ERCOFTAC flat plate T3A test case (Kulkarni *et al.*, 2019). The final form of the correlations are

$$F_{length} = \min \left[\exp \left(7.168 - 0.01173 \tilde{Re}_{\theta_t} \right) + 0.5, 300 \right] \quad (4.44)$$

$$Re_{\theta_c} = \min \left(0.665 \tilde{Re}_{\theta_t} + 66.5, \tilde{Re}_{\theta_t} \right) \quad (4.45)$$

A Neumann boundary condition (zero flux) is applied for γ at the wall and at inlet $\gamma = 1$.

The transport equation for the transition momentum-thickness Reynolds number $\tilde{Re}_{\theta_t}$ is

$\tilde{Re}_{\theta_t}$ - equation

$$\frac{\partial(\rho \tilde{Re}_{\theta_t})}{\partial t} + \frac{1}{J} \frac{\partial}{\partial x_j} \left[\rho \langle U_i \rangle \tilde{Re}_{\theta_t} \beta_i^j - \frac{\sigma_{\theta_t}(\mu + \mu_t)}{J} \left(\frac{\partial \tilde{Re}_{\theta_t}}{\partial x_m} \beta_i^j \beta_i^m \right) \right] = P_{\theta_t} \quad (4.46)$$

The source term is given by the following relation which ensures that the transported scalar $\tilde{Re}_{\theta_t}$ matches the local value of Re_{θ_t} outside the boundary layer

$$P_{\theta_t} = C_{\theta_t} \frac{\rho}{t} \left(Re_{\theta_t} - \tilde{Re}_{\theta_t} \right) (1.0 - F_{\theta_t}) \quad (4.47)$$

$$t = \frac{500\mu}{\rho U^2} \quad (4.48)$$

Outside the boundary layer, the $P_{\theta t}$ relation ensures that the transported scalar $\tilde{Re}_{\theta t}$ matches with the local value of $Re_{\theta t}$ which is calculated from the empirical correlation given in Eqs. 4.52 and 4.53. The time scale t is used in the source term to scale with the convective and diffusive terms of the transport equation. The blending function, $F_{\theta t}$ becomes zero in the freestream and one in the boundary layer and is defined as

$$F_{\theta t} = \min \left(\max \left(F_{wake} e^{-\left(\frac{y}{\delta}\right)^4}, 1.0 - \left(\frac{\gamma C_{e2} - 1}{C_{e2} - 1} \right)^2 \right), 1.0 \right) \quad (4.49)$$

$$F_{wake} = e^{-\left(\frac{Re_{\omega}}{10^5}\right)^2}; \quad Re_{\omega} = \frac{\rho \omega y^2}{\mu} \quad (4.50)$$

$$\delta = \frac{50\Omega y}{U} \delta_{BL} \quad \delta_{BL} = \frac{15}{2} \theta_{BL}; \quad \theta_{BL} = \frac{\tilde{Re}_{\theta t} \mu}{\rho U}; \quad (4.51)$$

The empirical correlations for $Re_{\theta t}$ which is function of turbulence intensity (T_u) is given by Langtry and Menter (2009) as follows

$$Re_{\theta t} = \begin{cases} \left[1173.51 - 589.428 T_u + \frac{0.2196}{T_u^2} \right] F(\lambda_{\theta}), & T_u \leq 1.3 \\ 331.50 [T_u - 0.5658]^{-0.671} F(\lambda_{\theta}), & T_u > 1.3 \end{cases} \quad (4.52)$$

$$F(\lambda_{\theta}) = \begin{cases} 1 - [-12.986\lambda_{\theta} - 123.66\lambda_{\theta}^2 - 405.689\lambda_{\theta}^3] e^{-\left(\frac{T_u}{1.5}\right)^{1.5}}, & \lambda_{\theta} \leq 0 \\ 1 + 0.275 [1 - e^{-35.0\lambda_{\theta}}] e^{-\left[\frac{T_u}{0.5}\right]}, & \lambda_{\theta} > 0 \end{cases} \quad (4.53)$$

$$\lambda_{\theta} = \frac{\rho \theta^2}{\mu} \frac{dU}{ds} \quad (4.54)$$

$$T_u = 100 \frac{\sqrt{\frac{2k}{3}}}{U} \quad (4.55)$$

where dU/ds is the acceleration along the streamwise direction and can be computed by taking the derivative of the local velocity U in the x_i directions and then summing the contribution of these derivatives along the streamwise flow direction.

$$U = (u_i u_j \delta_{ij})^{\frac{1}{2}} \quad (4.56)$$

$$\frac{dU}{dx_1} = \frac{1}{2U} \left[2u_j \frac{1}{J} \beta_{1k} \frac{\partial u_j}{\partial x_k} \right] \quad (4.57)$$

$$\frac{dU}{dx_2} = \frac{1}{2U} \left[2u_j \frac{1}{J} \beta_{2k} \frac{\partial u_j}{\partial x_k} \right] \quad (4.58)$$

$$\frac{dU}{dx_3} = \frac{1}{2U} \left[2u_j \frac{1}{J} \beta_{3k} \frac{\partial u_j}{\partial x_k} \right] \quad (4.59)$$

$$\frac{dU}{ds} = \left[\frac{U_1}{U} \frac{dU}{dx_1} + \frac{U_2}{U} \frac{dU}{dx_2} + \frac{U_3}{U} \frac{dU}{dx_3} \right] = \frac{u_i u_j}{U^2} \frac{1}{J} \beta_{ik} \frac{\partial u_j}{\partial x_k} \quad (4.60)$$

The following limiters are suggested by Langtry and Menter (2009) in order to ensure numerical stability.

$$-0.1 \leq \lambda_\theta \leq 0.1 \quad Tu \geq 0.027 \quad Re_{\theta t} \geq 20$$

The boundary condition for $\tilde{Re}_{\theta t}$ at the wall is zero flux. The boundary condition for $\tilde{Re}_{\theta t}$ at an inlet is calculated from the empirical correlation given Eq. 4.52 based on the inlet turbulence intensity.

The model constants used in γ and $\tilde{Re}_{\theta t}$ equation are given in the following table 4.2.

Table 4.2: Model constants for γ and $\tilde{Re}_{\theta t}$ equation

$C_{e1} = 1.0$	$C_{a1} = 2.0$	$\sigma_f = 1.0$
$C_{e2} = 50$	$C_{a2} = 0.06$	$\sigma_{\theta t} = 2.0$
$C_{\theta t} = 0.03$	$s_1 = 2$	

k - equation

The transport equation of the kinetic energy k used in the SST turbulence model is modified to handle the effects of transition. The modified equation is as follows

$$\frac{\partial(\rho k)}{\partial t} + \frac{1}{J} \frac{\partial}{\partial x_j} \left[\rho \langle U_i \rangle k \beta_i^j - \frac{(\mu + \mu_t \sigma_k)}{J} \left(\frac{\partial k}{\partial x_m} \beta_i^j \beta_i^m \right) \right] = \tilde{P}_k - \tilde{D}_k \quad (4.61)$$

$$\tilde{P}_k = \gamma_{eff} P_k \quad \tilde{D}_k = \min(\max(\gamma_{eff}, 0.1), 1.0) D_k \quad (4.62)$$

The production (P_k), dissipation (D_k) used in SST turbulence model (Menter, 1994; Subrahmanya and Rajani, 2017) are given as below.

$$P_k = 2\mu_t |S|^2 = 2\mu_t S_{ik} S_{ki} \quad D_k = \beta^* \rho k \omega \quad (4.63)$$

where

$$S_{ik} \text{ is the mean strain rate} = \frac{1}{2} \frac{1}{J} \left[\frac{\partial \langle U_i \rangle}{\partial x_j} \beta_j^k + \frac{\partial \langle U_k \rangle}{\partial x_j} \beta_j^i \right]$$

The effective intermittency, γ_{eff} is obtained by

$$\gamma_{eff} = \max(\gamma, \gamma_{sep}) \quad (4.64)$$

In order to handle separation-induced transition the γ_{sep} is written as

$$\gamma_{sep} = \min \left(s_1 \max \left[0, \left(\frac{Re_\nu}{3.235 Re_{\theta c}} \right) - 1 \right] F_{reattach}, 2 \right) F_{\theta t} \quad (4.65)$$

where Re_ν and $F_{\theta t}$ are obtained from Eqs. 4.42 and 4.49.

$$F_{reattach} = e^{-\left(\frac{R_T}{20}\right)^4} \quad (4.66)$$

where R_T is obtained from Eq. 4.41

The boundary condition for k at solid wall is zero whereas k at the inlet is prescribed based on the inlet flow parameters of the problem.

ω - equation

The dissipation equation ω given below is same as that of SST turbulence model (Menter, 1994; Subrahmanya and Rajani, 2017)

$$\begin{aligned} \frac{\partial(\rho\omega)}{\partial t} + \frac{1}{J} \frac{\partial}{\partial x_j} \left[\rho \langle U_i \rangle \omega \beta_i^j - \frac{(\mu + \mu_t/\sigma_\omega)}{J} \left(\frac{\partial \omega}{\partial x_m} \beta_i^j \beta_i^m \right) \right] &= \frac{\gamma \rho}{\mu_t} P_k - \beta \rho \omega^2 \\ &+ 2\rho(1 - F_1)\sigma_{\omega 2} \frac{1}{\omega} \frac{1}{J^2} \frac{\partial (k \beta_i^j)}{\partial x_j} \frac{\partial (\omega \beta_i^j)}{\partial x_j} \end{aligned} \quad (4.67)$$

In the $\gamma - Re_\theta$ SST transition model, the function F_1 used in the above equation is given by

$$F_1 = \max(F_{1SST}, F_3); \quad F_3 = e^{-\left(\frac{R_y}{120}\right)^8}; \quad R_y = \frac{\rho y \sqrt{k}}{\mu} \quad (4.68)$$

Where the F_{1SST} is given by :

$$\begin{aligned} F_{1SST} &= \tanh(arg_1^4) \\ arg_1 &= \min \left[\max \left(\frac{\sqrt{k}}{\beta^* \omega d}, \frac{500\nu}{d^2 \omega} \right); \frac{4\rho k}{\sigma_{\omega 2} C D_{k\omega} d^2} \right] \\ C D_{k\omega} &= \max \left(2\rho \frac{1}{\sigma_{\omega 2}} \frac{1}{\omega} \frac{1}{J^2} \frac{\partial (k \beta_i^j)}{\partial x_j} \frac{\partial (\omega \beta_i^j)}{\partial x_j}, 10^{-10} \right) \end{aligned}$$

where d is the distance from the field point to the nearest wall.

In this model, the turbulent (or) eddy viscosity is evaluated in the following way as suggested by Malan *et al.* (2009)

$$\mu_T = \rho k T \quad (4.69)$$

$$T = \min \left[\frac{1}{\max \left(\omega; \frac{SF_{2SST}}{a_1} \right)}, \frac{0.6}{\sqrt{3}S} \right]$$

Where the F_{2SST} is given by :

$$F_{2SST} = \tanh(arg_2^2)$$

$$arg_2 = \max \left(\frac{2\sqrt{k}}{\beta^* \omega d}; \frac{500\nu}{d^2 \omega} \right)$$

The model constants of SST model are evaluated following a linear combination of the constants used in the standard $k - \epsilon$ (suffixed with 1) and the Wilcox $k - \omega$ model (suffixed with 2) as

$$\phi = F_{1SST}\phi_1 + (1 - F_{1SST})\phi_2 \quad \text{where; } \phi = \sigma_k, \sigma_w, \gamma \text{ or } \beta$$

The constants are :

$$\beta^* = 0.09, \kappa = 0.41, a_1 = 0.31$$

$$\sigma_{k1} = 1/0.85, \sigma_{\omega1} = 1/0.5, \beta_1 = 0.075, \gamma_1 = \beta_1/\beta^* - \sigma_{\omega1}\kappa^2/\sqrt{\beta^*}$$

$$\sigma_{k2} = 1.0, \sigma_{\omega2} = 1/0.856, \beta_2 = 0.0828, \gamma_2 = \beta_2/\beta^* - \sigma_{\omega2}\kappa^2/\sqrt{\beta^*}.$$

The boundary condition for ω used at the wall :

$$\omega_{\text{wall}} = \frac{6\mu}{\rho\beta_1(\Delta d_1)^2}; \quad \Delta d_1 = \text{near wall distance}$$

4.4 Finite Volume Method for Flow Solution

The transformation of the nonlinear Navier-Stokes equations (Eq. 4.1) to discrete algebraic equations and the finite volume formulation used in the present algorithm is described briefly in the

following sub sections.

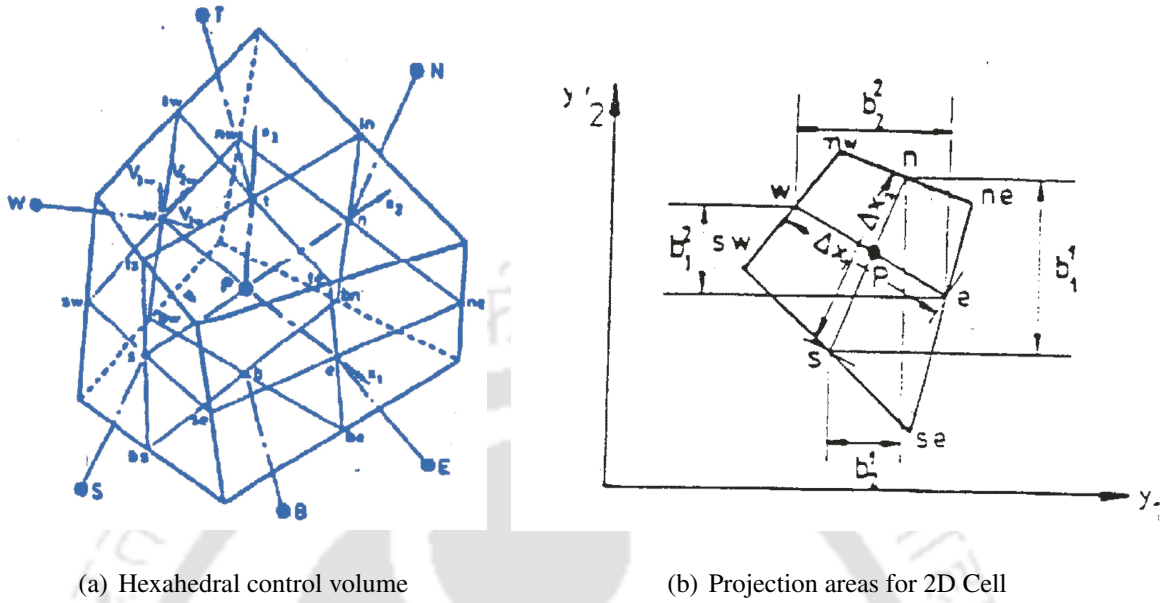


Figure 4.1: Typical control volume and cell face projection area

4.4.1 Control volume and variable arrangement

In any finite volume method, the governing differential equations for flow are to be integrated over a finite number of control volumes covering the computation domain obtained by the grid generation procedure. Fig. 4.1(a) shows a typical 3D hexahedral control volume and the relevant variable arrangement used in the present method. The grid generation procedure calculates the coordinates of the control volume vertices which are simply joined by linear segments to form the six boundary planes and hence the volume. All the variables are stored at the geometric center P of the control volume. The six neighbouring control volume centers are denoted by N, S, E, W, T and B for the north, south, east, west, top and bottom neighbours respectively. The face center points n, s, e, w, t and b are located at the intersection of the lines joining the midpoint of the opposite edges, eg., ne and se are the middle points of the edge forming the east edge with e at its center. These points specified on edges are used for locating the flow variables and their gradients on the control volume faces. The Cartesian velocities at the control volume faces of control volume are computed and finally only the total convective fluxes at the cell faces are stored.

4.4.2 Geometrical parameters

The most important geometrical parameters required in the present formulation are the metric coefficients β_i^j , which may always be expressed in terms of the projection area of the control volume faces as follows:

$$\beta_i^j = b_i^j \Delta x_j / (\Delta x_1 \Delta x_2 \Delta x_3) \quad (4.70)$$

where the projection areas b_i^j for the control volume around P are shown in Fig. 4.1(b) for a two-dimensional situation on $y_1 - y_2$ plane. The other geometrical quantities of interest are the normal distance of a near boundary node from the boundary surface, the volume and the projection of boundary face areas on the Cartesian reference planes. The calculation of these quantities however involves only some simple arithmetic operations.

4.4.3 Flux balance equation

Integration of the conservation equations over the control volumes, using the divergence theorem of Gauss, yields the resulting flux balance equation for each control volume and each variable given by

$$\int_{\Delta V} \frac{\partial \phi}{\partial t} dV = \int_{\Delta V} S_\phi dV - (I_e - I_w + I_n - I_s + I_t - I_b) \quad (4.71)$$

where, I_m represents the total convective cum diffusive flux across the face ' m ' of the control volume (Fig. 4.1(a)). Each of the surface fluxes I_e, I_w etc. is made of three distinct parts, namely a *Convective* contribution I^C , a *Normal Diffusive* contribution I^{DN} , and a *Cross-Diffusive* part I^{DC} . Eq. 4.71 involves no approximation and represents the finite volume analogue of the relevant conservation equation. The algebraic expression for convective and diffusive fluxes of ϕ through the cell face w , for example, can be written in summation convention as:

$$I_w^C = \rho_w b_{jw}^1 V_{jw} \phi_w = C_w \phi_w \quad (4.72)$$

$$I_w^{DN} = -(\mu/\Delta V)_w B_{1w}^1 (\phi_P - \phi_W) \quad (4.73)$$

$$I_w^{DC} = (\mu/\Delta V)_w [B_{2w}^1 (\phi_{nw} - \phi_{sw}) + B_{3w}^1 (\phi_{tw} - \phi_{bw})] \quad (4.74)$$

where, $B_j^i = b_k^i b_k^j$, b_j^i are the projection area of cell faces on the Cartesian planes and ΔV is the cell volume.

The diffusion flux, I^{DN} , arising out of the gradient along a direction normal to the cell face, is coupled to the total convective flux, I^C , to calculate the main coefficients of the difference equations. The cross-derivative diffusion flux, I^{DC} , is however treated explicitly as a pseudo-source term to avoid the possibility of producing negative coefficients in an implicit treatment. The numerical evaluation of the different terms in Eq. 4.71 requires the calculation of geometric factors for the non-orthogonal control volumes and the introduction of a suitable spatial discretisation scheme for interpolating quantities at the cell faces (ϕ_w) from their nodal values at either side of the face considered. The evaluation of the cell face value ϕ_w appearing in the convective flux I_w^C is critical for successful convergence of the algorithm and a variety of convective discretisation schemes have been incorporated in the 3D-PURLES flow code. The unsteady term in Eq. 4.71 is replaced either by the first or second order temporal discretisation scheme.

4.4.4 Spatial discretisation

The cell face fluxes are evaluated using the chosen discretisation scheme, which basically decides the value or the gradient of the flow variable at the cell faces. The so-called Central Difference scheme using linear inter-nodal variation is used for diffusive fluxes. A variety of low-diffusion high order Upwind schemes for the convective fluxes have therefore been formulated, assessed for their relative performance and used in the present algorithm. Different spatial discretisation schemes available in this code are, upwind (Patankar, 1980), second order accurate central difference (Patankar, 1980), a third order accurate QUICK (Quadratic Upwind Interpolation for Convection Kinematics) scheme proposed by Leonard (1979), second order accurate linear upwinding

(Peric, 1985) and a second order accurate composite scheme, HLP (Hybrid Linear Parabolic Approximation) of Zhu (1991). The detailed algebra of the schemes are discussed by Kulkarni *et al.* (2001). For the present work, the upwind, central difference and QUICK spatial discretisation schemes are used which are described briefly as below.

Pure Upwind Scheme:

The first order accurate Pure Upwind scheme which takes care of the direction of the velocity signal propagation and assumes the variable value at the cell face to be the same as the value at the immediate upstream node as shown in Fig. 4.2(a).

Central Difference Scheme (CDS):

The second order accurate Central Difference Scheme (CDS) interpolates the value of the variable at the cell face by assuming a linear variation between the values of the variable at the cell centres upstream and downstream of a cell face considered as shown in Fig. 4.2(b).

QUICK (Quadratic Upwind Interpolation for Convective Kinematics) Scheme:

The third order accurate QUICK scheme assumes a quadratic upwind interpolation for the cell face value of the variable by passing a second order polynomial through the downstream node, upstream node and one node further upstream nodes of the cell face in question depending on the flow direction Fig. 4.2(c).

Deferred Correction Procedure:

The numerical instabilities for higher order schemes are avoided using a deferred correction procedure of Khosla and Rubin (1974). In this procedure, a suitable weighting function ($0 \leq \lambda_{DS} \leq 1$) is used to blend the flux from the desired scheme (DS) with upwind fluxes which allows some small numerical diffusion but ensures numerical stability of the solution.

$$\text{Flux} = \lambda_{DS} \text{ (Desired Scheme)} + (1 - \lambda_{DS}) \text{ (Upwind Scheme)} \quad (4.75)$$

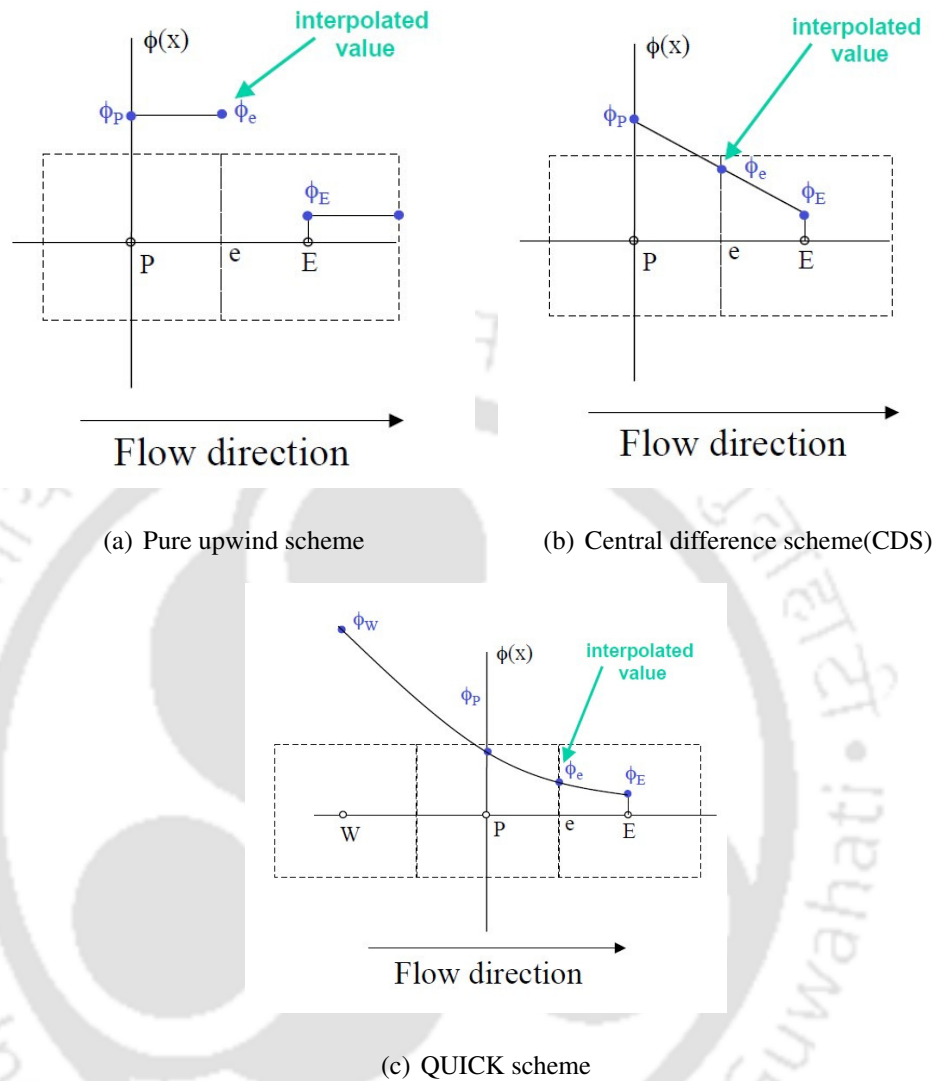


Figure 4.2: Schematic sketch of the different spatial discretisation schemes

4.4.5 Temporal discretisation

The time derivative appearing in the momentum equations is discretised using either the first order Euler implicit scheme or second order backward implicit scheme given below. The first order scheme uses two level time information whereas the second order scheme uses the three level time information.

First order Euler implicit scheme:

$$\left. \frac{\partial \phi}{\partial t} \right|_{(n+1)} = \frac{\phi^{(n+1)} - \phi^{(n)}}{\Delta t} \quad (4.76)$$

Second order backward implicit scheme:

$$\left. \frac{\partial \phi}{\partial t} \right|_{(n+1)} = \frac{1.5\phi^{(n+1)} + 0.5\phi^{(n-1)} - 2\phi^{(n)}}{\Delta t} \quad (4.77)$$

where $\phi^{(n+1)}$, $\phi^{(n)}$ and $\phi^{(n-1)}$ are values of the flow variable ϕ at $n + 1$, n and $n - 1$ time steps respectively and $\Delta t = t^{(n+1)} - t^{(n)}$ is the time step size. These two temporal discretisation schemes can be expressed in a generalized form as follows

$$\left. \frac{\partial \phi}{\partial t} \right|_{(n+1)} = \frac{(1 + \gamma)\phi^{n+1} + \gamma\phi^{(n-1)} - (1 + 2\gamma)\phi^n}{\Delta t} \quad (4.78)$$

where $\gamma = 0$ for the Two Level and $\gamma = 0.5$ for the Three level versions of the scheme: Irrespective of the scheme used, the implementation of the scheme is always done in an implicit manner leading to a system of linear equations coupling the variable values over the field.

4.4.6 Finite volume equation

The convective ($C_{i\phi}$) and diffusive ($D_{i\phi}$) fluxes computed according to the discretisation scheme used, when replaced in Eq. 4.71, lead to the conservation of fluxes in the form of a quasilinear equation for the respective flow variable solved for. The source terms are often non-linear and in order to enhance the numerical stability, the source terms for the steady state equations are linearised as:

$$\int JS_{\phi} dx_1 dx_2 dx_3 = SU^{\text{steady}} + \phi_P SP^{\text{steady}} \quad (4.79)$$

Replacing the different fluxes and the source terms for the spatial and temporal derivatives, the flux balance equation can be recast into the following quasilinear form.

$$\Delta V \left(\frac{(1 + \gamma)\phi_P^{(n+1)} + \gamma\phi_P^{(n-1)} - (1 + 2\gamma)\phi_P^{(n)}}{\Delta t} \right) + \left(\sum_{nb} A_{nb} - SP^{\text{steady}} \right) \phi_P^{(n+1)} = \sum_{nb} A_{nb}\phi_{nb}^{(n+1)} + SU^{\text{steady}} \quad (4.80)$$

Eq. 4.80 simplifies further to the following form:

$$\left(\sum_{nb} A_{nb} - SP^{\text{unsteady}} \right) \phi_P^{(n+1)} = \sum_{nb} A_{nb}\phi_{nb}^{(n+1)} + SU^{\text{unsteady}}$$

where,

$$SU^{\text{unsteady}} = SU^{\text{steady}} - \frac{\Delta V}{\Delta t} (\gamma\phi^{(n-1)} - (1 + 2\gamma)\phi^{(n)})$$

$$SP^{\text{unsteady}} = SP^{\text{steady}} - \frac{\Delta V}{\Delta t} (1 + \gamma)$$

The coefficient A_{nb} represents the combined effect of convection and diffusion at the six faces of the cell denoted by the suffix nb , SU and SP are the components of the linearised source term S_ϕ . The detailed derivation of Eq. 4.80 expressing A_{nb} as a function of cell face projection areas, velocity and diffusion coefficient at cell faces and variable values at the cell centre is given by Majumdar *et al.* (1992). The transformation of continuity equation to an equation of pressure-correction, described in the following subsection.

4.4.7 Computation of pressure field

The present algorithm uses the pressure-velocity solution strategy SIMPLE (Semi Implicit Pressure Linked Equation) algorithm proposed by Patankar and Spalding (1972) which has been modified for collocated variable arrangement. The pressure-velocity solution strategy transforms the continuity equation into a field equation for pressure correction using momentum equations as link between pressure and velocity corrections. The problem of the so-called checkerboard splitting encountered using a collocated variable arrangement is avoided by employing the principle of Momentum Interpolation (Majumdar, 1988; Majumdar *et al.*, 1992). This gives the cell-face velocity components, U_{iw} (as an illustration, at the cell west face 'w') and hence the link between pressure and velocity correction U'_{iw} as:

$$U_{iw} = \alpha_v \left[\overline{H_P^i} + \overline{D_2^i(P_s - P_n)} + \overline{D_3^i(P_b - P_t)} + \overline{D_1^i(P_w - P_p)} \right] + (1 - \alpha_v)U_{iw}^0 \quad (4.81)$$

$$U'_{iw} = \alpha_v \overline{D_1^i}(p'_w - p'_p) \quad (4.82)$$

where $H_P^i = \left(\sum_{nb} A_{nb}^i U_{nb}^i + SU \right) / A_P^i$ and $D_k^i = b_k^i / A_P^i$; A_{nb}^i represents the coefficients for neighbouring nodes, the superscript '0' is the value at the previous iteration, A_P^i is the value of A_P for U_i at the node P , b_k^i is the relevant projection area of the cell face (Fig.4.1(b)), α_v is an under-relaxation parameter and the expressions with overbar represent the linear average of the same quantities evaluated at the cell-centers P and W adjacent to the face ' w '. Now the conservation of mass in a control volume may be expressed as:

$$C_e - C_w + C_n - C_s + C_t - C_b = 0 \quad (4.83)$$

where the face mass flux C_w , for example, may be written as follows using ' i ' as the summation index for three directions.

$$C_w = \rho_w b_{iw}^1 (U_{iw} + U'_{iw}) \quad (4.84)$$

Substituting U_{iw} and U'_{iw} for all ' i ' from equations similar to Eqs. 4.81 and 4.82 respectively in mass flux expression similar to Eq. 4.84 for each of the six faces in Eq. 4.83, one obtains the equation of pressure correction in a form very similar to Eq. 4.80. Once the field of pressure correction is solved, corrections are applied to the cell-centre pressure and velocities, to all the cell-face velocities and finally to the convective mass-fluxes to satisfy the cell wise continuity.

4.4.8 Boundary conditions

In addition to the discretised governing equations the complete specifications of the mathematical problem requires incorporation of proper boundary conditions for all the variables. The most

frequently encountered boundary conditions and the relevant treatment required in the present procedure are described below. The appropriate boundary conditions used for the flow past an airfoil are shown in Fig.4.3.

- Farfield: The farfield is treated either as an inflow boundary condition or an outflow boundary condition depending on the sign of the convective flux on the relevant face.
 - Inflow: The values of all dependent variables are normally known at inflow boundaries. In the present problem a uniform flow is prescribed at the inflow boundaries.
 - Outflow(Exit): Similar to inflow, the profiles of the variables may also be prescribed at the outlet. However, in general, very little is known apriori about the outlet and therefore it is, advisable to place the outflow boundaries far downstream with no reverse flow so that the results in the region of interest are practically insensitive to the treatment at outflow boundary. The field values at outflow are extrapolate from upstream neighbours in the interior of the computation domain. In case of unsteady flow computation, convective outflow condition is used which assumes the total derivative $\frac{D\phi}{Dt} = 0$ for any flow variable ϕ .
- Symmetry: The conditions to be satisfied at an axis or plane of symmetry are of zero flux across the plane or axis in question. The convective fluxes are reduced to zero through prescription of zero convective coefficients and the diffusive fluxes are explicitly forced to zero. Further, the boundary values are updated using the interior field values so that the required gradient condition is fulfilled.
- Walls: According to the no slip condition at wall, the velocity and the convective flux at any node lying on the wall surface should be zero relative to the wall. The diffusive fluxes at the wall are also set to zero. The wall shear effect is simulated through appropriate source terms in momentum equation.
- Cut boundary: The multiblock computations require a special treatment for cut boundaries at the block-to-block interface in order to simulate these boundary nodes exactly like any other

interior flow solution nodes. For the inter block cut boundaries two overlapping (dummy) control volumes are provided on either side of the cut boundary as shown in the right-hand side of Fig. 4.3. Now for control volume on one side of the cut, the field variable is solved for as an interior node of the relevant block (R_1 and L_2) and the field values at the dummy nodes (L_1 and R_2) are transferred from the solution of the immediate neighbouring block.

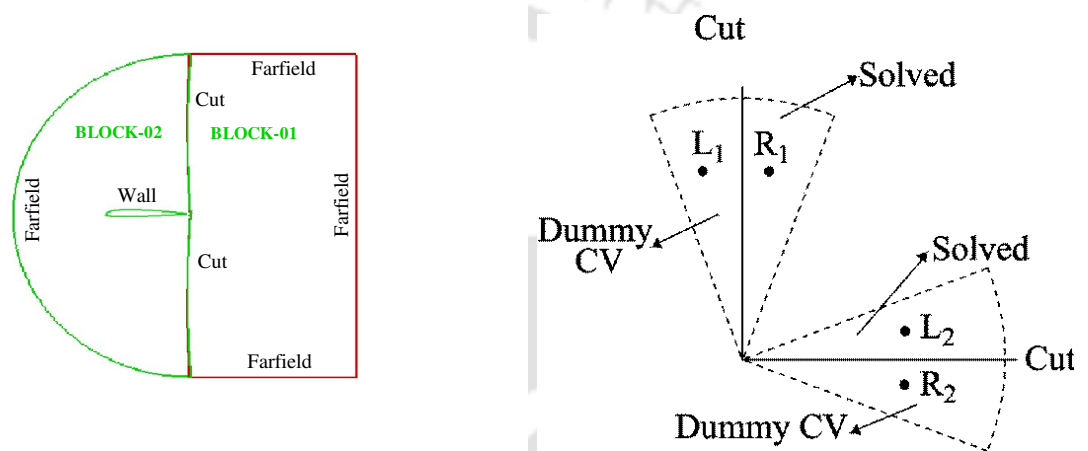


Figure 4.3: Boundary conditions for a two-block computation of flow past an airfoil

4.4.9 Multiblock computation for implicit solvers

With the rapid development of parallel computers during the last decade, the concept of multidomain/multiblock computation has been accepted by the CFD community as a very efficient technique for numerical solution of flow equations for very large scale problems requiring order of millions of control volumes covering the computation domain. Multidomain computation is often obligatory for CFD analysis of turbulent flow past complex configurations. The geometrical complexity requires use of multiblock grid where different blocks are often arranged to accommodate different kind of geometries covering the whole computation domain. Each block consists of a few physical flow boundaries and a few artificial computational boundaries (block to block interfaces)

required for the purpose of computation of flow in each block as a separate domain. The most critical consideration in any multidomain computation scheme is how to organise the in-process data transfer from one block to the neighbouring one in such a way that the artificially cut boundary interface plane separating the two adjacent blocks is treated exactly like any other interior cell face of the domain in question.

The domain decomposition technique divides the whole computational domain into a number of subdomains or blocks depending on the complexity of the problem considered and the memory availability of the computer used. Computation is carried out separately for each block and the inter-block influences are effected in the computation through appropriate data transfer facilities for a pre-decided overlapping region between two adjacent blocks. Suppose a numerical solution is sought for an elliptic equation $L\phi = 0$ in the rectangular computation domain ABCD as shown in Fig. 4.4, where L stands for a differential operator. In a simple two-block situation, the domain ABCD is partitioned into two rectangles AQRD and PBCS which share the two columns of overlapping control volumes on either side of the cut boundary. Initial field values of the flow variable ϕ are prescribed at all the nodes and one sweep of the solution of ϕ is carried out separately for the two blocks. After each sweep, field data at the cell centre and the fluxes at appropriate faces of the overlapping control volumes are transferred to the respective block as boundary conditions. The values of these flow variables on each of two planes are written to and read from direct access file with suitable index identification system to ensure that field data transfer takes place at the right location of the domain. The progress of the numerical process is often sensitive to the value of the pressure field transferred from the adjacent blocks which often has unrealistic values in the early phase of iteration, leading to a total collapse of the numerical process. Consideration of overall continuity for the whole computation domain including the flow through the cut boundaries and corresponding velocity field correction at the outflow planes for every iteration is essential in order to enhance the numerical convergence of the scheme.

The other important fact to consider for incompressible flow computation using domain decomposition methodology is to ensure overall mass continuity at every iteration.

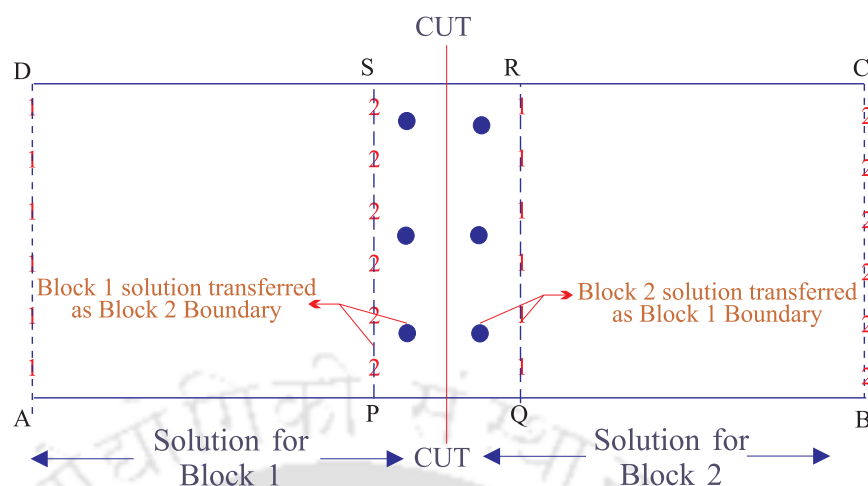


Figure 4.4: Schematic arrangement of domain decomposition

4.4.10 Parallelisation of the flow solution algorithm

Successful validation of a multiblock flow solution algorithm (Mohan *et al.*, 2000) on a sequential machine ensures the correctness of the implementation of the domain decomposition methodology for numerical solution of flow problems of a very large size. However since only one block is computed at any point of time the CPU time required is often prohibitive when the number of blocks is quite large. A realistic turn-around time for real life design problem with adequate resolution necessitates speeding up of the computation. In a parallel computation environment, each one of the multiple processors can compute the flow for one or more block in parallel and hence speed up computation. The computation speed is certainly increased since more than one blocks may be computed simultaneously. But in case of the multiprocessor parallel computation, the transfer of processor-interface data as boundary conditions from one processor to its neighbours again consumes some communication time in addition to the solution computation time taken by each processor in use. More the number of processors used, more will be the number of processor interfaces, requiring more communication time.

In the typical situation of a flow domain consisting of 16 blocks computed by four processors in parallel, for example, each processor can be assigned four blocks to be computed sequentially as shown in Fig. 4.5. However it is not obligatory to assign same number of blocks in each processor.

The block distribution among the processors available is decided mostly by the consideration of load balancing. Each processor is attempted to be loaded with approximately same number of control volumes so that the computation time is more or less of the same order for each processor. The figure clearly distinguishes the physical boundaries from the processor to processor interfaces and block to block interfaces. The data transfer procedure for inter- and intra- processor cut are made effective through the read/write statements operated on the direct access files, as stated in the previous section.

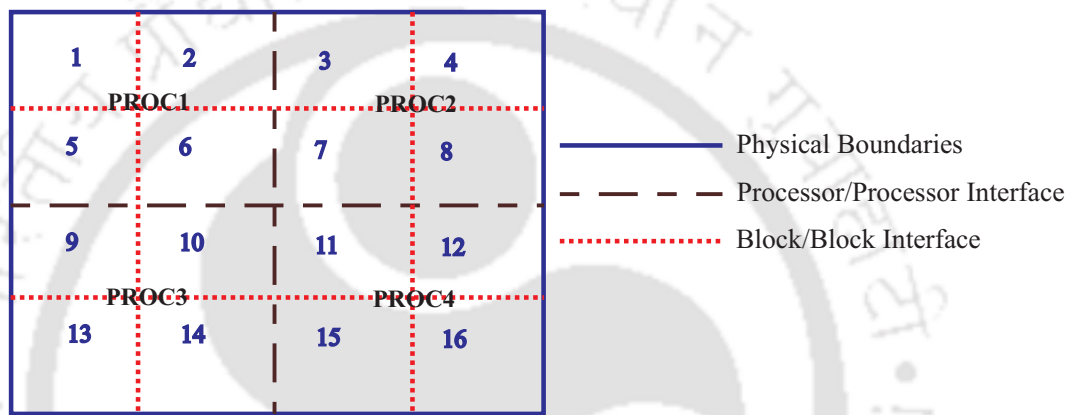


Figure 4.5: Parallelisation scheme

The main program controlling the execution is loaded on each of the processors used, where one of the processors is identified as the host processor. After every iteration sweep, the sum of residues from all control volumes are collected from each processor and the global residue is computed in the host for checking against the specified convergence criterion.

4.4.11 Computation scheme

The present method solves the set of the discretised equations for the complete domain using an iterative algorithm sequentially, taking one equation at a time. The iteration takes care of both the strong coupling amongst the equations as well as the non-linearities involved in the governing NS equations. The linearised form of equation to be solved for a variable with implicit under relaxation

is:

$$\phi_P = \frac{\sum A_{nb}\phi_{nb} + SU + (1 - \alpha_\phi)\phi_P^0 A_p}{A_p/\alpha_\phi}$$

For computation of the flow, the unknown ϕ in these equations are the three Cartesian velocity components, the turbulence scalars and the pressure correction that modifies the pressure and velocity field after every iteration. Since the equations are non-linear, the coefficients A_{nb} are functions of the dependent variables themselves and the algorithm must provide effective means for calculating these coefficients. Strictly speaking, such coupled non-linear system of equations need simultaneous solution of all the variables. But the iterative algorithm used in the present code updates the variables in a sequential (decoupled) manner and the solution is arrived at iteratively.

The code is provided with two different types of linear solvers, (i) an Alternating Direction type line iterative solver based on Tri-Diagonal Matrix Algorithm and (ii) the Strongly Implicit Procedure (SIP) of Stone (1968). For momentum and scalar equations where the convective terms are quite strong, only one inner sweep of the linear solver is enough to bring down the residues to a reasonably low level. Since a good mass conservation is crucial at every outer iteration for efficient and reliable solution using a segregated (decoupled) approach, a given percentage of the initial residuum, as proposed by Vandoormal and Raithby (1984), is prescribed as termination criterion for the inner sweep of pressure correction. For the present problem, the under relaxation factor used for the velocity components (α_v) was 0.8 and for pressure field (α_P) was 0.2 [$\alpha_P = (1 - \alpha_v)$]. For all the turbulent scalars the under relaxation parameter used was 0.5.

4.4.12 Sequence of the multiblock flow solution algorithm

The sequence in which the unsteady calculation is carried out is as follows:

1. Set all field values of flow variables (ϕ^0) as per the initial condition ($t = 0$).
2. Solve the discretised momentum equations (Eq. 4.80) to obtain U_1 , U_2 and U_3 field using an iterative procedure in the so called inner iteration loop. For the momentum equations only inner iteration loop is carried out since these equations are mainly dominated by the convective terms.

3. Compute cell-face velocities using momentum interpolation between adjacent control volumes; compute cell-wise mass imbalance and solve for the pressure correction (p') field iteratively (Eq. 4.80) in the 10 inner iteration loop or till the 10% fall in residue.
4. Correct the pressure field and velocities at cell center and cell faces.
5. Solve if required, the discretised conservation equation of passive scalar representing temperature, concentration or turbulence scalars iteratively, using only one inner iteration loop similar to the momentum equations.
6. For turbulent flow update the eddy viscosity (μ_t) in URANS procedure or SGS eddy viscosity μ_{sgs} in LES procedure. Steps (2) to (6) thus complete one outer iteration loop.
7. Repeat steps (2) to (6) with updated pressure, velocities and scalar field till convergence between two consecutive time steps.
8. Advance by one time step ($t = t + \Delta t$), repeat steps (2) to (7) till the prescribed number of time steps are computed.

If we are solving steady state solution, then only steps 1-7 are to be followed.

4.4.13 Convergence criterion

The convergence of overall solution is declared when the maximum L2 norm of the residues over all computational nodes is less than a prescribed limit, the so-called convergence criterion. The residual R_ϕ^n at a node P for the variable after n iterations may be calculated as:

$$R_\phi^n = \sqrt{\sum_N \left(\sum_{nb} A_{nb} \phi_{nb} + SU - A_P \phi_P^{(n-1)} \right)^2}$$

The convergence criterion may be expressed as:

$$\max(R_\phi^n / RNORM) \leq CC$$

where N = total number of computational nodes, the suffix nb denotes the six neighbouring cell centres around the hexahedral computational cell; $RNORM$ = a reference value used for normalization, for example, the total inlet flux of the corresponding flow variable and CC = convergence criterion. Fixing $CC = 10^{-5}$ for this code in general was found to be sufficient to obtain the desired accuracy for all most all the flow simulations carried out.

4.5 Summary

In this chapter, finite volume method (FVM) formulation and governing equations used in the in-house pressure based incompressible flow solution code 3D-PURLES are discussed in detail. The spatial and temporal discretisation schemes, boundary conditions, multiblock computation and convergence criterion are also discussed in this chapter.

Chapter 5

Validation and Sensitivity Study

5.1 Background

This chapter discusses the validation and sensitivity study carried out for the low Re SD7003 airfoil at $Re = 6 \times 10^4$. The formation of laminar separation bubble (LSB) and flow undergoing transition is the major characteristics of this low Re airfoil which effects its aerodynamic performance. The transitional and aerodynamic behaviour predicted by the in-house flow code 3D-PULRES for this airfoil is validated against the available measurement data and other computational data. This validation study is carried out using the correlation based $\gamma - Re_\theta$ SST transition model available in 3D-PURLES at two angles of attack ($\alpha = 4^\circ$ and $\alpha = 8^\circ$) for which detailed experimental data are available. The accuracy of any flow solution code greatly depends on the quality of the grid generated and the flow solution methodology. This chapter also discusses the sensitivity study of the grid quality and flow solution control parameter on the accuracy of the flow characteristics predicted for the SD7003 airfoil. The grid quality parameters considered for this study are farfield distance, grid spacing near the leading and trailing edge of the airfoil, near-wall distance and different grid topologies. The flow control parameters which are more likely to effect the flow solution like the spatial discretisation schemes, freestream turbulence intensity and inlet eddy viscosity ratio are selected for this sensitivity study. The influence of the above grid and flow parameters on the aerodynamic characteristics in terms of lift coefficient (C_L), drag coefficient (C_D) & lift to drag ratio (L/D) and

the transition characteristics in terms of transition onset location (X_T/C) & length of laminar separation bubble (LSB) are analysed. The method used to estimate the transition onset location and length of the laminar separation bubble for present study is discussed in the following subsection.

5.2 Computational Details

All the present 2D simulations were carried out using a C-grid topology, the details of the grid generated is discussed in Chapter 3. In this study, spatial discretisation used for the convective flux of the momentum equations was QUICK (Leonard, 1979) scheme and upwind scheme for the convective flux of the turbulent scalar equations. The QUICK discretisation scheme was coupled to deferred correction procedure (Khosla and Rubin, 1974) *i.e.* combining 10% of Upwind fluxes and 90% of QUICK fluxes in order to have stability in the convergence. Whereas, the central difference discretisation scheme was used for diffusion (viscous) flux appearing in the momentum and turbulent scalar equations. The boundary conditions used for these simulations are shown in Fig. 5.1. The farfield boundary is treated either as an inflow (flow is prescribed) or as an outflow (zero gradient) depending on the sign of the convective flux on the relevant face. For the inflow condition, uniform velocity, turbulent viscosity ($\mu_t = \mu$), freestream turbulence intensity (Tu) and turbulence scalars ($k = k_T = 1.5(UT_u/100)^2$, $k_L = 0$, $\omega = C_\mu \rho k / \mu_t$, $\gamma = 1$ and $\tilde{Re}_{\theta t}$ is obtained from the empirical correlation (Eq. 4.52)) are prescribed. No-slip boundary condition for the velocities are applied at the airfoil boundary and the turbulent scalars are prescribed based on the model used as discussed in Section 4.4.8. The Neumann boundary conditions are applied for the pressure at all the boundaries. The convergence criteria for all the computations carried out in this work was fixed to be 10^{-5} for the continuity and momentum equations.

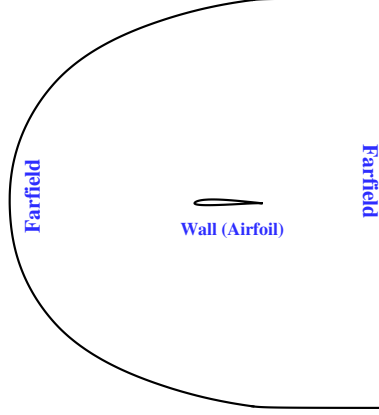


Figure 5.1: Boundary conditions used for the airfoil flow simulations

5.2.1 Estimation of Transition Parameters

Determination of transition onset location

In the present study the transition onset location is estimated by two different approaches one based surface pressure (C_p) distribution and second one based on Reynolds shear stress threshold criteria as suggested by Ol *et al.* (2005); Radespiel *et al.* (2006). The flow undergoing laminar separation, transition and reattachment will be reflected in the surface pressure distribution curve as shown in Fig. 5.2. The strong adverse pressure gradient causes the flow separation and the laminar separated region is denoted by the flattening of the pressure and this pressure plateau is terminated by a sudden decrease in the C_p which denotes the onset of transition. The pressure keeps decreases till the flow becomes fully turbulent and flow reattachment occurs at this point beyond which the change in C_p on the surface remains almost constant.

In the Reynolds shear stress threshold criteria the transition onset location is the point on the airfoil surface where a sudden increase in the normalized Reynolds (turbulent) shear stress (τ_{xz}/U_∞^2) is observed beyond 0.1%. The Reynolds shear stress (τ_{xz}) is computed based on Boussinesq's approximation as

$$\tau_{xz} = \mu_t \left(\frac{\partial U_x}{\partial Z} + \frac{\partial U_z}{\partial X} \right)$$

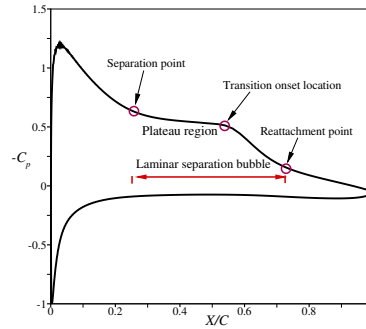


Figure 5.2: Determination of transition onset based on surface pressure distribution

Fig. 5.3 shows the normalized Reynolds shear stress profiles at different locations along with the contours on the airfoil surface. The sudden increase in the Reynolds shear stress profiles is marked as transition onset and the same is reflected as higher values in the contours. This criteria has been successfully used by other researcher's (Yuan *et al.*, 2005; Windte *et al.*, 2006; Lian and Shyy, 2007; Castonguay *et al.*, 2010; Shome and Radle, 2013) and in our earlier work (Manu *et al.*, 2014; Subrahmanya *et al.*, 2015; Kulkarni *et al.*, 2018) to estimate the transition onset location for airfoil at low Reynolds number.

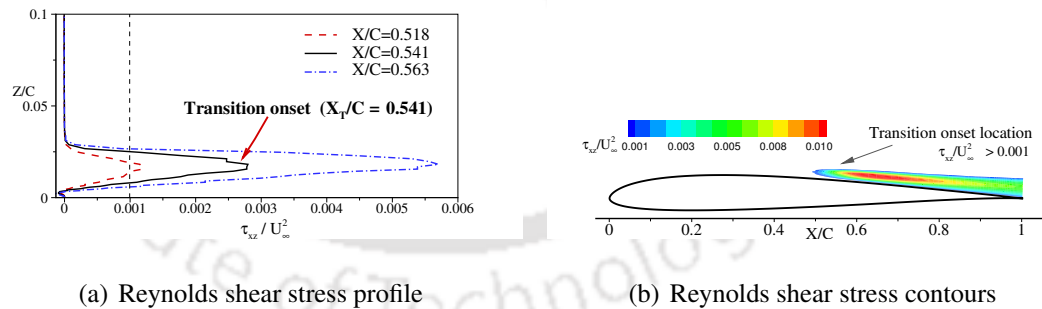


Figure 5.3: Determination of transition onset based on normalized Reynolds shear stress

Determination of length of laminar separation bubble

The length of the separation bubble is determined by the surface skin-friction (C_f) distribution and/or by surface streamlines. The change of sign from positive to negative in the surface skin-friction curve (Fig. 5.4(a)) indicates the separation point and at reattachment point the sign of C_f

reverses and the difference between the reattachment point and separation point gives the length of separated region as marked in the figure. The typical surface streamlines around an airfoil is shown in Fig. 5.4(b) where the streamlines detach from surface at the separation point and at the reattachment point the streamlines attach to airfoil surface again as indicated in the figure.

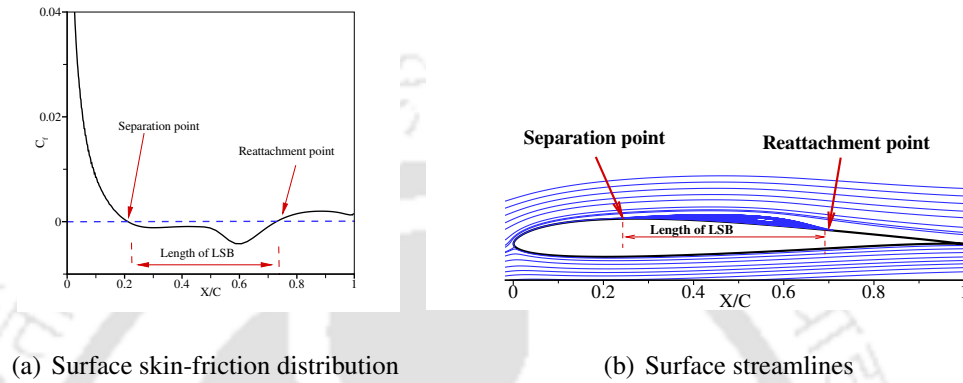


Figure 5.4: Determination of length of laminar separation bubble (LSB)

5.3 Validation Study

As a first step, validation study is carried out for SD7003 airfoil at two angle of attack ($\alpha = 4^\circ$ and $\alpha = 8^\circ$) for which measurement data for aerodynamic characteristics (Selig *et al.*, 1995) at $Re = 61400$ and transition characteristics (Radespiel *et al.*, 2006) at $Re = 60000$ are available in literature. The computed aerodynamic coefficients are compared with measurement data of Selig *et al.* (1995) and transition characteristics are compared with measurement data of Radespiel *et al.* (2006). The present computed results are also validated against the Implicit Large Eddy Simulation (ILES) data of Galbraith and Visbal (2008). The comparison of the aerodynamic and transition characteristics for $\alpha = 4^\circ$ and $\alpha = 8^\circ$ are tabulated in Table 5.1 and Table 5.2 respectively. The results clearly shows that computed lift coefficient matches reasonably well with the measurement data and also with the ILES computational data for both the angles of attack. However, the drag coefficient was grossly overpredicted when compared to the measurement data. This overprediction was observed even in the ILES computation of Galbraith and Visbal (2008). This difference in the drag coefficient predicted by the measurement and both the computations cannot be explained because of the lack

of the details of the experimental setup and the flow conditions used in experiments. Further Selig *et al.* (1995) experimental data is only available for the aerodynamic characteristics without any clarity regarding transition onset and flow separation. The transition onset location obtained by the present computation matches well with the both the measurement data of Radespiel *et al.* (2006) and ILES computational data of Galbraith and Visbal (2008) for both the angles of attack. On the other hand, the length of LSB computed by the present computation as well as that obtained by the ILES computations is grossly overpredicted when compared to the measurement data. This overprediction in the length of LSB by the present computation is due to the delay in the reattachment of the separated flow. This delay in the reattachment by present computation may be attributed to the lower turbulence level beyond the transition point in the separated flow region when compared to the measurement data as discussed in the next chapter. The enhanced drag coefficient obtained by the computations may be attributed to the longer laminar separation bubble predicted by the computations when compared to the experimental results. In the present computation refining the grid along the airfoil and in the boundary layer did not reduce the length of the laminar separation bubble as discussed in the next section.

Table 5.1: Comparison of flow characteristics for $\alpha = 4^\circ$

Methodology	C_L	C_D (Counts)	L/D	X_T/C	Length of LSB
Present computation	0.59	238	24.78	0.54	0.51
ILES (Galbraith and Visbal, 2008)	0.58	224	25.89	0.55	0.42
Measurement (Selig <i>et al.</i> , 1995)	0.60	163	36.80	NA	NA
Measurement (Radespiel <i>et al.</i> , 2006)	NA	NA	NA	0.55	0.32

Table 5.2: Comparison of flow characteristics for $\alpha = 8^\circ$

Methodology	C_L	C_D (Counts)	L/D	X_T/C	Length of LSB
Present computation	0.88	454	19.38	0.15	0.29
ILES (Galbraith and Visbal, 2008)	0.91	431	21.11	0.18	0.24
Measurement (Selig <i>et al.</i> , 1995)	0.93	299	31.10	NA	NA
Measurement (Radespiel <i>et al.</i> , 2006)	NA	NA	NA	0.14	0.12

5.4 Sensitivity Study for SD7003 Airfoil

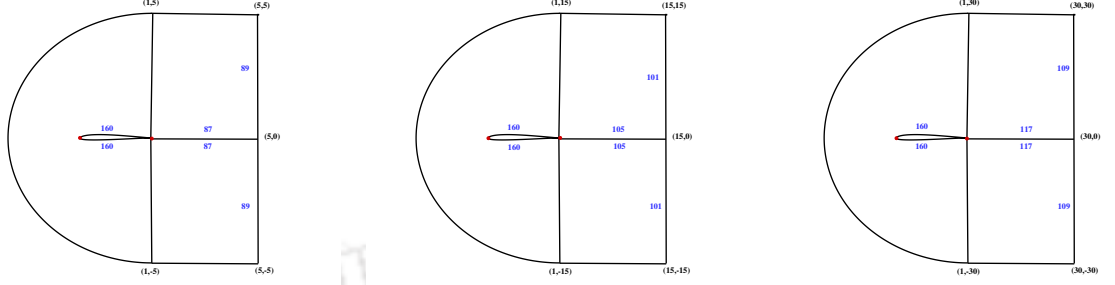
5.4.1 Influence of grid quality on flow solution accuracy

The major grid quality parameters considered for the present study which may influence accuracy of the flow solution are (i) Farfield boundary distance, (ii) Airfoil leading edge and trailing edge resolution, (iii) Near-wall resolution while maintaining the near-wall y_{nw}^+ to be less than one and (iv) Grid topology.

Influence of farfield boundary distance

The location of the farfield boundary plays an important role in deciding the accuracy of the numerical solution. The ideal condition is to assume the farfield boundary at a distance (R_{out}) where non-reflecting boundary conditions can be used. For the present study, three different farfield distances *viz.* $5C$, $15C$ and $30C$ have been used as shown in the Fig. 5.5. The three grids are obtained by keeping same number of grid points and growth rate along the airfoil while the number of grid points are increased in wall normal direction with increasing farfield distance in order to keep the same near-wall distance and growth rate. The results obtained in terms of flow characteristics (aerodynamic and transition) of the airfoil are tabulated in Table 5.3. The table clearly indicates the solution obtained for different farfield distance are found to be quite insensitive to the location of the farfield boundary beyond the farfield distance of $15C$. Therefore, for the present study all the

flow computations have been carried out using the farfield boundary distance, $R_{out} = 15C$.



(a) $R_{out} = 5C$, Grid size: 491×89 (b) $R_{out} = 15C$, Grid size: 527×101 (c) $R_{out} = 30C$, Grid size: 551×109

Figure 5.5: Schematic diagrams showing farfield distance (R_{out}) for SD7003 airfoil

Influence of airfoil leading edge and trailing edge resolution

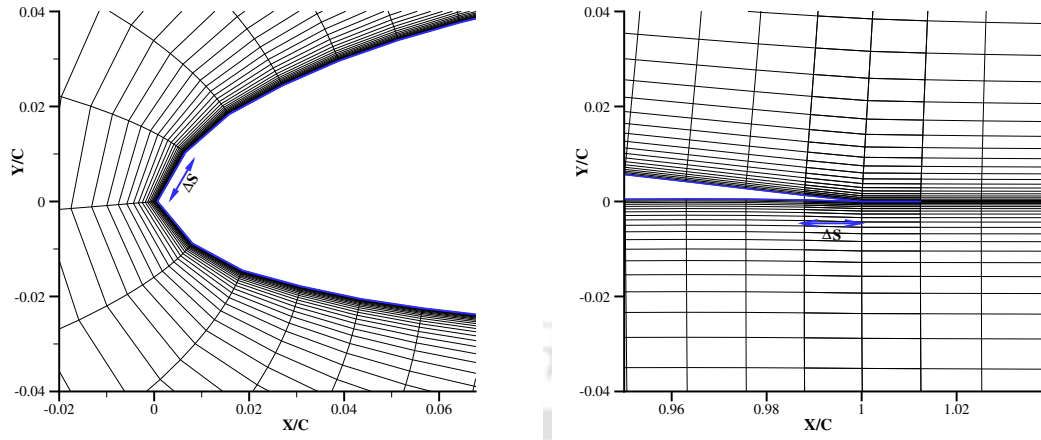
The accuracy of the numerical flow solution around the airfoil generally depends on the grid resolution (ΔS) near the leading edge (LE) and trailing edge (TE) areas where the flow gradients are expected to be higher. In order to resolve the LE and TE properly, grids with three different grid spacing (ΔS) viz. $1.2 \times 10^{-2}C$, $2.4 \times 10^{-3}C$ and $3.5 \times 10^{-4}C$ are obtained by keeping same growth rate but increasing the number of grid points along the airfoil surface as shown in the Fig. 5.6. The number of grid points and the near-wall distance in the normal direction is maintained same for all the three grids. The computed airfoil flow characteristics (aerodynamic and transition) for the three grids are given in Table 5.4. It is evident from this table that one needs to have the leading and trailing edge resolution of the airfoil to be at least around $\Delta S = 2.4 \times 10^{-3}C$ and it is also found that the solution is insensitive to further refinement. Based on this study, the LE and TE resolution of the airfoil is fixed at $\Delta S = 2.4 \times 10^{-3}C$ for all the other computations.

Table 5.3: Influence of farfield distance on flow characteristics

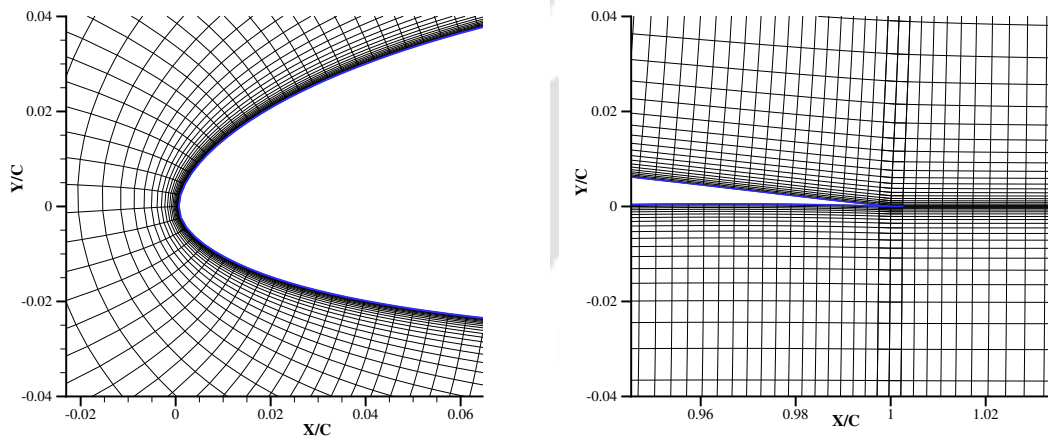
α	R_{out}	Grid Size	C_L	C_D (Counts)	L/D	X_T/C	Length of LSB
2°	$5C$	491×89	0.34	206	16.50	0.80	0.62
	$15C$	527×101	0.35	204	17.15	0.79	0.62
	$30C$	551×109	0.35	203	17.24	0.79	0.62
4°	$5C$	491×89	0.58	246	23.57	0.54	0.55
	$15C$	527×101	0.59	238	24.78	0.54	0.51
	$30C$	551×109	0.59	236	25.00	0.53	0.51
8°	$5C$	491×89	0.88	466	18.88	0.16	0.34
	$15C$	527×101	0.88	454	19.38	0.15	0.29
	$30C$	551×109	0.88	454	19.38	0.15	0.29

Table 5.4: Influence of airfoil leading & trailing edge resolution on flow characteristics

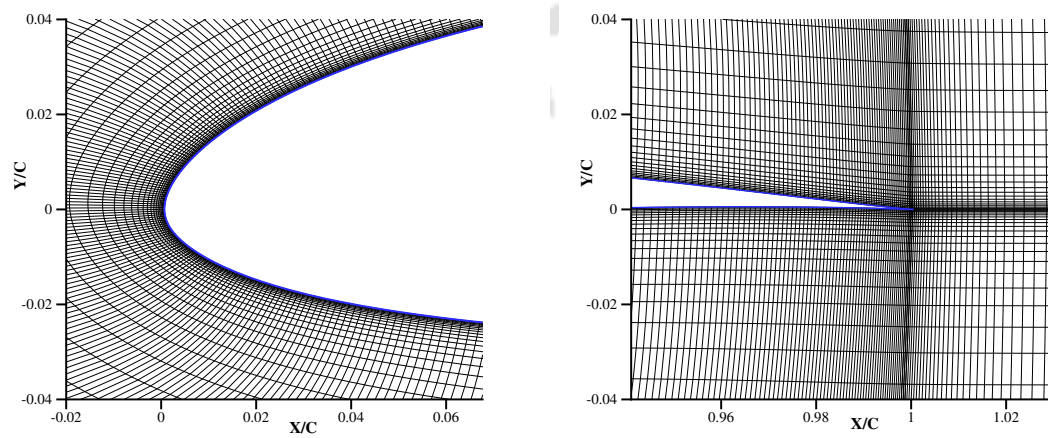
α	LE and TE Resolution	Grid Size	C_L	C_D (Counts)	L/D	X_T/C	Length of LSB
2°	$1.2 \times 10^{-2}C$	329×101	0.38	187	20.32	0.78	0.62
	$2.4 \times 10^{-3}C$	527×101	0.35	204	17.15	0.79	0.62
	$3.5 \times 10^{-4}C$	849×101	0.35	204	17.15	0.79	0.63
4°	$1.2 \times 10^{-2}C$	329×101	0.58	235	24.68	0.48	0.52
	$2.4 \times 10^{-3}C$	527×101	0.59	238	24.78	0.54	0.51
	$3.5 \times 10^{-4}C$	849×101	0.59	238	24.78	0.54	0.51
8°	$1.2 \times 10^{-2}C$	329×101	0.80	633	12.63	0.16	0.96
	$2.4 \times 10^{-3}C$	527×101	0.88	454	19.38	0.15	0.29
	$3.5 \times 10^{-4}C$	849×101	0.88	462	19.04	0.15	0.29



(a) $\Delta S = 1.2 \times 10^{-2}C$, Grid size: 329×101



(b) $\Delta S = 2.4 \times 10^{-3}C$, Grid size: 527×101



(c) $\Delta S = 3.5 \times 10^{-4}C$, Grid size: 849×101

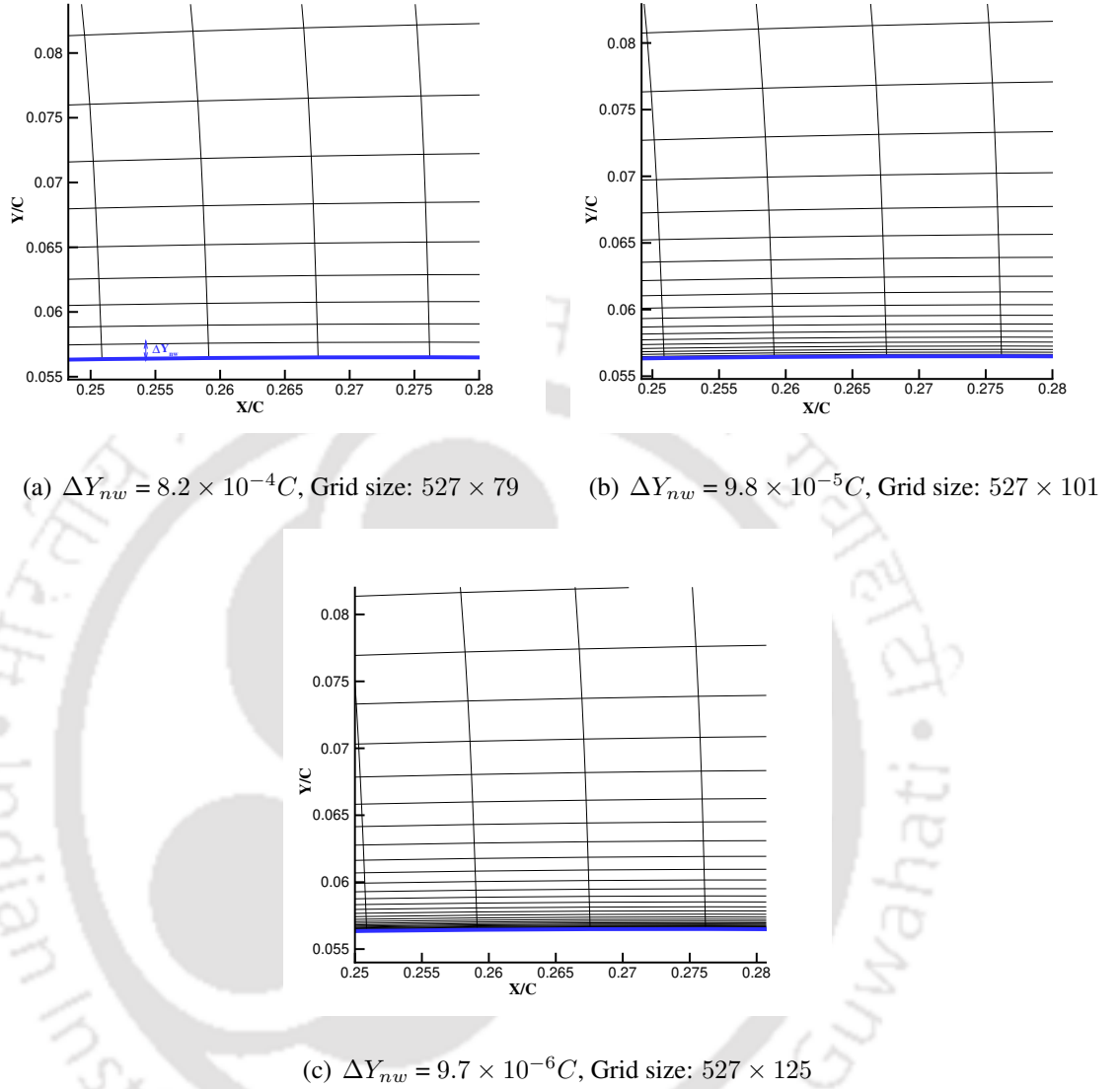
Figure 5.6: Leading edge and trailing edge grid resolution (ΔS) for SD7003 airfoil

Influence of airfoil near-wall resolution

It is well known that boundary layer resolution plays an important role on the accuracy of the numerical flow results. In order to resolve the airfoil near-wall spacing adequately so as to have more grid points in the boundary layer and maintain a $y_{nw}^+ < 1$, grids with three different near-wall grid distances (ΔY_{nw}) viz. $8.2 \times 10^{-4}C$, $9.8 \times 10^{-5}C$ and $9.7 \times 10^{-6}C$ are obtained by keeping same growth rate but increasing the number of grid points along the wall normal direction as shown in the Fig. 5.7. Table 5.5 compares the flow characteristics obtained by the three different near-wall resolution. This comparison clearly shows that the near-wall resolution of $8.2 \times 10^{-4}C$ with 79 points in wall normal direction appears to be sufficient for the angles of attack analysed. However to be on the safer side $\Delta Y_{nw} = 9.8 \times 10^{-5}C$ with 101 grid points in the wall normal direction has been maintained for all the other flow simulations.

Table 5.5: Influence of airfoil near-wall resolution on flow characteristics

α	Near-wall Resolution	Grid Size	C_L	C_D (Counts)	L/D	X_T/C	Length of LSB
2°	$8.2 \times 10^{-4}C$	527×79	0.35	206	16.99	0.79	0.63
	$9.8 \times 10^{-5}C$	527×101	0.35	204	17.15	0.79	0.62
	$9.7 \times 10^{-6}C$	527×125	0.35	204	17.15	0.79	0.62
4°	$8.2 \times 10^{-4}C$	527×79	0.58	239	24.26	0.53	0.56
	$9.8 \times 10^{-5}C$	527×101	0.59	238	24.78	0.54	0.51
	$9.7 \times 10^{-6}C$	527×125	0.59	238	24.78	0.54	0.51
8°	$8.2 \times 10^{-4}C$	527×79	0.87	474	18.35	0.16	0.39
	$9.8 \times 10^{-5}C$	527×101	0.88	454	19.38	0.15	0.29
	$9.7 \times 10^{-6}C$	527×125	0.88	454	19.38	0.15	0.29

Figure 5.7: Influence of grid near-wall distance (ΔY_{nw}) for SD7003 airfoil

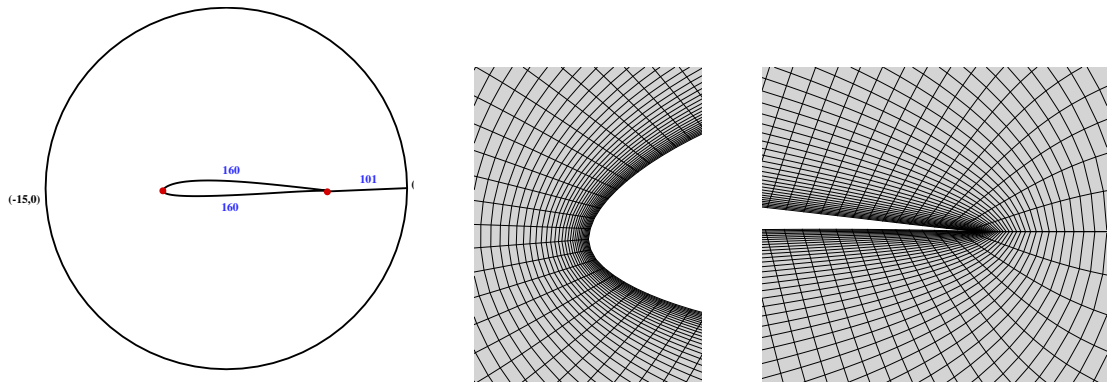
Influence of grid topology

In this section, effect of three different grid topologies (O-grid, C-grid and H-grid) on the accuracy of the airfoil flow solution is analysed. The grid topology nomenclature is based on their resembles with the English alphabets. The in-house grid generation code is used to generate the three different grid topology around SD7003 airfoil as shown in Fig. 5.8. In this figure the zoomed view of the grid near the leading and trailing edge obtained by three different grid topologies are also shown. The grid resolution on the airfoil surface ($\Delta S = 2.4 \times 10^{-3} C$ with 320 points) and near-wall resolution

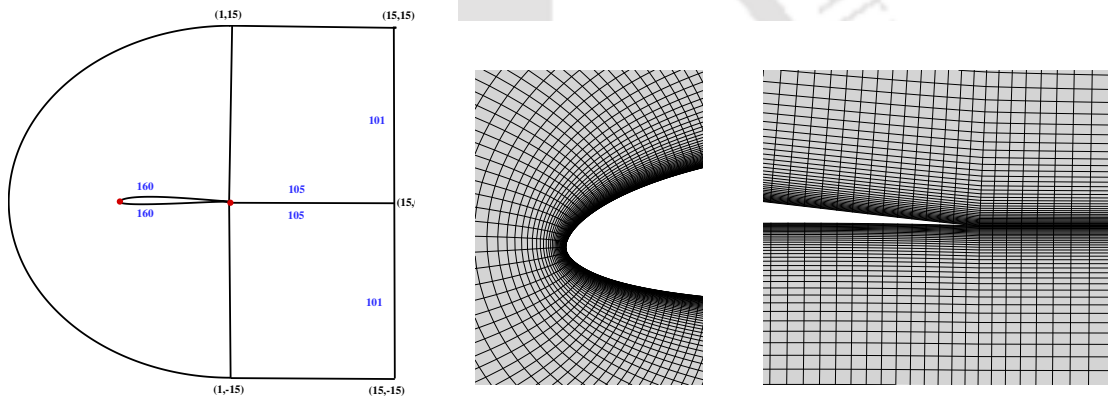
($\Delta Y_{nw} = 9.8 \times 10^{-5}C$ with 101 points) are maintained to be same for all the three grid topologies. The airfoil aerodynamic and transition characteristics obtained from the three grid topologies are compared and shown in Table 5.6. The table clearly indicates the 3D-PURLES flow solution is independent of the grid topology. The O-grid which has advantage of lesser grid points leading to lesser computational time but has major drawback of O-grid topology is that the airfoil trailing edge and wake region can not resolved very well. This topology has highly skewed and wedge type cells near the trailing edge and the wake region (Fig. 5.8(a)) which may sometimes hamper the convergence of the flow solution especially for very fine near-wall resolution. Hence O-grid is generally not preferred for airfoil type of geometries. On the other hand, for both C-grid and H-grid the grid lines are well aligned at the trailing edge and wake region. The control of the resolution of the boundary layer along with wake region is better for these topologies when compared to the O-grid topology. The C-grid topology is preferred and used for all our present simulations because the number grid nodes are much lesser than the H-grid topology and moreover for H-grid topology the grid is distorted near the leading edge of the airfoil (Fig. 5.8(c)) which is likely to reduce the accuracy of the lift coefficient.

Table 5.6: Influence of grid topology on flow characteristics

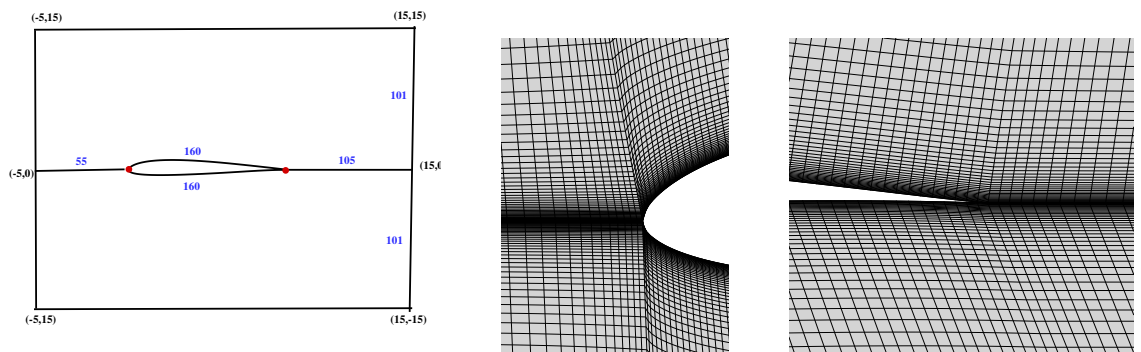
α	Grid topology	Grid Size	C_L	C_D (Counts)	L/D	X_T/C	Length of LSB
2°	O-Type	319×101	0.37	206	17.96	0.81	0.63
	C-Type	527×101	0.35	204	17.15	0.79	0.62
	H-Type	638×202	0.34	210	16.19	0.81	0.61
4°	O-Type	319×101	0.59	243	24.27	0.52	0.55
	C-Type	527×101	0.59	238	24.78	0.54	0.51
	H-Type	638×202	0.57	252	22.61	0.54	0.55
8°	O-Type	319×101	0.89	467	19.05	0.15	0.31
	C-Type	527×101	0.88	454	19.38	0.15	0.29
	H-Type	638×202	0.86	506	16.99	0.16	0.37



(a) O-grid, Grid size: 319×101



(b) C-grid, Grid size: 527×101



(c) H-grid, Grid size: 638×202

Figure 5.8: Schematic diagrams and zoomed view near leading and trailing edge of the SD7003 airfoil for different grid topology

5.4.2 Influence of flow solution control parameters on solution accuracy

The major flow solution control parameters considered in this study are (i) Spatial discretisation scheme (ii) Freestream turbulence intensity and (iii) Inlet eddy viscosity ratio.

Influence of spatial discretisation schemes

The accuracy of any solution algorithm depends to a large extent on the discretisation schemes used for numerical simulation of the differential operators in the relevant equations which often involve both temporal (with respect to time) and spatial (with respect to the coordinate directions in space) derivatives. In the finite volume method used in 3D-PURLES, the spatial derivatives are transformed to fluxes across the cell faces and the discretisation scheme basically decides the interpolation to be used for evaluation of cell face value of the variable or its gradients from the nodal values at either side of the face. The diffusive flux are discretised using the central difference scheme and the accuracy of the present numerical solution depends on the scheme used to discretise the convective fluxes. The major task is how to evaluate the value of a flow variable at any cell face lying between two consecutive cell centres. The three different flux discretisation schemes used in the present study are discussed in Section 4.4.4.

Table 5.7 shows the influence of three different spatial discretisation schemes on the airfoil flow characteristics in terms of aerodynamic & transition characteristics. As expected, the pure upwind scheme which is associated with large numerical diffusion has underpredicted lift coefficient and overpredicted the drag coefficient leading to a lower L/D ratio. However the other two schemes (QUICK and CDS) predicted almost similar results with QUICK being a third order accurate scheme is used for all the further computations.

Influence of freestream turbulence intensity

The freestream turbulence intensity (Tu) of 0.1% has been used for all the computations carried out at low Re using 3D-PULRES. This Tu is fixed based on the value used in experiments by various researchers (Radespiel *et al.*, 2006; Ol *et al.*, 2005) for conducting experiments on low Re

Table 5.7: Influence of spatial discretisation scheme on flow characteristics

α	discretisation scheme	Grid Size	C_L	C_D (Counts)	L/D	X_T/C	Length of LSB
2°	Upwind	527×101	0.24	220	10.90	0.93	0.55
	QUICK	527×101	0.35	204	17.15	0.79	0.62
	CDS	527×101	0.36	201	17.91	0.79	0.62
4°	Upwind	527×101	0.55	309	17.79	0.60	0.75
	QUICK	527×101	0.59	238	24.78	0.54	0.51
	CDS	527×101	0.59	236	25.00	0.54	0.51
8°	Upwind	527×101	0.78	711	10.97	0.17	0.94
	QUICK	527×101	0.88	454	19.38	0.15	0.29
	CDS	527×101	0.88	453	19.42	0.15	0.29

airfoils. In this section, numerical investigations are carried out in order to understand the effects of freestream turbulence intensity (Tu) on the aerodynamic and transition characteristics of low Re SD7003 airfoil. For this study, two different Tu values are considered by decreasing and increasing the the level of freestream turbulence intensity by a decade with respect to 0.1%. Table 5.8 shows the influence of freestream turbulence intensity on the airfoil flow characteristics in terms of aerodynamic & transition characteristics. The table clearly indicates that the only lowering the freestream turbulence intensity below 0.1% is found to influence mainly the aerodynamic characteristics. As expected, increasing the Tu beyond 0.1% is observed to have no significant influence on airfoil flow parameters. The detailed studied of the effect of freestream turbulence intensity for range of angles of attack at two different Reynolds number are discussed in Appendix A.

Influence of inlet eddy viscosity ratio

The eddy viscosity ratio (μ_t/μ) is the ratio between the turbulent viscosity (μ_t) and the laminar viscosity (μ). Eddy viscosity ratio is often also called as turbulent viscosity ratio or simply viscosity

Table 5.8: Influence of freestream turbulence intensity on flow characteristics

α	Tu %	Grid Size	C_L	C_D (Counts)	L/D	X_T/C	Length of LSB
2°	0.01	527×101	0.30	211	14.21	0.83	0.63
	0.1	527×101	0.35	204	17.15	0.79	0.62
	1.0	527×101	0.36	196	18.36	0.79	0.62
4°	0.01	527×101	0.58	257	22.56	0.54	0.63
	0.1	527×101	0.59	238	24.78	0.54	0.51
	1.0	527×101	0.59	228	25.87	0.54	0.51
8°	0.01	527×101	0.81	633	12.79	0.16	0.47
	0.1	527×101	0.88	454	19.38	0.15	0.29
	1.0	527×101	0.88	435	20.22	0.15	0.29

ratio. This ratio is used to prescribe the turbulent dissipation rate (ϵ) and the turbulent specific dissipation (ω) at the inlet as follows

$$\epsilon_{in} = \frac{\rho C_\mu k_{in}^2}{\mu} \times \left(\frac{\mu_t}{\mu}\right)^{-1} \quad (5.1)$$

$$\omega_{in} = \frac{\rho C_\mu k_{in}}{\mu} \times \left(\frac{\mu_t}{\mu}\right)^{-1} \quad (5.2)$$

where $k_{in} = 1.5 \times U_{in} \times Tu_{in}$ and C_μ is the turbulence model constant. The eddy viscosity ratio gives the information about the strength and influence of the turbulent viscosity as compared to the laminar viscosity. The inlet eddy viscosity ratio of 1 is generally fixed for most of the simulations carried out for the various geometries using the in-house code 3D-PURLES. The present exercise is to study the effect of inlet eddy viscosity ratio on the airfoil flow characteristics, when eddy viscosity ratio is decreased and increased by a decade (0.1 and 10). The results in terms of aerodynamic and transition characteristics obtained by the three inlet eddy viscosity ratio are tabulated in Table 5.9. The table clearly indicates that the aerodynamic and transition characteristics are insensitive to the

inlet eddy viscosity ratio thus justifying our assumption of inlet $\mu_t/\mu = 1$.

Table 5.9: Influence of inlet eddy viscosity ratio on flow characteristics

α	μ_t/μ	Grid Size	C_L	C_D (Counts)	L/D	X_T/C	Length of LSB
2°	0.1	527×101	0.35	204	17.15	0.79	0.62
	1.0	527×101	0.35	204	17.15	0.79	0.62
	10.0	527×101	0.35	204	17.15	0.79	0.62
4°	0.1	527×101	0.59	239	24.68	0.53	0.55
	1.0	527×101	0.59	238	24.78	0.54	0.51
	10.0	527×101	0.59	238	24.78	0.54	0.51
8°	0.1	527×101	0.88	461	19.08	0.15	0.34
	1.0	527×101	0.88	454	19.38	0.15	0.29
	10.0	527×101	0.88	454	19.38	0.15	0.29

5.5 Summary

The validation study carried out indicated that the lift coefficient and the transition onset obtained by the present computation matched well with both the experimental data and ILES computational data. On the other hand, when compared to the experiments, the length of the LSB obtained by both the computations were observed to be grossly over predicted which has significantly enhanced the drag coefficient. The influence of different grid quality parameters and flow solution control parameters on the the accuracy in predicting the flow characteristics of SD7003 airfoil at $Re = 6 \times 10^4$ using 3D-PURLES were studied. The parameters considered in the grid generation procedure which are likely to influence the flow solution accuracy are namely farfield distance, grid spacing near the leading and trailing edge of the airfoil, near-wall distance and different grid topologies. Whereas for the flow solution procedure, the different parameters such as spatial discretisation schemes, freestream turbulence intensity and inlet eddy viscosity ratio were considered to study their influence on the

accuracy of flow solution. Based upon this study, C-grid with farfield placed at a distance of $15C$ having a leading and trailing edge grid resolution of $(\Delta S) = 2.4 \times 10^{-3}C$ and grid near-wall distance of $\Delta Y_{nw} = 9.8 \times 10^{-5}C$ were found to be good enough for this low Reynolds number. Whereas for the flow solution code using third order accurate QUICK spatial discretisation scheme with freestream turbulent intensity of 0.1% and inlet eddy viscosity ratio of one was found to be sufficient to obtain reasonable accurate solution for SD7003 airfoil at $Re = 6 \times 10^4$. This study will help us in fixing the above parameters for all our future computations which will be carried out for low Re airfoils operating in the low Re flow regime.



Chapter 6

Flow Analysis of SD7003 Airfoil

6.1 Background

The performance of the two RANS based transition models namely $k_T - k_L - \omega$ and $\gamma - Re_\theta$ available in 3D-PURLES are also analysed and discussed for the SD7003 airfoil in this chapter. The performance of the transition models are assessed by comparing the aerodynamic and transition characteristics with both experiments (Selig *et al.*, 1995; Radespiel *et al.*, 2006) and ILES computation (Galbraith and Visbal, 2008). This chapter also discusses the effect of Reynolds number in the low Re regime on the performance of this airfoil.

6.2 Performance of the Transition Model

This section discusses the performance of the $k_T - k_L - \omega$ and $\gamma - Re_\theta$ SST transition models on the flow, aerodynamic and transition characteristics of SD7003 airfoil at $Re = 6 \times 10^4$.

The surface pressure distribution coefficient (C_p) and surface skin-friction coefficient (C_f) predicted by two transition models at $\alpha = 4^\circ$ and $\alpha = 8^\circ$ are compared with the ILES data of Galbraith and Visbal (2008) as shown in Fig. 6.1 and a fairly good match is obtained. As discussed in the previous section, the computed surface pressure curve obtained by the two transition models at both the angles of attack clearly indicate the formation of the LSB and onset of transition which is quite

similar to that of the ILES predictions. The transition onset, transition region and the length of the LSB obtained by the $\gamma - Re_\theta$ SST is found to be in a very good agreement with the ILES data. On the other hand, the $k_T - k_L - \omega$ has predicted a delayed transition onset and overpredicted the length of LSB as well as the length of the transition zone. The transition onset location (X_T/C) estimated based on the C_p distribution curve for the $\gamma - Re_\theta$ model is 0.554 for $\alpha = 4^\circ$ and 0.179 for $\alpha = 8^\circ$ which is in close agreement with measurement data and other computation results (Table 6.1 and Table 6.2). The transition onset location obtained by $k_T - k_L - \omega$ model based on the C_p curve for $\alpha = 4^\circ$ and $\alpha = 8^\circ$ is $X_T/C = 0.670$ and $X_T/C = 0.243$ respectively which clearly indicates that this model has overpredicted the transition onset location. The C_f plot at these two angles of attack clearly indicates that both the models have predicted the separation point (indicated by change of sign from positive to negative) little upstream and the reattachment (indicated by change of sign from negative to positive) quite downstream leading to an increase in the length of the LSB when compared to the ILES data with the difference being more enhanced for $k_T - k_L - \omega$ transition model. However, the maximum bubble height predicted by the $\gamma - Re_\theta$ SST model closely matches with the ILES results with the $k_T - k_L - \omega$ transition model grossly overpredicting the height. The separation point obtained the $\gamma - Re_\theta$ SST transition model for both the angles is found to be in a better agreement with other LES and DNS computations as shown in Table 6.1 and Table 6.2. It can be further noted that all the computations in general have underpredicted the separation when compared to the measurement data. The overprediction of the reattachment point by the present computation may be attributed to the inability of the transition model to rapidly produce enough turbulence kinetic energy in the shear layer to induce flow reattachment. A similar delay in the reattachment point was reported by other researchers using the transition models (Rival *et al.*, 2010; Shome and Radle, 2013)

The normalized Reynolds shear stress contours and streamlines obtained by the present computation using the two transition models are compared with the PIV measurements of Radespiel *et al.* (2006) as shown in Fig. 6.2 and Fig. 6.4 for the two angles. The computed normal profiles of the Reynolds shear stress (τ_{xz}) obtained at different locations on the airfoil surface by the transition models for the two angles of attack are also shown in Fig. 6.3 and Fig. 6.5. The transition onset location shown in these figures is determined based on Reynolds shear stress threshold criterion as

discussed in the previous section. It is quite obvious from the contours and normal profiles of the turbulent shear stress that the $k_T - k_L - \omega$ model has underpredicted the transition onset location at both the angles of attack which is contrary to that obtained by C_p curve where the transition onset is overpredicted. On the other hand, the transition onset location predicted by $\gamma - Re_\theta$ SST model using the shear stress threshold criteria and C_p curve is identical and found to have a better match with measurement and other computations as indicated in Table 6.1 and Table 6.2.

The variation with angle attack for the transition onset location determined based on Reynolds shear stress threshold criterion and the features of the LSB in terms of separation point, reattachment point and its length are shown in Fig. 6.6. Similar to the ILES results, it is observed that the transition onset location (Fig. 6.6(a)) and the separation point (Fig. 6.6(b)) predicted by the present computation shifts upstream with increase in angle of attack with $\gamma - Re_\theta$ SST model having a closer match and $k_T - k_L - \omega$ model underpredicting both these quantities at all angles of attack. The transition onset (Fig. 6.6(a)) predicted by $\gamma - Re_\theta$ model is found to have good agreement with measurement data of Radespiel *et al.* (2006) as well. Further it may be noted that the transition to turbulent flow is predicted in the wake of the airfoil by $k_T - k_L - \omega$ model and $\gamma - Re_\theta$ SST model for $\alpha < 4^\circ$ and $\alpha < 2^\circ$ respectively indicating the flow to be fully laminar on the airfoil surface for these angles of attack. The separation location predicted by the $\gamma - Re_\theta$ SST model is found to be in a good agreement with the ILES data for all angles of attack but both these computations have predicted the location to be upstream of the airfoil when compared to the measurement data of Radespiel *et al.* (2006). The $k_T - k_L - \omega$ model on the other hand, has predicted the separation point further upstream resulting in a larger discrepancy with the measurement data. The reattachment point (Fig. 6.6(c)) obtained by the ILES data was found to move upstream leading to continuous decrease in the length of LSB (Fig. 6.6(d)) with increases in angle of attack. On the other hand, the reattachment point obtained by present computation using the two transition models is found to move upstream only upto $\alpha = 8^\circ$ and beyond this angle of attack the reattachment point moves downstream. This shift in the reattachment point downstream beyond $\alpha = 8^\circ$ is because the length of separation bubble is found to increase beyond this angle which is clearly depicted by the streamline plot as shown in Fig. 6.7. The increase in the size of the separation bubble may be because it is close to the stall angle particularly for the $\gamma - Re_\theta$ SST model. Both the transition models have predicted a delay in

the reattachment point leading to a longer LSB when compared to ILES data at all angles of attack with $k_T - k_L - \omega$ model predicting a larger bubble. The delay in the reattachment obtained by the present computations as mentioned earlier may be due to the lower level of turbulence predicted by these transition models and their inability to rapidly generate the turbulence in the shear layer (Fig. 6.2 and Fig. 6.4) beyond the transition point when compared to the measurements (Radespiel *et al.*, 2006). Similar to the variation of the reattachment point with angle of attack, the length of the LSB obtained by the two transition models is also found to increase beyond $\alpha = 8^\circ$. The streamline plots at different angles of attack as shown in Fig. 6.7 clearly indicate that the laminar separation pattern obtained by the two transition models is quite similar till $\alpha = 8^\circ$ with $k_T - k_L - \omega$ model predicting a longer separation bubble. A large separation bubble which is leaving the airfoil surface at its trailing edge is predicted by $\gamma - Re_\theta$ SST model at $\alpha = 10^\circ$ and beyond which the typical flow pattern is obtained over the airfoil beyond the stall angle. The flow pattern obtained by the $k_T - k_L - \omega$ model which is found to differ from $\gamma - Re_\theta$ SST model beyond $\alpha = 8^\circ$ shows the formation of a large bubble attached to the airfoil from $\alpha = 10^\circ$ till $\alpha = 16^\circ$. The length of this separation bubble for $\alpha \geq 10^\circ$ is found remains constant which is also indicted in Fig. 6.6(d) but the height of the bubble increases with increase in angle of attack resulting in a continuous increase in the lift coefficient and due to this the stall angle was not predicted by $k_T - k_L - \omega$ model. The inability of the $k_T - k_L - \omega$ model to predict stall may be due to the fact that this model can not handle separated flow transition in the present formulation as reported by Turner (2012).

The variation of the lift coefficient (C_L) and drag coefficient (C_D) obtained by the two transition models are compared with ILES data (Galbraith and Visbal, 2008) and measurement data of Selig *et al.* (1995) and is shown in Fig. 6.8. The lift coefficient obtained by the two transition models is found to match quite well with ILES data and measurement data upto $\alpha = 8^\circ$ beyond which some discrepancies are observed. This discrepancies in $\gamma - Re_\theta$ SST is because this model predicted an early stall angle (9°) when compared to ILES data ($\cong 11^\circ$). Since the measurement data of Selig *et al.* (1995) is available only upto $\alpha = 10^\circ$ with no information of the stall angle. However, it is worthwhile to note that Ol *et al.* (2005) in their paper on the measurements carried out on SD7003 airfoil at $Re = 6 \times 10^4$ in three facilities mention that the maximum C_L was obtained at $\alpha = 11^\circ$ in their AFRL water tunnel. The measurements of Rival and Tropea (2010) carried out in the Eiffel

wind tunnel, Technical University of Darmstadt for this airfoil and Reynolds number also indicate an early stall around $\alpha = 8^\circ$. The early stall prediction by $\gamma - Re_\theta$ SST model may be attributed to the longer LSB obtained by this model. Longer LSB in general reduces the lift and increases drag leading to an early stall. On the other hand, for the $k_T - k_L - \omega$ model has not predicted the stall angle till $\alpha = 16^\circ$ as explained earlier. The computations in general have overpredicted the variation of the drag coefficient, C_D (Fig. 6.8(b)) when compared to the measurement data of Selig *et al.* (1995) with $k_T - k_L - \omega$ model having the maximum difference. This higher drag obtained by $k_T - k_L - \omega$ model was expected as this model has predicted a much longer LSB. However, the variation of C_D obtained by the $\gamma - Re_\theta$ SST model and ILES (Galbraith and Visbal, 2008) is almost similar.

Table 6.1: Comparison of separation, reattachment and transition onset location at $\alpha = 4^\circ$

Data set	Type	$Tu\%$	X_S/C	X_T/C	X_R/C
IAR (Ol <i>et al.</i> , 2005)	Experiment	0	0.33	0.57	0.63
TU-BS (Ol <i>et al.</i> , 2005)	Experiment	0.1	0.30	0.53	0.62
AFRL-HFWT (Ol <i>et al.</i> , 2005)	Experiment	≈ 0.1	0.18	0.47	0.58
Radespiel <i>et al.</i> (2006)	Experiment	0.08	0.30	0.55	0.62
CartondeWiert and Hillewaert (2012)	DNS	NA	0.209	NA	0.654
CartondeWiert and Hillewaert (2012)	ILES	NA	0.207	NA	0.647
Galbraith and Visbal (2008)	ILES	NA	0.23	0.55	0.65
Uranga <i>et al.</i> (2009)	ILES	0	0.24	0.51	0.60
Castonguay <i>et al.</i> (2010)	ILES	0	0.23	0.52	0.65
Zhou and Wang (2010)	ILES	NA	0.227	0.521	0.685
Boom and Zingg (2013)	ILES	NA	0.21	0.51	0.64
Yuan <i>et al.</i> (2005)	SGS-LES	0	0.25	0.49	0.60
Present computation	$k_T - k_L - \omega$ model	0.1	0.172	0.477	0.84
Present computation	$\gamma - Re_\theta$ SST model	0.1	0.218	0.541	0.73

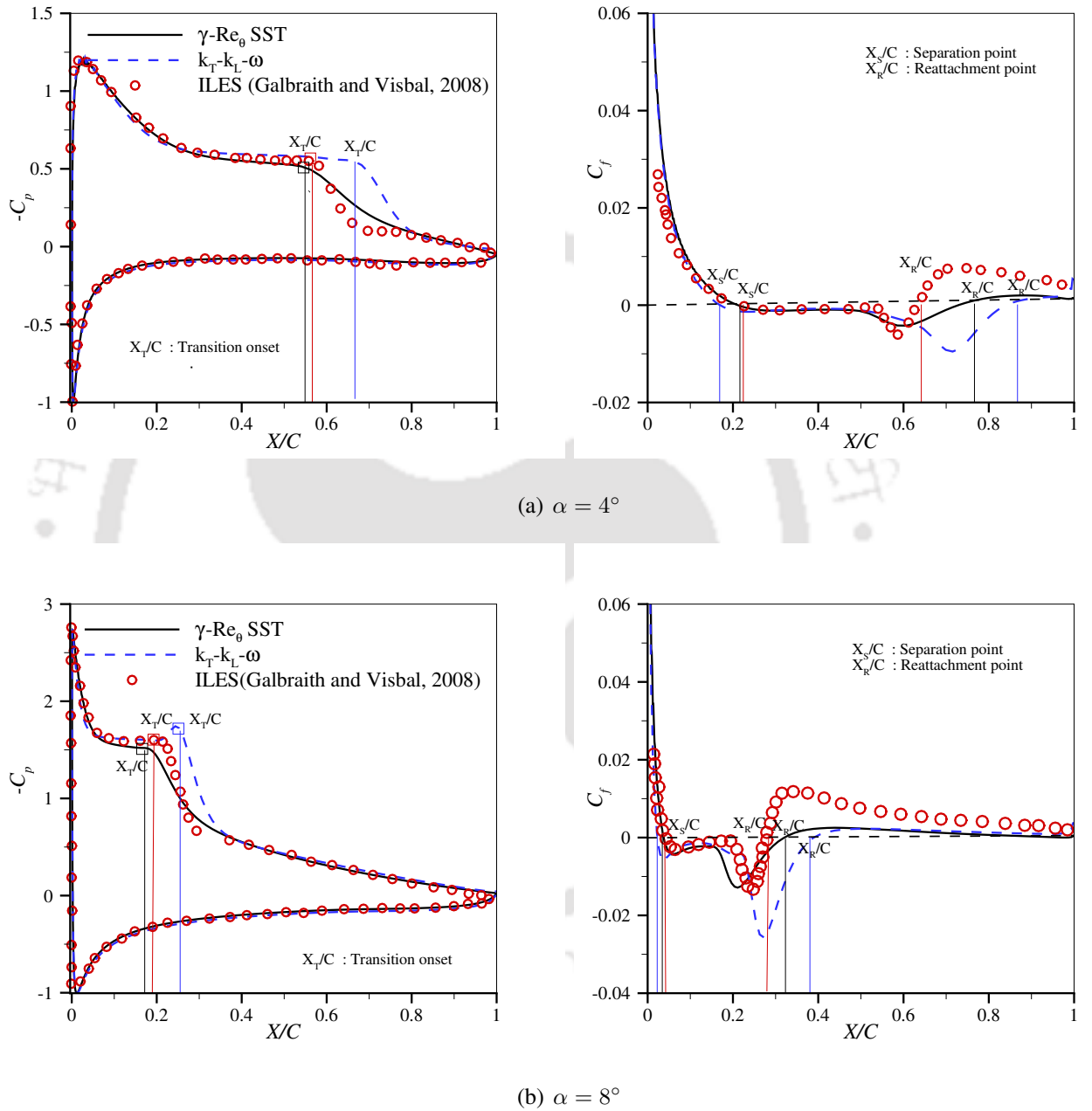


Figure 6.1: Effect of transition models on the variation of C_p and C_f at different α
(Left figure : Surface pressure coefficient, Right figure : Skin friction coefficient)

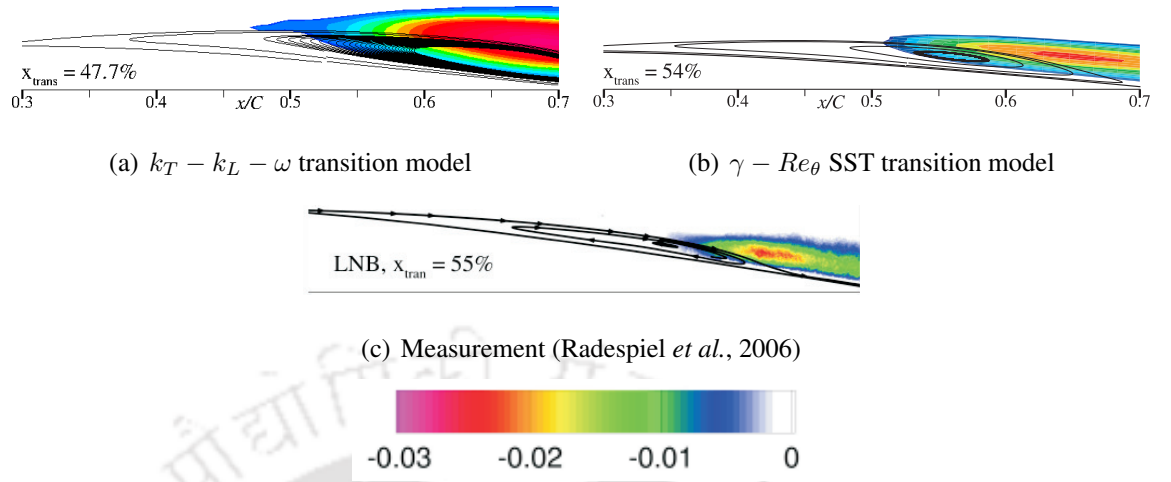


Figure 6.2: Comparison of normalized Reynolds shear stress contours at $\alpha = 4^\circ$

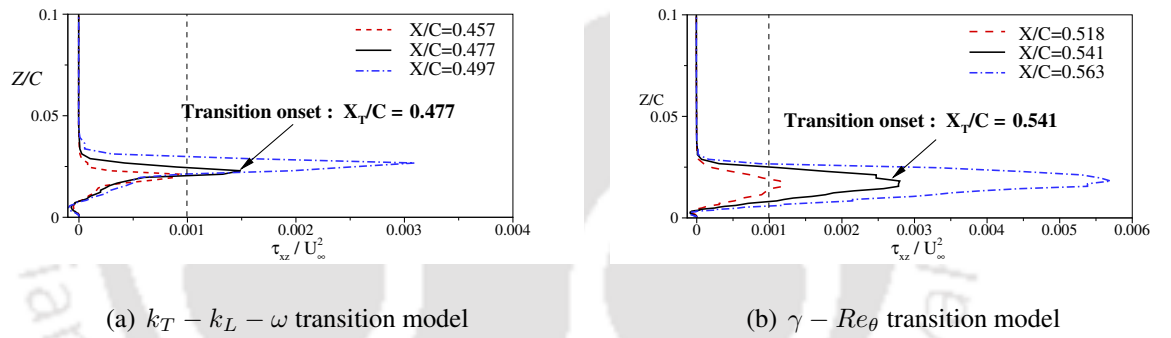


Figure 6.3: Normalized Reynolds shear stress profiles at $\alpha = 4^\circ$

Table 6.2: Comparison of separation, reattachment and transition onset location at $\alpha = 8^\circ$

Data set	Type	$Tu\%$	X_S/C	X_T/C	X_R/C
Radespiel <i>et al.</i> (2006)	Experiment	0.8	0.06	0.14	0.18
Galbraith and Visbal (2008)	ILES	NA	0.04	0.18	0.28
Germann and Visbal (2012)	ILES	NA	0.023	NA	0.259
Boom and Zingg (2013)	ILES	NA	0.037	0.105	0.20
Present computation	$k_T - k_L - \omega$ model	0.1	0.024	0.124	0.38
Present computation	$\gamma - Re_\theta$ SST model	0.1	0.039	0.152	0.331

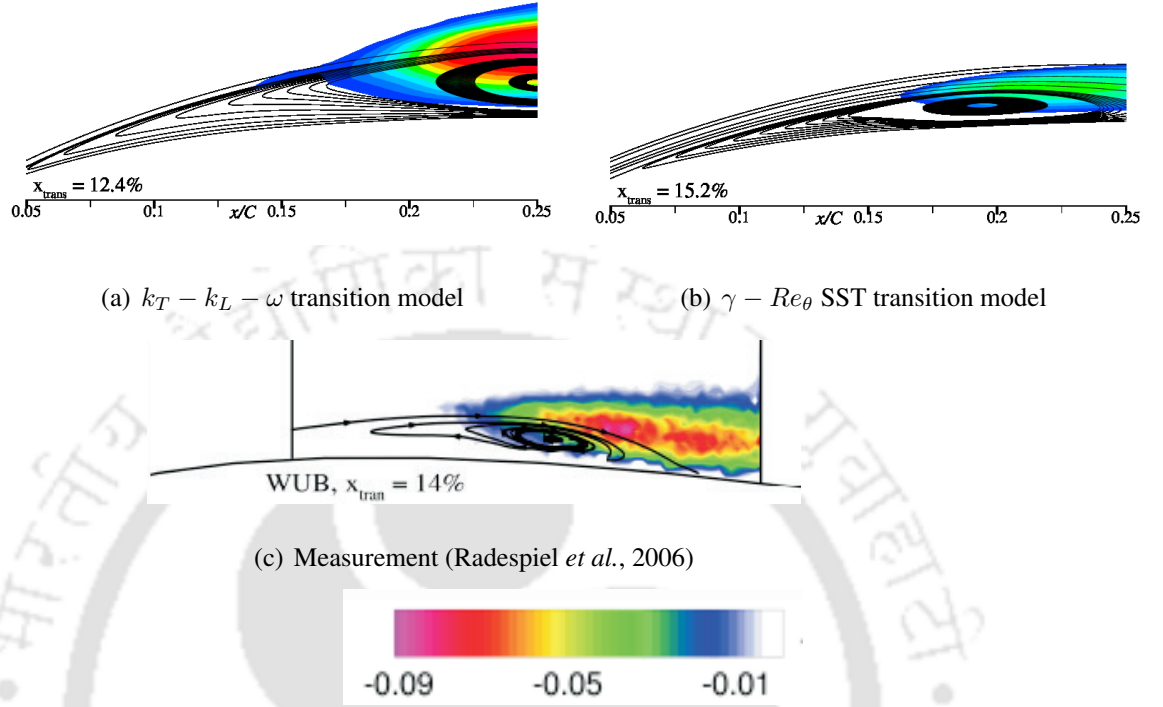


Figure 6.4: Comparison of normalized Reynolds shear stress contours at $\alpha = 8^\circ$

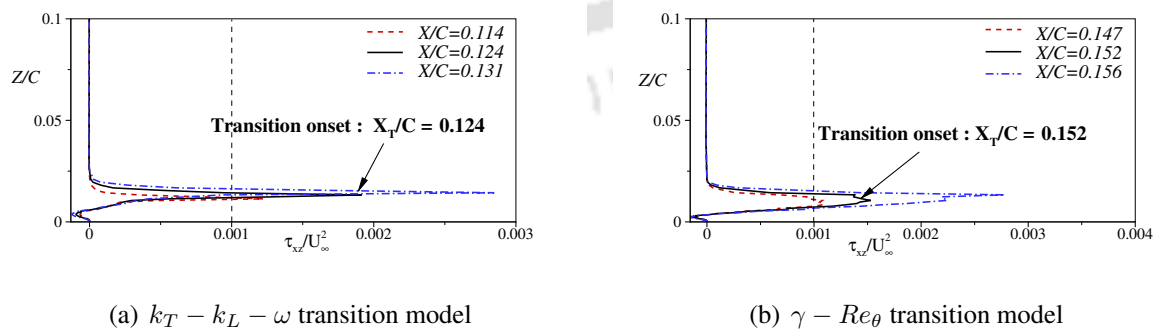


Figure 6.5: Normalized Reynolds shear stress profiles at $\alpha = 8^\circ$

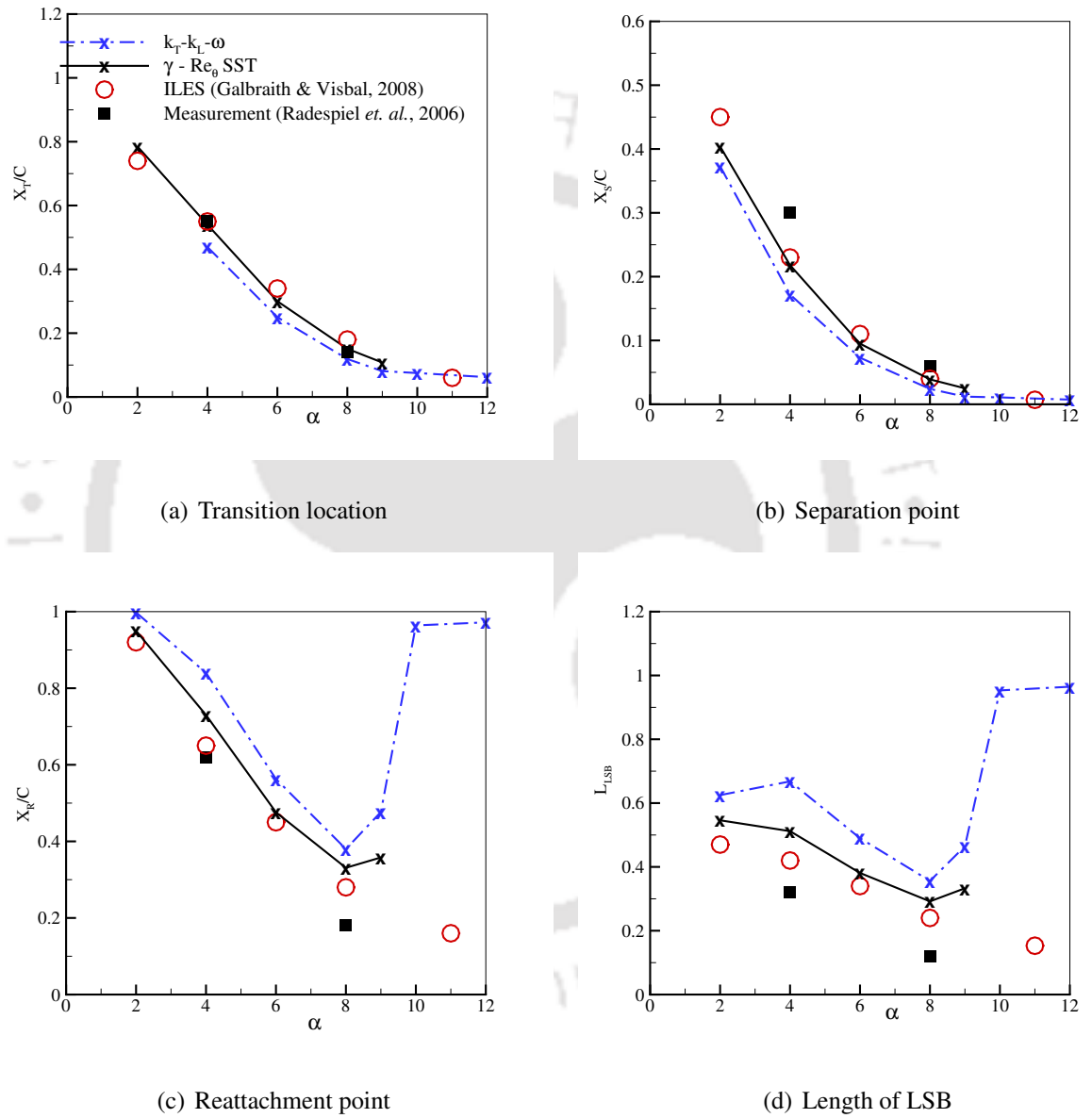


Figure 6.6: Variation of transition onset and length of LSB for SD7003 airfoil

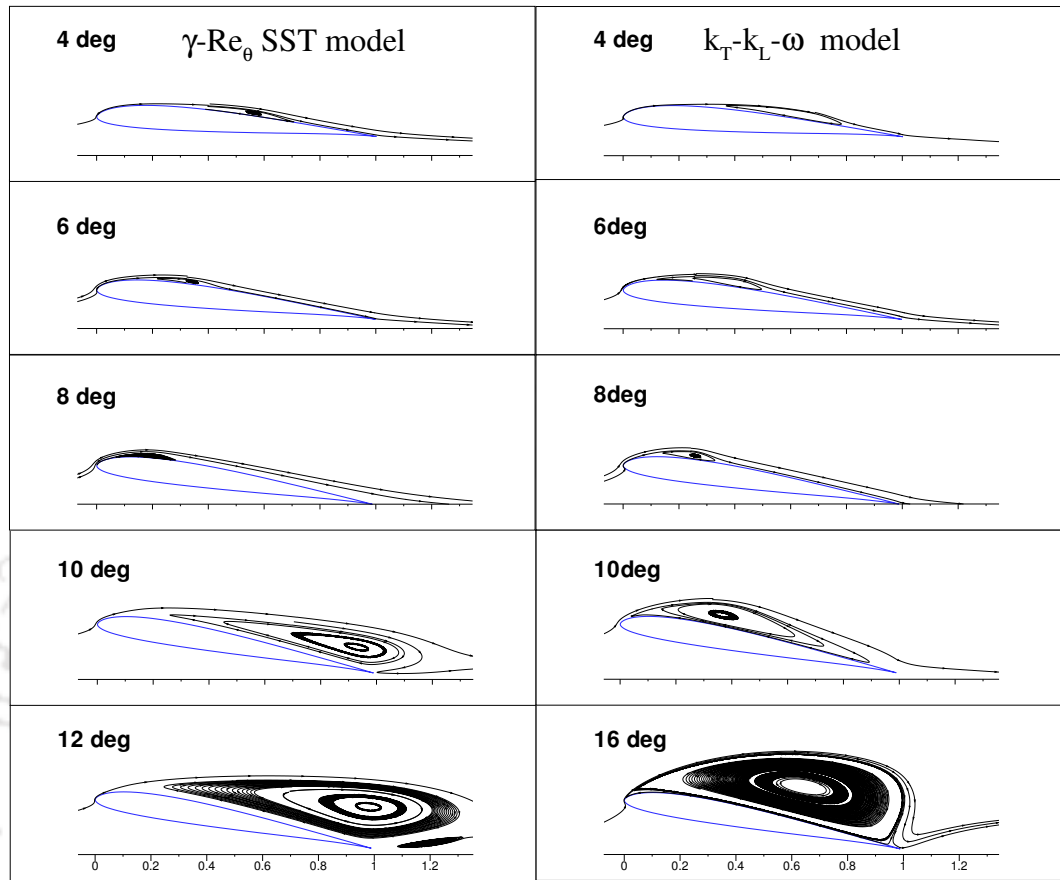


Figure 6.7: Effect of transition models on surface streamlines

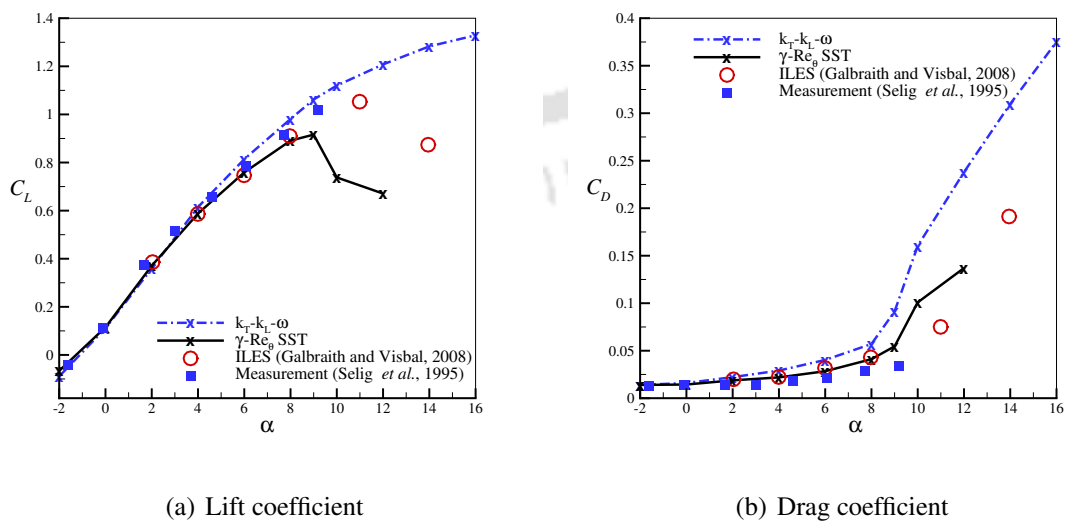
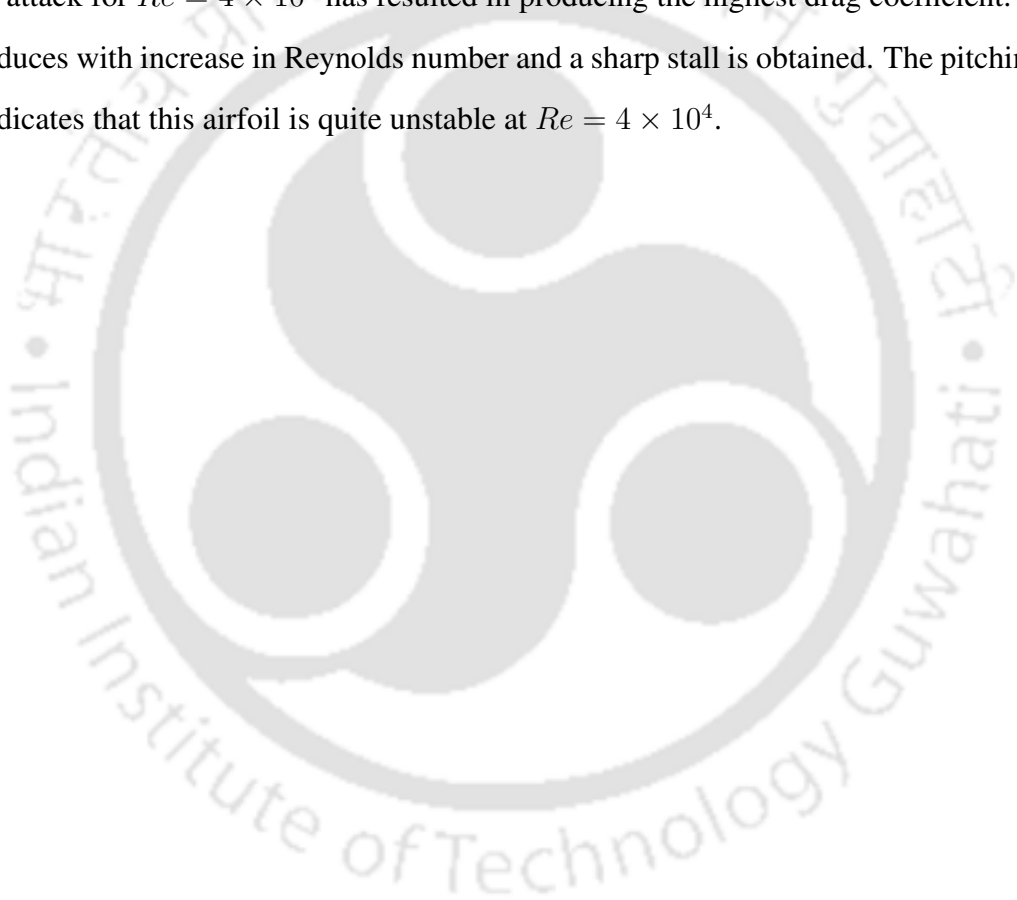


Figure 6.8: Comparison of aerodynamic coefficients with angle of attack

6.3 Influence of Reynolds Number on the Airfoil Performance

In this section the flow past SD7003 airfoil at three Reynolds number viz. $Re = 4 \times 10^4$, $Re = 6 \times 10^4$ and $Re = 1.5 \times 10^5$ are simulated using the $\gamma - Re_\theta$ SST transition model. The influence of the Reynolds number on the performance of this airfoil is analysed by comparing the transition onset location, laminar separation bubble and aerodynamic coefficients. The influence of Reynolds number on the aerodynamic and transition characteristics is discussed in details for $\alpha = 4^\circ$. The Reynolds shear stress contours and surface streamlines shown in Fig. 6.9 for $\alpha = 4^\circ$ clearly indicates the transition onset and formation of the laminar separation bubble for the three Reynolds number. From the Reynolds shear stress contours it is clearly evident that the transition onset moves upstream of the airfoil as Reynolds number increases and it further observed that increasing the Reynolds number has resulted in the reduction in the level of turbulence (as indicated by lower level of the Reynolds shear stress contours). This reduction in the turbulence level at higher Reynolds number because lesser fluid is dragged near the surface leading to a reduced shear stress in the boundary layer. The streamlines plot clearly shows the formation of the laminar separation bubble (the separation occurring before the transition onset) whose strength in terms of the length and height of the bubble is found to reduce with increase in Reynolds number. This feature of the shifting of the transition onset and reduction of the length of the LSB with increase in Reynolds number is clearly depicted in the line plot as shown in Fig. 6.10. The influence of Reynolds number on the aerodynamic coefficients is shown in Fig. 6.11. The aerodynamic performance at $\alpha = 4^\circ$ improves as the Reynolds number increases in term of higher lift coefficient and lower drag coefficient leading to an enhanced L/D ratio. The reduction in the drag coefficient at higher Re is because of its smaller size of the laminar separation bubble. The overall influence of the Reynolds number on the performance of SD7003 airfoil is shown in Fig. 6.12 and Fig. 6.13. The onset of transition location (Fig. 6.12) is observed to move upstream with increase in Reynolds number for all the angle of attack till stall. The flow at the lowest Re remains laminar to a greater extent as the transition onset is delayed when compared to the other two Reynolds number. The length of the laminar separation bubble is found to be largest for the lowest Re at all the angle of attack till stall. It is also noted that for all the Reynolds number the length of separation bubble initially increases

upto certain angles of attack and beyond which the length reduces. This feature is more prominent for the lowest Re with the length of LSB increasing upto a higher angle of attack compared to the other two Reynolds number. The overall performance of SD7003 is found to improve with increase in Reynolds number in terms of delayed stall angle, higher lift coefficient and lower drag resulting in a higher L/D ratio with almost constant pitching moment indicating more stability. The present computation could capture the soft stall and the nonlinearity in the lift coefficient curve expected at low Reynolds number (Winslow *et al.*, 2018). Due to the increase in the size of the LSB with angle of attack for $Re = 4 \times 10^4$ has resulted in producing the highest drag coefficient. The nonlinearity reduces with increase in Reynolds number and a sharp stall is obtained. The pitching moment curve indicates that this airfoil is quite unstable at $Re = 4 \times 10^4$.



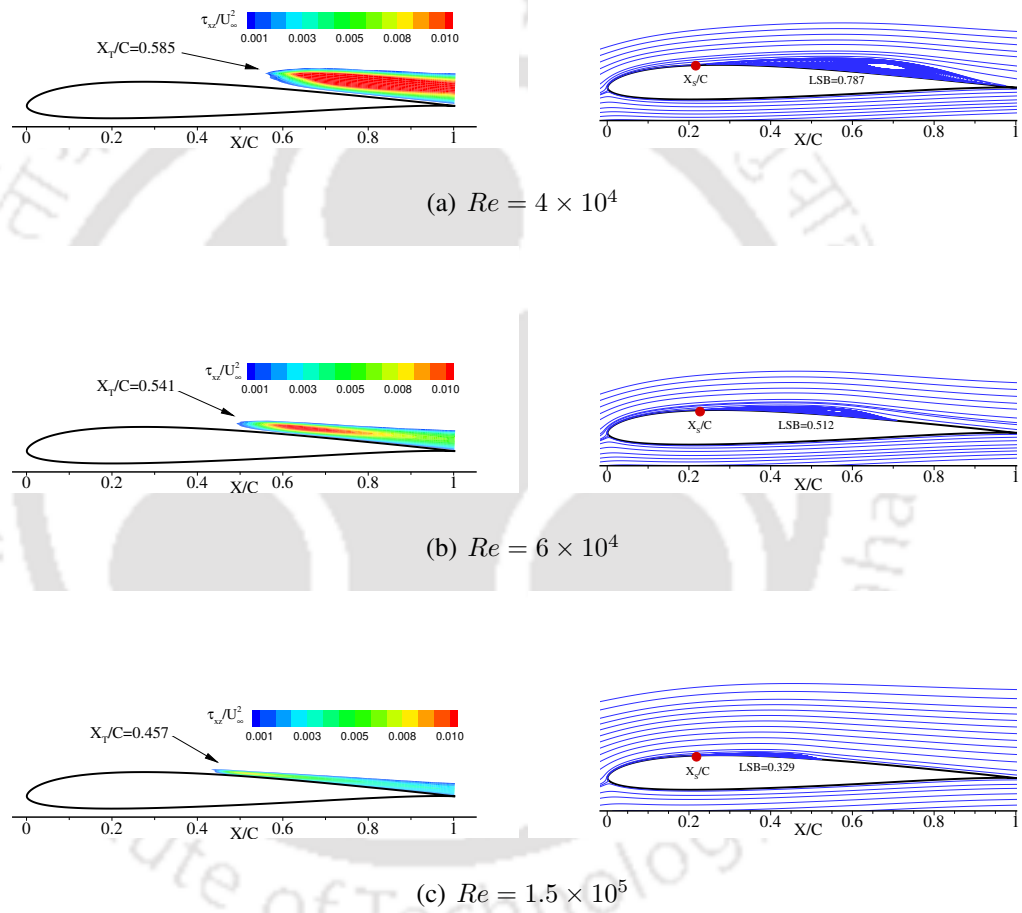


Figure 6.9: Reynolds shear stress contours and streamlines at different Re for $\alpha = 4^\circ$
(Left figure : Reynolds shear stress contours, Right figure : Surface streamlines)

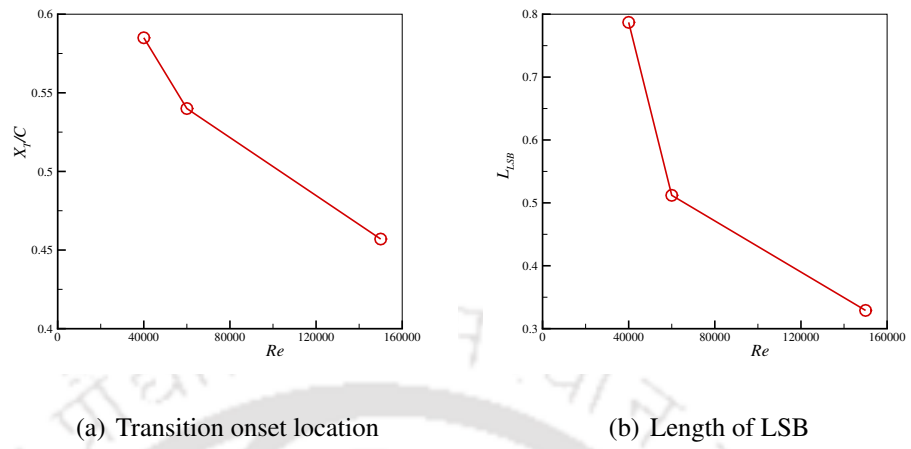


Figure 6.10: Effect of Reynolds number on transition characteristics for $\alpha = 4^\circ$

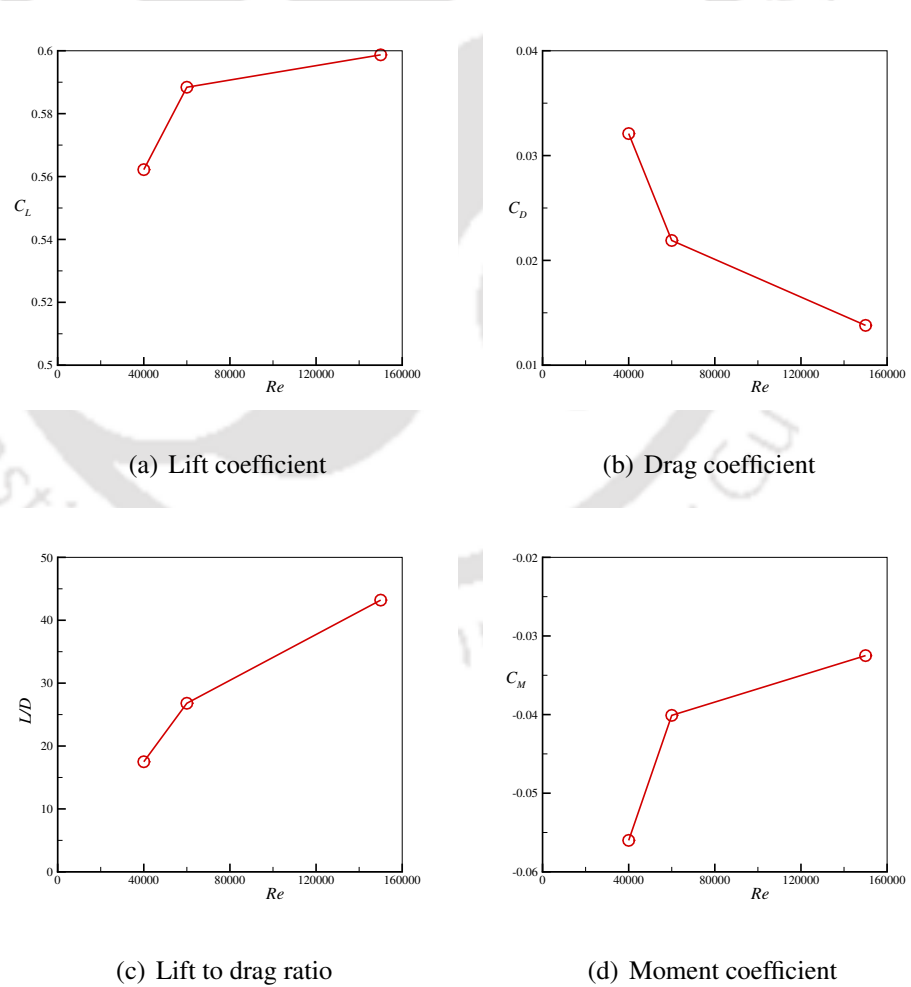


Figure 6.11: Effect of Reynolds number on aerodynamic characteristics for $\alpha = 4^\circ$

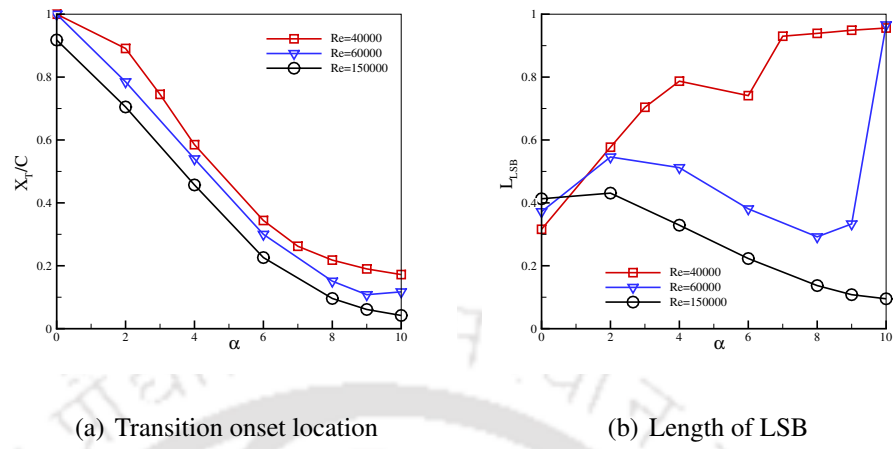


Figure 6.12: Effect of Reynolds number on transition characteristics

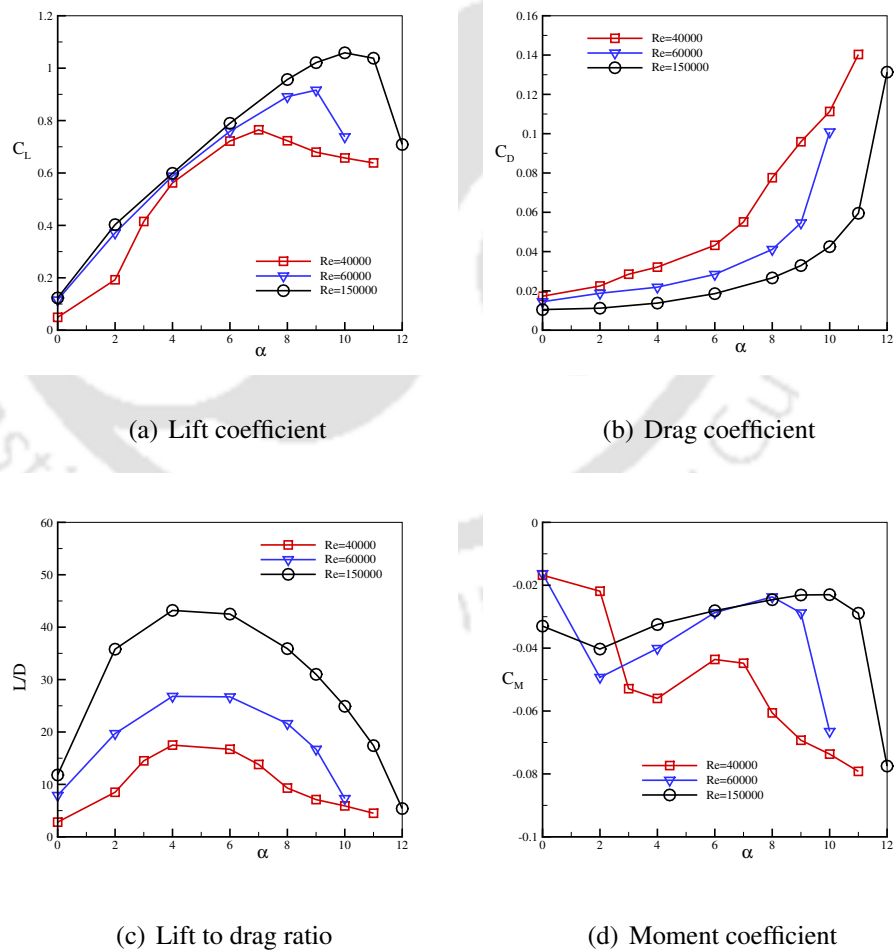


Figure 6.13: Effect of Reynolds number on aerodynamic characteristics

6.4 Summary

The $k_T - k_L - \omega$ and $\gamma - Re_\theta$ SST transition models were successfully used to predict the transition and aerodynamic characteristics of the SD7003 airfoil at $Re = 6 \times 10^4$ and study their relative performance. The two transition models could capture the transition onset location as well as the laminar separation bubble. The transition onset location obtained by $\gamma - Re_\theta$ SST transition model was found to be in good agreement with the measurement and other computations at all angles of attack but $k_T - k_L - \omega$ model predicted the location upstream of the airfoil which deviated by 15% on the average with the $\gamma - Re_\theta$ SST model. The computations in general have predicted the separation point upstream on the airfoil surface as compared to the measurement data. The separation location obtained by $\gamma - Re_\theta$ SST model is found to be in close agreement with that predicted by ILES data of Galbraith and Visbal (2008). The $k_T - k_L - \omega$ model similar to the transition location had predicted the separation location upstream leading to a larger discrepancy with the measurement data. However, both the transition models have predicted the reattachment downstream of the airfoil surface when compared to measurements as well as other LES computations the discrepancy being higher for $k_T - k_L - \omega$ model. The delayed in the reattachment point obtained by the present computation has resulted in the longer LSB. This delay in the reattachment point may be attributed to the lower level of turbulence predicted by these transition models and their inability to rapidly produce enough turbulence kinetic energy in the shear layer to induce early flow reattachment beyond the transition point. The longer LSB obtained by the $\gamma - Re_\theta$ SST model has lead to an early stall when compared to the ILES data. However this early stall was observed in the experiments conducted by Rival and Tropea (2010). On the other hand, the $k_T - k_L - \omega$ model failed to predict stall angle as this model in the present formulation is not designed to handle separated flow transition. It was further noticed that the computations in general predicted a slightly higher drag compared to the measurements with $\gamma - Re_\theta$ SST model following the ILES data closely. The drag predicted by the $k_T - k_L - \omega$ model was much higher as the LSB obtained by this model was much larger both in terms of length and height when compared to $\gamma - Re_\theta$ SST model. The flow pattern obtained by the $k_T - k_L - \omega$ model showed the formation of a large bubble attached to the airfoil surface with length of the bubble remaining constant beyond $\alpha = 8^\circ$ but the height of the bubble was found to increases

with increase in angle of attack resulting in a continuous increase in the lift coefficient and due to this the stall angle was not predicted even at $\alpha = 16^\circ$. The transition and aerodynamic performance of the SD7003 airfoil were found to be influenced by the change in Reynolds number in the low Re regime (4×10^4 to 1.5×10^5). The aerodynamic performance was observed to improve with increase in Reynolds number from $Re = 4 \times 10^4$ to $Re = 1.5 \times 10^5$ which is mainly because the strength of the laminar separation bubble is weaker. The soft stall and nonlinearity in lift coefficient curve features expected at low Re flows were captured by the present simulation at the lowest Reynolds number ($Re = 4 \times 10^4$).



Chapter 7

Flow Analysis of SD Airfoil Series at Low Re

7.1 Background

The airfoil geometry is mainly characterized by its camber, location of the maximum camber and thickness. In this chapter, systematic numerical study has been carried out to investigate the effect of camber and thickness which significantly influences the aerodynamic and transition performance of the airfoil. This study was carried out for two Reynolds numbers viz. 6×10^4 and 2×10^5 which falls in the lower and upper end of the low Reynolds number flow regime ($10^4 - 10^5$). The airfoils having different camber and thickness for this study were selected from the SD airfoil series such that the measurement data are available in open literature for above selected Reynolds number. The present computational results are validated only for the aerodynamic data as the measurements are available only for the aerodynamic coefficients.

7.2 Influence of Varying Camber on Flow Characteristics

7.2.1 Aerodynamic characteristics

To understand the effect of camber on the aerodynamic performance at low Reynolds numbers, three airfoils namely SD7090, SD8040 and SD7032 as shown in Fig. 7.1 are selected from low

speed airfoil data of Selig *et al.* (1989) in SD airfoil series such that they have similar thickness but varying camber. The difference in change in camber between highest camber airfoil (SD7032) to lowest camber airfoil (SD7090) is about 95%. The flow analysis for these airfoils were carried out at $Re = 6 \times 10^4$ and 2×10^5 using the $\gamma - Re_\theta$ transition model. The aerodynamic coefficients obtained at these two Reynolds number for all the three airfoils are compared with the measurement data of Selig *et al.* (Selig *et al.*, 1989) as shown in Fig. 7.2 to Fig. 7.7.

The aerodynamic coefficients obtained at $Re = 6 \times 10^4$ (Fig. 7.2 to Fig. 7.4) was found to have fairly good match with the measurement data for all the three airfoil with varying camber. The lift coefficient for airfoil with the lowest camber (SD7090) at lower angles of attack was found to be slightly underpredicted as compared to the measurement data (Fig. 7.2). This discrepancy reduced for airfoils having higher camber SD8040 and SD7032 as shown in Fig. 7.3 and Fig. 7.4 respectively. However, the stall angle predicted for SD7090 was closer to the measurement data as compared to the other two airfoils having higher camber. The present computation has predicted an early stall for the airfoil with higher camber and this discrepancies with the measurement data was found to increase with increase in camber. This discrepancies in the stall angle is because in the present computation the stall angle obtained was independent of the airfoil camber unlike that obtained by the measurement data. The drag polar matched very well with the measurement data at lower lift coefficient for all the three airfoils. However for all the three airfoils, some discrepancies were found at higher lift coefficient. The difference between the measurement and computation in the drag polar at higher lift coefficient was also reflected in the L/D plots.

The computed aerodynamic coefficient at $Re = 2 \times 10^5$ (Fig. 7.5 to Fig. 7.7) was found to have better agreement with the measurement when compared to that at the lower Re . In this case, the maximum difference in the predicted aerodynamic coefficients with the measurement data was observed for SD8040 airfoil. However, the predicted stall angle was found to be in good agreement with the measurement data for all the three airfoils as stall angle obtained by the experiments were almost independent of the airfoil camber.

The effect of the camber on the aerodynamic coefficients predicted at the two Reynolds number are summarized in Fig. 7.8 and Fig. 7.9. The lift to drag ratio (Fig. 7.8(c)) at $Re = 6 \times 10^4$ did signifi-

cantly change when the airfoil camber was increased. However, the lift coefficient and maximum lift coefficient (Fig. 7.8(a)) were found to increase with camber while the stall angle remained almost constant. The drag coefficient (Fig. 7.8(b)) obtained at lower angles of attack (lower C_L) for the three airfoils were not significantly different. However, at higher C_L , the drag coefficient obtained by the three airfoils was found to differ with the airfoil with lowest camber producing the lowest drag coefficient. The slope of moment coefficient (C_M) is almost identical for all the three airfoils with its value decreasing with increase in camber. The effect of camber at higher Re (Fig. 7.9) was similar to that obtained at lower Re but the influence of camber on the aerodynamic coefficients were more pronounced at $Re = 2 \times 10^5$.

The effect of camber at the two Reynolds number on key aerodynamic parameters (stall angle, $(L/D)_{max}$) are compared with available measurements and shown in Fig. 7.10. The stall angle obtained by measurements at $Re = 6 \times 10^4$ was found to increase with the airfoil camber. However, the predicted stall angle is independent of the airfoil camber which has lead to the discrepancies observed for the airfoil with highest camber. The stall angle was delayed for all the three airfoils when the Reynolds number was increased. At this $Re = 2 \times 10^5$, computations indicate that the stall angle is independent of airfoil camber and found to be in a better agreement with the measurement data. The variation of maximum lift to drag ratio with the airfoil camber at $Re = 6 \times 10^4$ was found to be almost constant for both computation and measurement with the computations predicting a lower $(L/D)_{max}$ ratio. On the other hand, at $Re = 2 \times 10^5$ the predicted $(L/D)_{max}$ ratio is found to be in a good agreement with the measurement data where the increasing the camber has enhanced the L/D ratio.

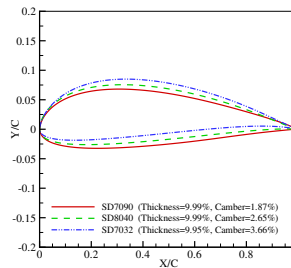
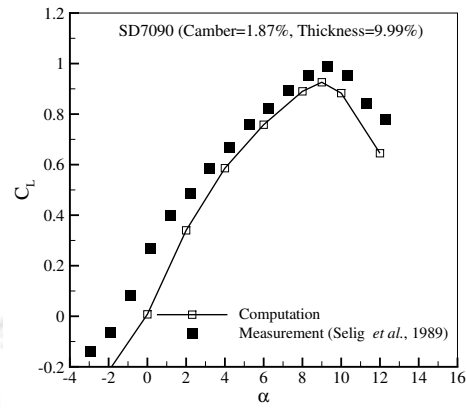
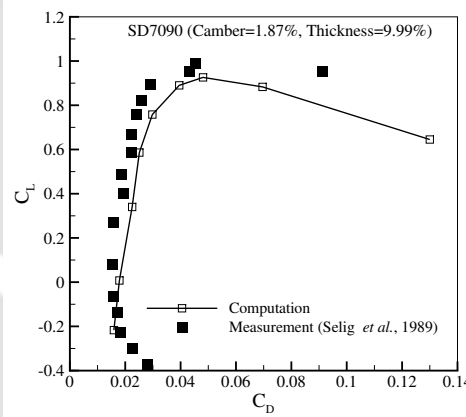


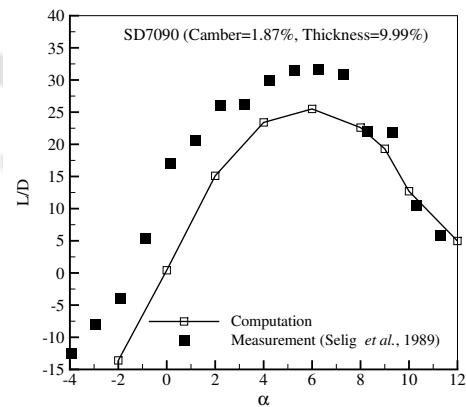
Figure 7.1: SD airfoil geometries having different camber



(a) Lift coefficient



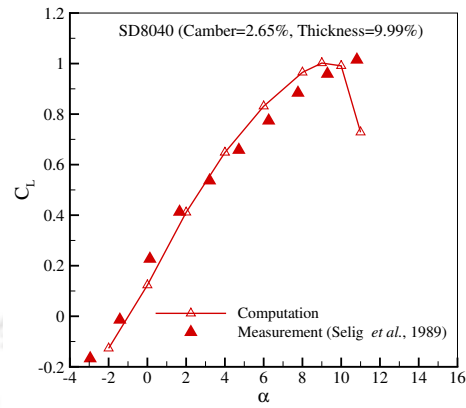
(b) Drag polar



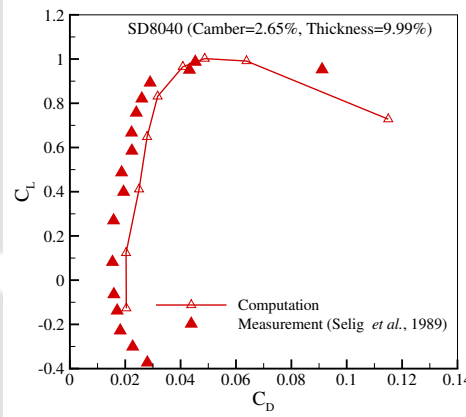
(c) Lift to drag ratio

Figure 7.2: Validation of the aerodynamic performance of SD7090 airfoil at lower Re

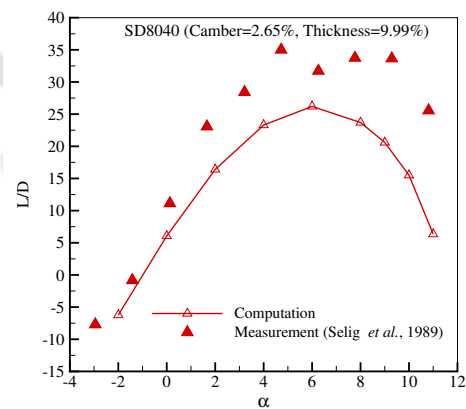
(Computation : $Re = 6 \times 10^4$, Measurement : $Re = 5.96 \times 10^4$)



(a) Lift coefficient



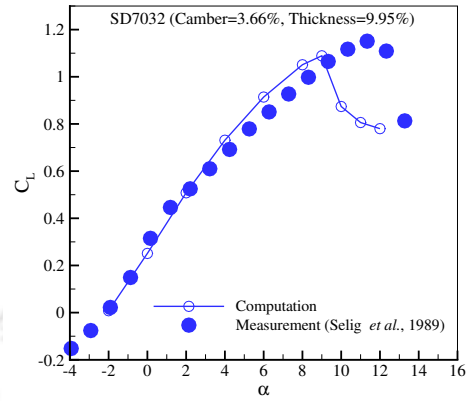
(b) Drag polar



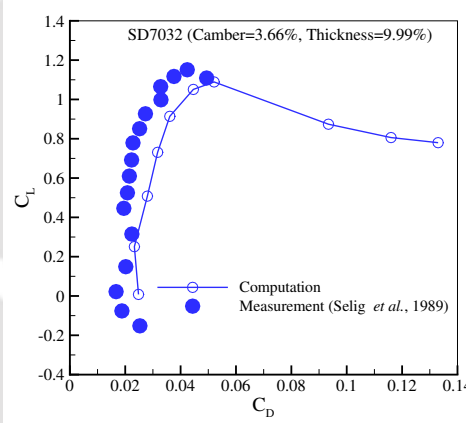
(c) Lift to drag ratio

Figure 7.3: Validation of the aerodynamic performance of SD8040 airfoil at lower Re

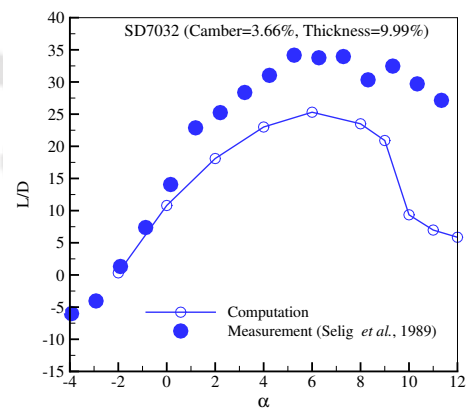
(Computation : $Re = 6 \times 10^4$, Measurement : $Re = 6.21 \times 10^4$)



(a) Lift coefficient



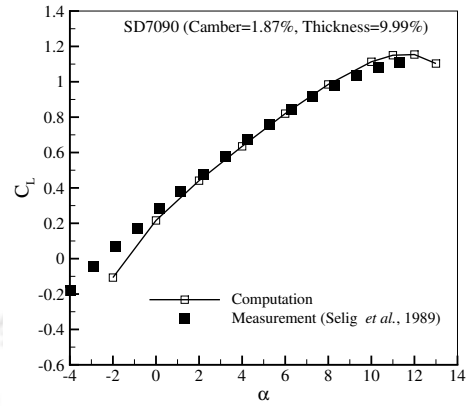
(b) Drag polar



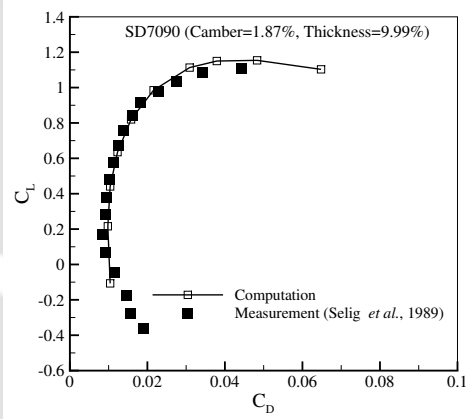
(c) Lift to drag ratio

Figure 7.4: Validation of the aerodynamic performance of SD7032 airfoil at lower Re

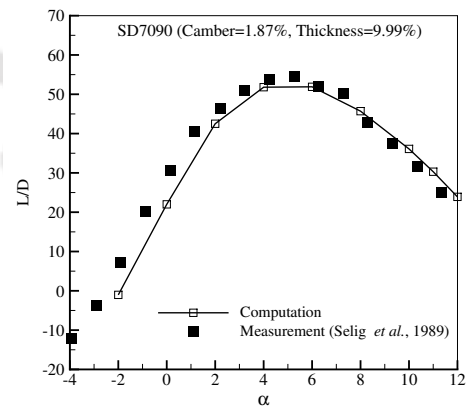
(Computation : $Re = 6 \times 10^4$, Measurement : $Re = 6.42 \times 10^4$)



(a) Lift coefficient



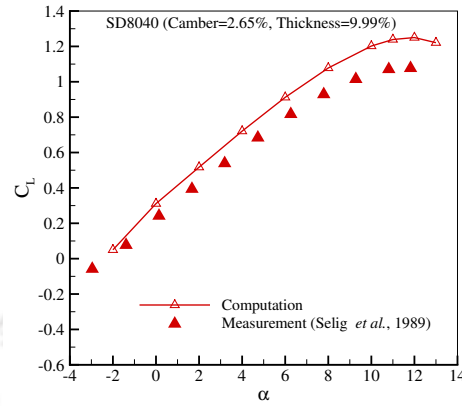
(b) Drag polar



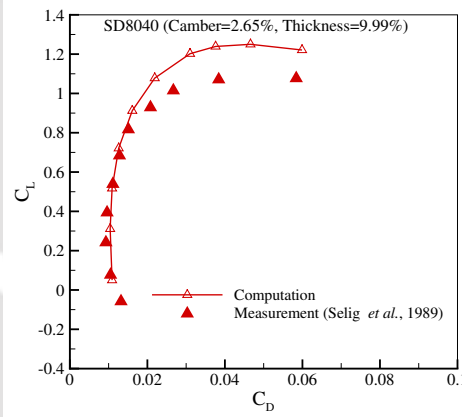
(c) Lift to drag ratio

Figure 7.5: Validation of the aerodynamic performance of SD7090 airfoil at higher Re

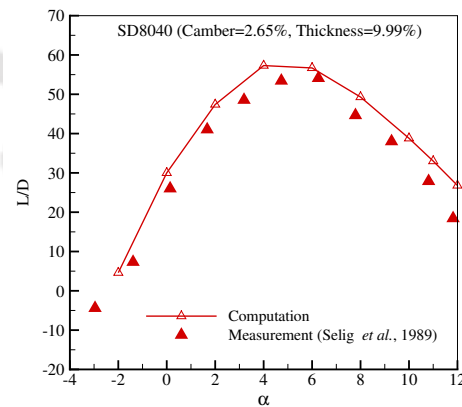
(Computation : $Re = 2 \times 10^5$, Measurement : $Re = 1.991 \times 10^5$)



(a) Lift coefficient



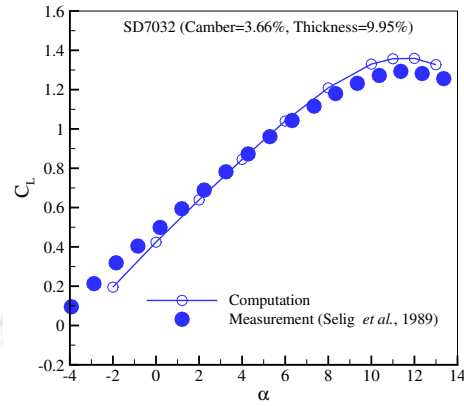
(b) Drag polar



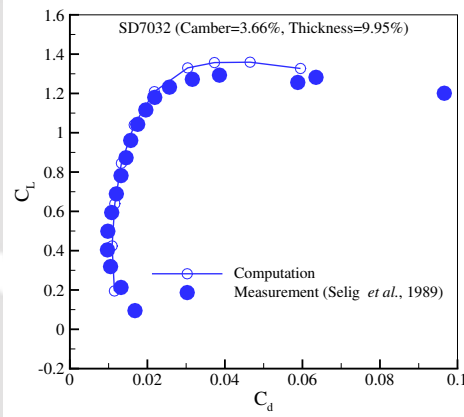
(c) Lift to drag ratio

Figure 7.6: Validation of the aerodynamic performance of SD8040 airfoil at higher Re

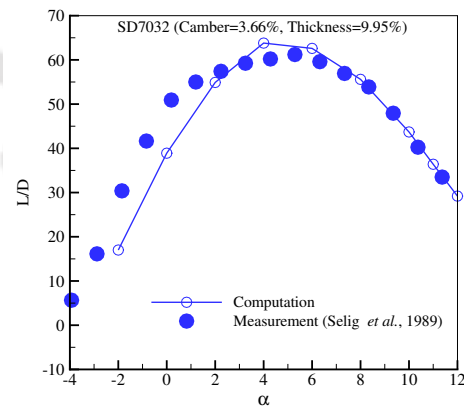
(Computation : $Re = 2 \times 10^5$, Measurement : $Re = 2.049 \times 10^5$)



(a) Lift coefficient



(b) Drag polar



(c) Lift to drag ratio

Figure 7.7: Validation of the aerodynamic performance of SD7032 airfoil at higher Re

(Computation : $Re = 2 \times 10^5$, Measurement : $Re = 2.001 \times 10^5$)

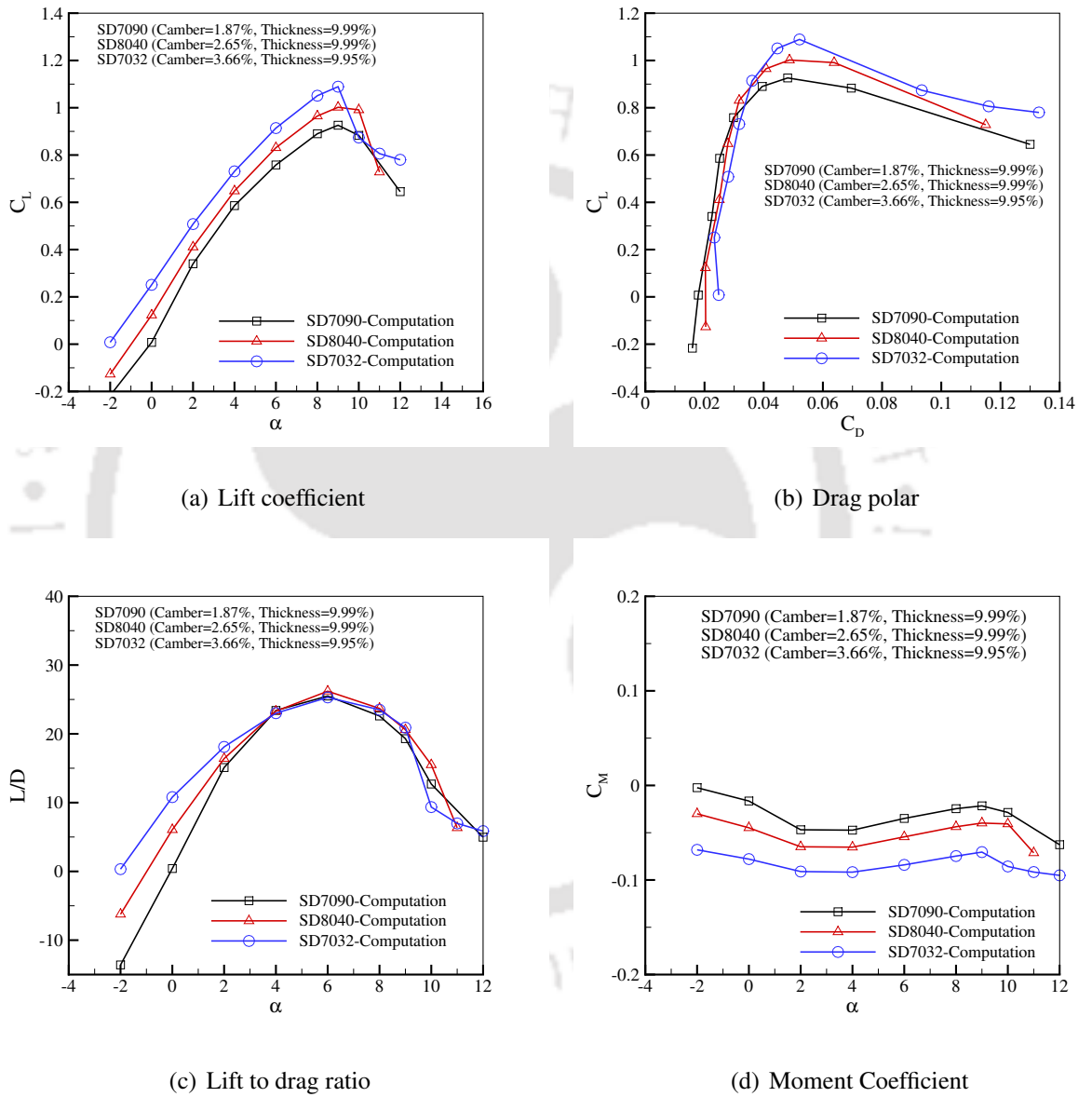
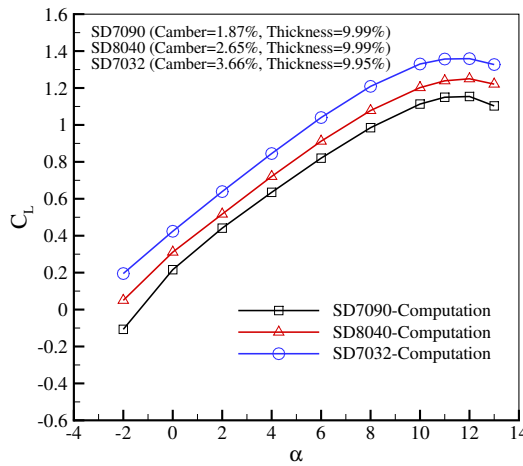
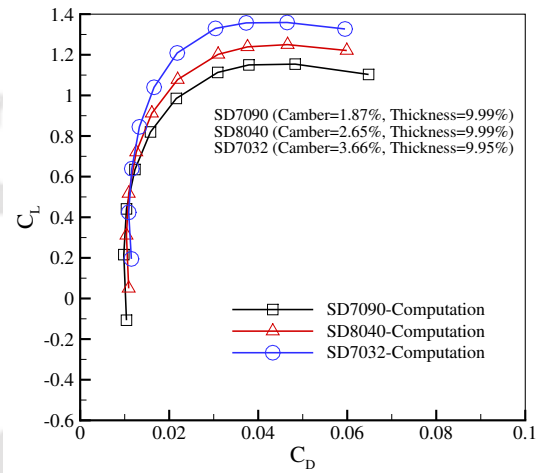


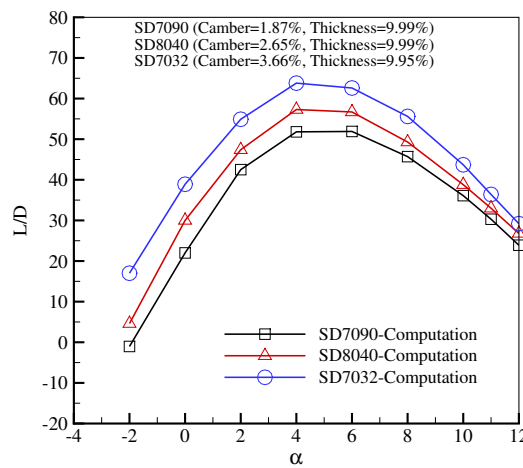
Figure 7.8: Influence of airfoil camber on aerodynamic performance at $Re = 6 \times 10^4$



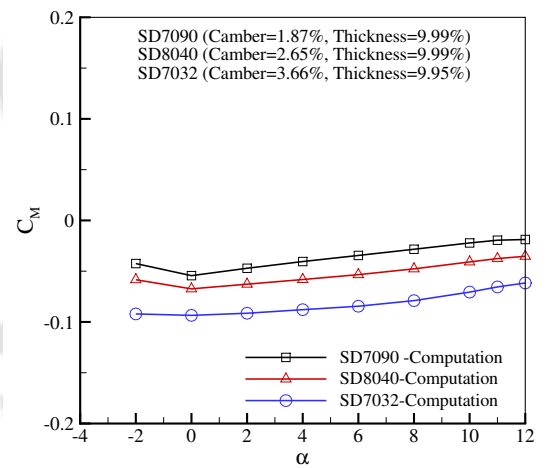
(a) Lift coefficient



(b) Drag polar

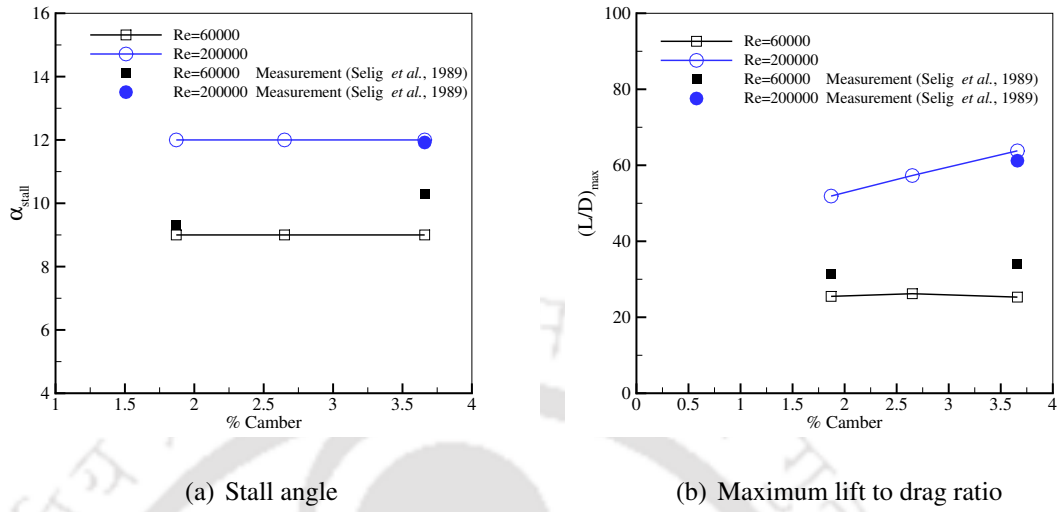


(c) Lift to drag ratio



(d) Moment Coefficient

Figure 7.9: Influence of airfoil camber on aerodynamic performance at $Re = 2 \times 10^5$

Figure 7.10: Influence of camber on aerodynamic parameters at different Re

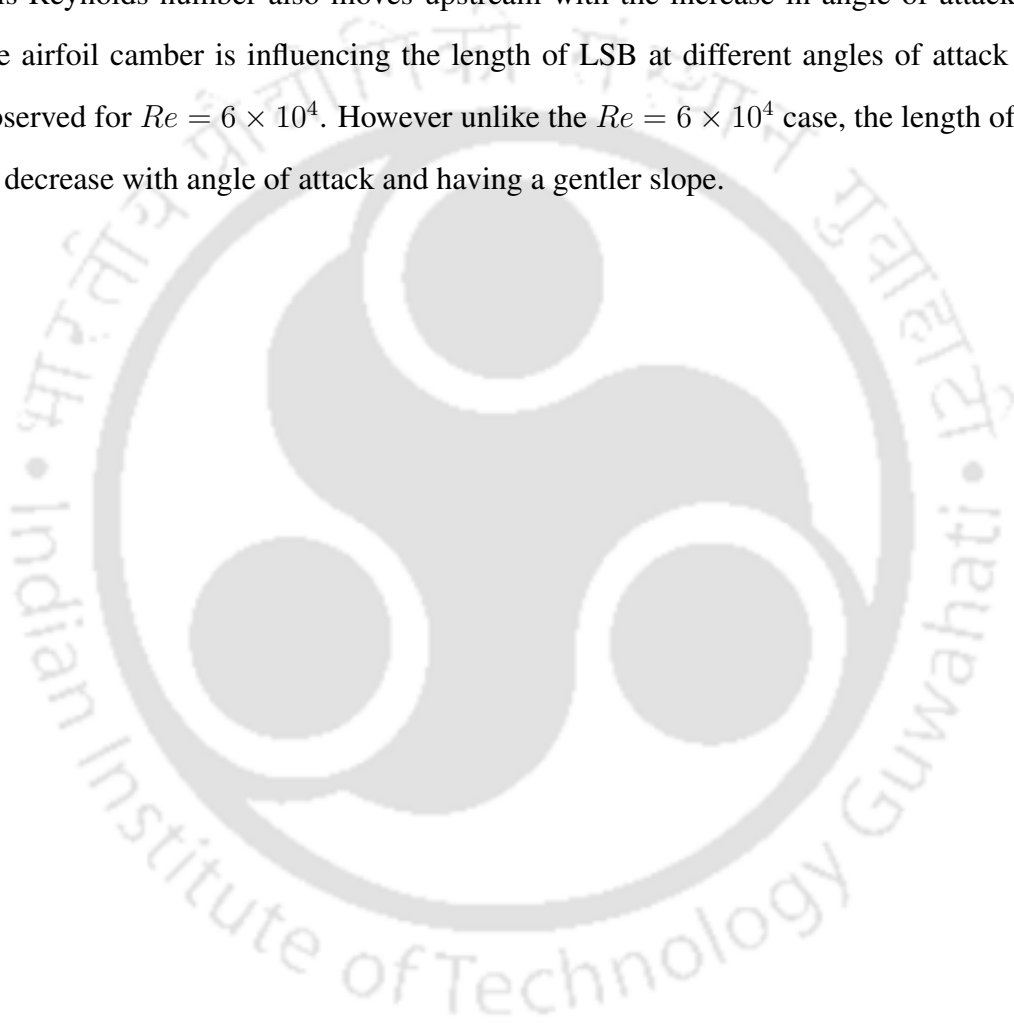
7.2.2 Transition characteristics

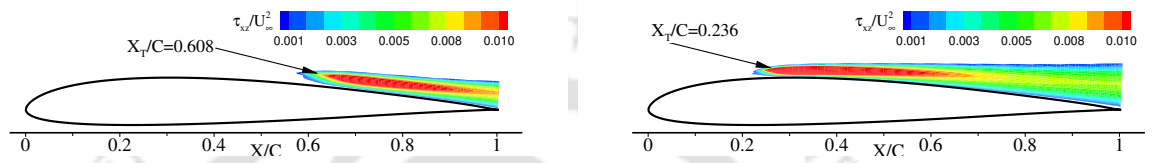
The influence of varying camber on the transition phenomenon at the two Reynolds number is discussed in this section. In this study, the transition phenomenon is characterised by transition onset location and length of laminar separation bubble (LSB). The effect of camber on the computed Reynolds shear stress contours and surface streamlines are analysed for the two Reynolds number at two typical angle of attack. The Reynolds shear stress contours (Fig. 7.11) and surface streamlines (Fig. 7.12) obtained for $Re = 6 \times 10^4$ at angle of attack 4° and 8° clearly indicate the transition onset and the formation of the separation bubble for the three airfoils with varying camber. It is clearly evident from the Reynolds shear stress contours at $Re = 6 \times 10^4$ that the transition onset location is independent of the airfoil camber for the lower angle of attack ($\alpha = 4^\circ$) but at higher angle of attack ($\alpha = 8^\circ$) increasing the camber has resulted in a downstream shift of the transition onset leading to a more laminar region for the airfoil with highest camber. The transition onset location as expected moves upstream for all the three airfoils when the angle of attack is increased from 4° to 8° . The airfoil camber at this Reynolds number is observed to have a mild influence on the inception of laminar separation point and the length of the LSB for both the angles of attack as

indicated in the streamlines plot. It is observed that at the lower angle of attack, increasing the camber has moved only the separation point slightly upstream which has lead to a small reduction in the length of the LSB. However at the higher angle of attack, increasing the airfoil camber has resulted in a increase in the length of the separation bubble as both the laminar separation point and turbulent reattachment point has moved downstream compared to the airfoil with lower camber. In the higher Reynolds number ($Re = 2 \times 10^5$) case, the level of the Reynolds shear stress contours for both the angles of attack (Fig. 7.13) is observed to be much lesser than that obtained at $Re = 6 \times 10^4$ for all the three airfoils. This reduction in the level of the Reynolds shear stress with increase in Reynolds number is also discussed in the previous Chapter. In spite of having lower level of the Reynolds shear stress, the transition onset for all the three airfoils at $Re = 2 \times 10^5$ is predicted earlier than compared to the lower Re case. It is further evident from the Reynolds shear stress contours that at higher Re , the airfoil camber has a notable influence on the transition onset location at both the angles of attack. Similar to the low Re case, it is found that increasing the camber has resulted in delaying the transition onset and thus making the flow to remain laminar to a greater extent. The surface streamlines (Fig. 7.14) at $Re = 2 \times 10^5$ clearly point out the formation of a weaker laminar separation bubble as compared to the lower Re case. Influence of airfoil camber on the formation of the separation bubble at $Re = 2 \times 10^5$ is similar to that observed for the lower Re case.

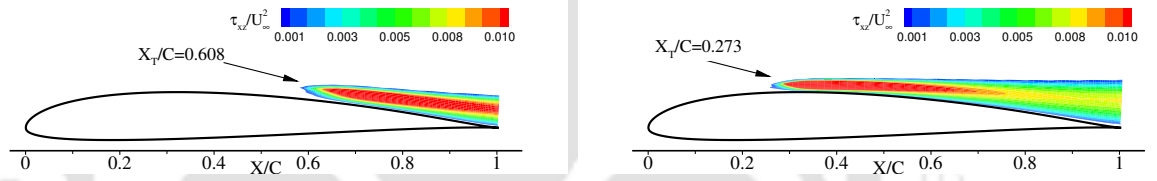
The influence of airfoil camber on the transition onset location and length of LSB for all the angles of attack is shown in Fig. 7.15 and Fig. 7.16 for $Re = 6 \times 10^4$ and $Re = 2 \times 10^5$ respectively. In the $Re = 6 \times 10^4$ case, it is observed that the transition onset at $\alpha = 4^\circ$ is not effected by the airfoil as observed in the Reynolds shear stress contour and discussed above. However for $\alpha > 4^\circ$, increasing the airfoil camber resulted in an upward shift of transition onset and this trend was found to be reversed for $\alpha < 4^\circ$. The transition onset for all the three airfoils moves upstream with angles of attack and at the stall angle the flow is almost fully turbulent. The length of LSB for $\alpha \leq 4^\circ$ was mildly influenced by airfoil camber as compared to higher angles of attack. The length LSB at lower angles of attack was found to reduce with increase in the airfoil camber and for $\alpha > 4^\circ$ increasing the airfoil camber resulted in a longer LSB which corroborates with the surface streamlines plots. For all the three airfoils at $Re = 6 \times 10^4$, a sharp increase in the length of LSB is observed till $\alpha = 4^\circ$ and beyond which the length of LSB reduces steeply. On

the other hand, at higher Reynolds number the effect of airfoil camber on the transition onset is mild at lower at angle of attack ($\alpha \leq 4^\circ$). However at higher angle of attack the transition onset is significantly influenced by the airfoil camber. At this Reynolds number, for all the angles of attack, the transition onset is found to be delayed with increase in the airfoil camber. This trend is also observed in the Reynolds shear stress contours as discussed above. The transition onset for this Reynolds number also moves upstream with the increase in angle of attack. The trend how the airfoil camber is influencing the length of LSB at different angles of attack is similar to that observed for $Re = 6 \times 10^4$. However unlike the $Re = 6 \times 10^4$ case, the length of LSB is observed to decrease with angle of attack and having a gentler slope.

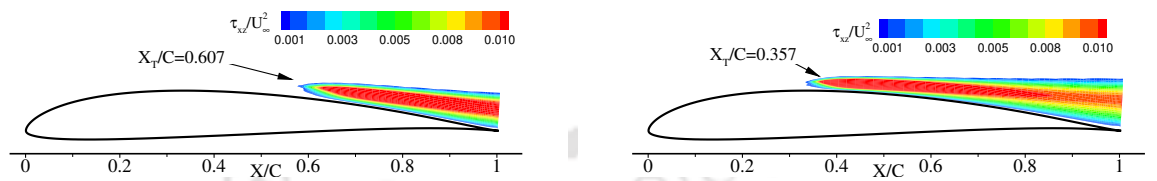




(a) SD7090 Airfoil (camber=1.87%, thickness=9.99%)



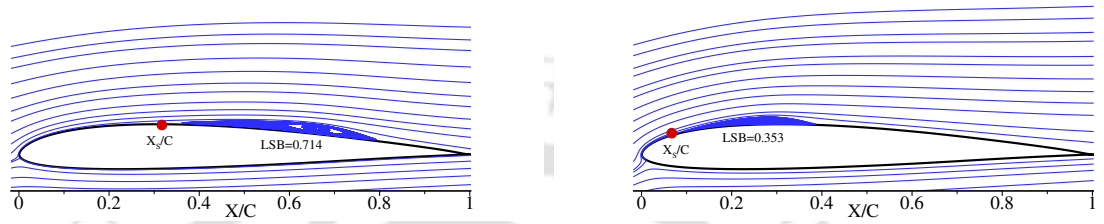
(b) SD8040 Airfoil (camber=2.65%, thickness=9.99%)



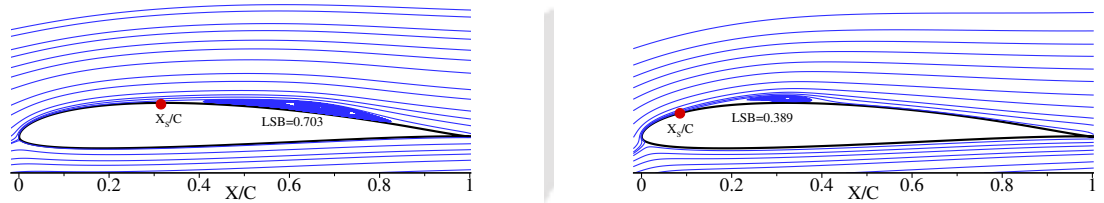
(c) SD7032 Airfoil (camber=3.66%, thickness=9.95%)

Figure 7.11: Influence of camber on Reynolds shear stress at $Re = 6 \times 10^4$

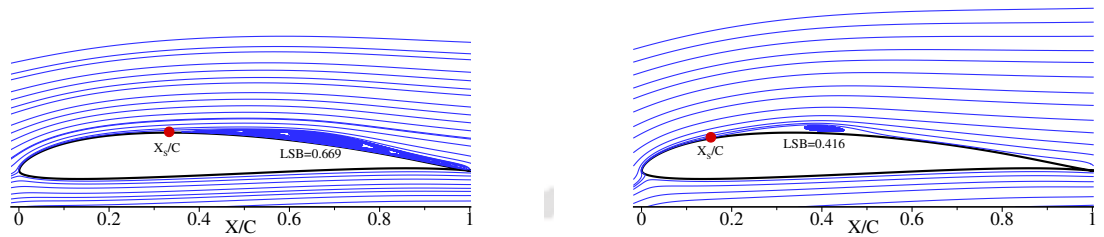
(Left figure : $\alpha = 4^\circ$, Right figure : $\alpha = 8^\circ$)



(a) SD7090 Airfoil (camber=1.87%, thickness =9.99%)



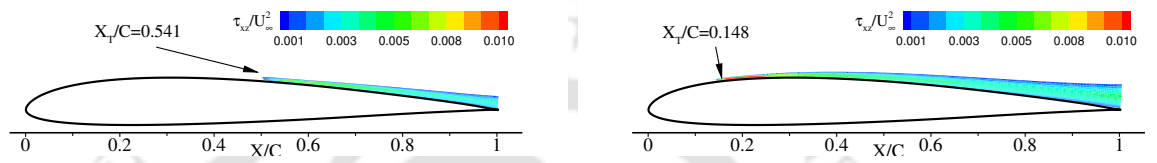
(b) SD8040 Airfoil (camber=2.65%, thickness =9.99%)



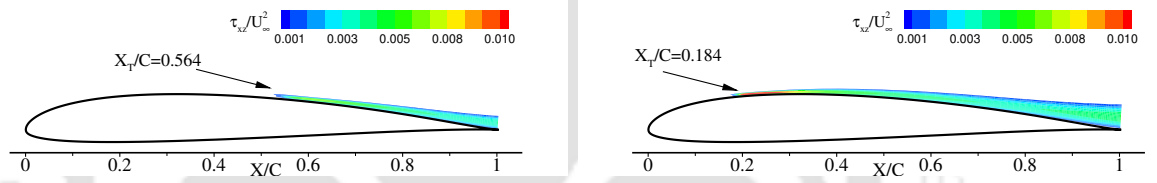
(c) SD7032 Airfoil (camber=3.66%, thickness =9.95%)

Figure 7.12: Influence of camber on surface streamlines at $Re = 6 \times 10^4$

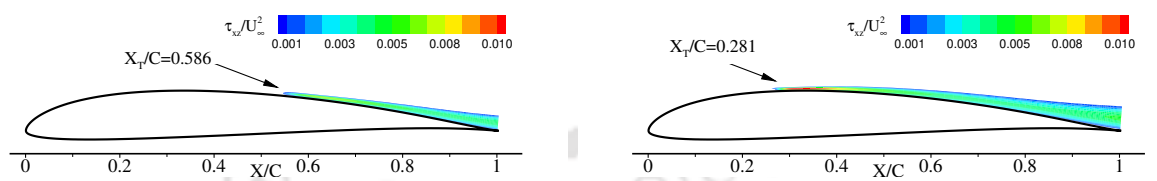
(Left figure : $\alpha = 4^\circ$, Right figure : $\alpha = 8^\circ$)



(a) SD7090 Airfoil (camber=1.87%, thickness =9.99%)



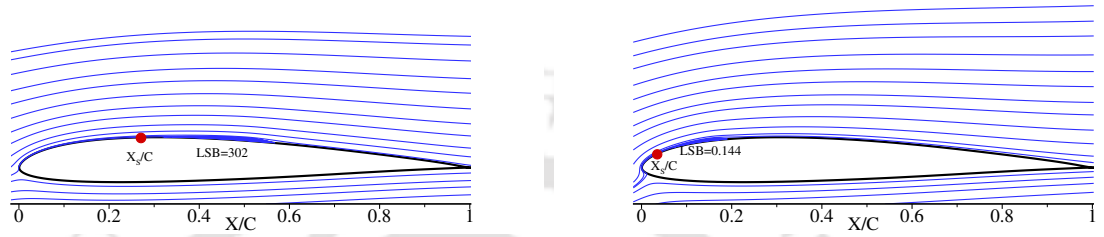
(b) SD8040 Airfoil (camber=2.65%, thickness =9.99%)



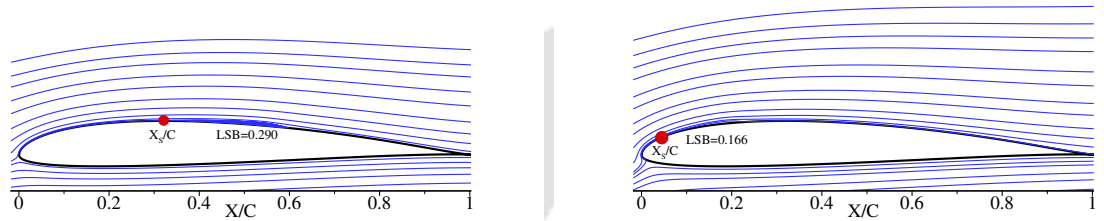
(c) SD7032 Airfoil (camber=3.66%, thickness =9.95%)

Figure 7.13: Influence of camber on Reynolds shear stress at $Re = 2 \times 10^5$

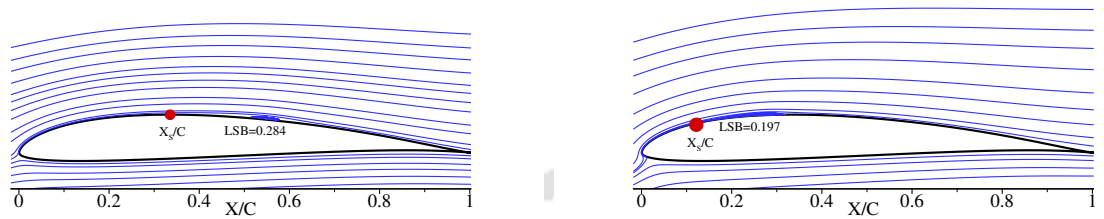
(Left figure : $\alpha = 4^\circ$, Right figure : $\alpha = 8^\circ$)



(a) SD7090 Airfoil (camber=1.87%, thickness =9.99%)



(b) SD8040 Airfoil (camber=2.65%, thickness =9.99%)



(c) SD7032 Airfoil (camber=3.66%, thickness =9.95%)

Figure 7.14: Influence of camber on surface streamlines at $Re = 2 \times 10^5$

(Left figure : $\alpha = 4^\circ$, Right figure : $\alpha = 8^\circ$)

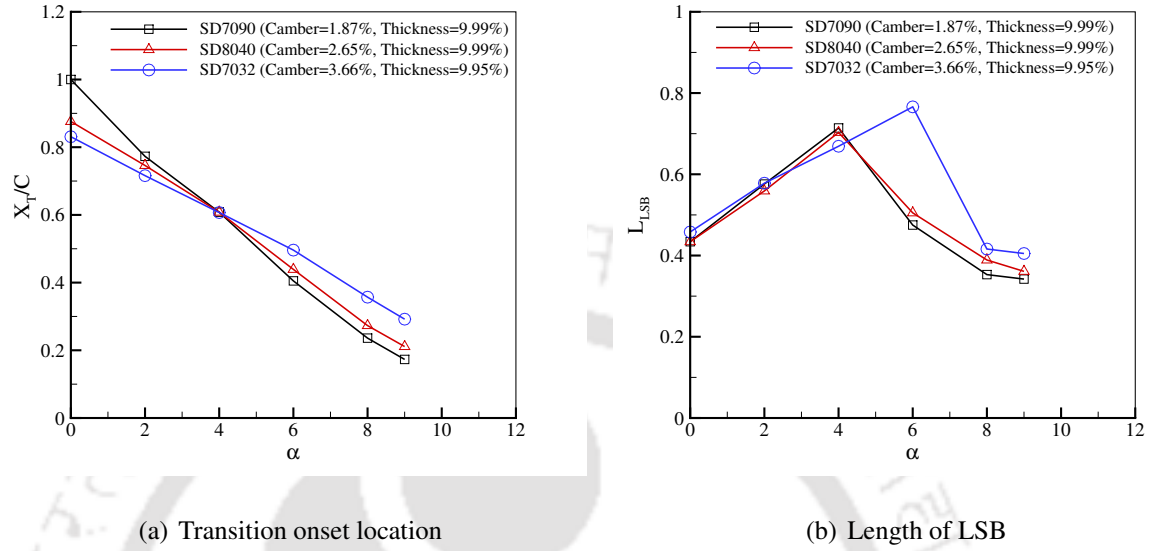


Figure 7.15: Influence of airfoil camber on transition characteristics at $Re = 6 \times 10^4$

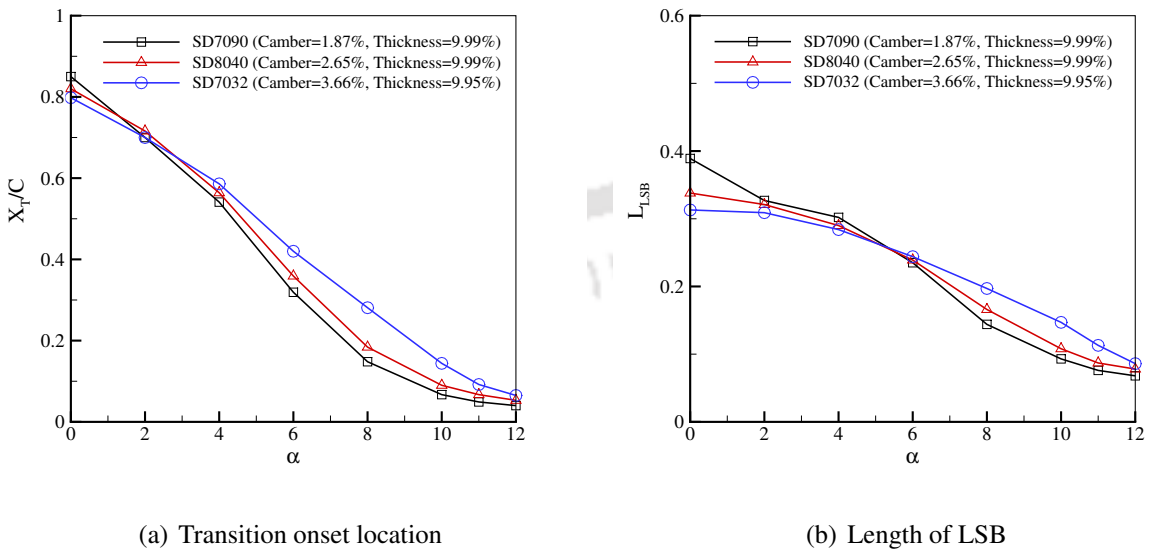


Figure 7.16: Influence of airfoil camber on transition characteristics at $Re = 2 \times 10^5$

7.3 Influence of Varying Thickness on Flow Characteristics

7.3.1 Aerodynamic characteristics

The effect of thickness on the aerodynamic and transition characteristics were carried out using $\gamma - Re_\theta$ transition model at the two Reynolds numbers as chosen for the effect of airfoil camber. In the low speed airfoil data of Selig *et al.* (1989) only two airfoils having similar camber and different thickness were available for SD series therefore only these two airfoils were considered for this study. The geometry of the SD2030 and SD5060 airfoils having similar camber and different thickness are given in Fig. 7.17 where the thickness differ by about only 10%. The validation of the aerodynamic coefficients was carried out just for SD5060 airfoil as Selig *et al.* (1989) measurement data close to two Reynolds number analysed were available only for this airfoil. The comparison of the aerodynamic coefficients for SD5060 airfoil at two Reynolds number are shown in Fig. 7.18 and Fig. 7.19. At the lower Reynolds number ($Re = 6 \times 10^4$), the computed lift coefficient was found to be in good agreement with the measurements (Selig *et al.*, 1989) but the drag polar was overpredicted which resulted in the underprediction of the (L/D) for all the angles of attack. The discrepancy observed with the measurement data for the drag polar curve and (L/D) curve was found to reduce significantly when the Reynolds number was increased to $Re = 2 \times 10^5$. The influence of the airfoil thickness on the aerodynamic performance at the two Reynolds number are given in Fig. 7.20 and Fig. 7.21. In the the lower Reynolds number case (Fig. 7.20), it is observed that increasing the thickness resulted in an increase in the lift coefficient and this increase in C_L was found to reduce with increase in angle of attack. However the variation of drag polar was not influenced to a great extent by the airfoil thickness. This difference in the C_L obtained by the two airfoils is primarily reflected in the lift to drag ratio curve where the airfoil with higher thickness is produced higher L/D at lower angles of attack ($\alpha \leq 6^\circ$). The slope of the pitching moment curve of both the airfoils is quite similar with the airfoil with higher thickness having lower value. On the other hand, when the Reynolds number was increased to 2×10^5 , the effect of the thickness was prominent only at higher angles of attack for the lift coefficient, drag polar and L/D ratio. However pitching moment slope and trend obtained by two airfoils was found to be different. The pitching

moment curve for airfoils with higher thickness was found to be linear with positive slope whereas for the airfoils with lower thickness the C_M curve was nonlinear with a negative slope upto $\alpha = 8^\circ$ which is close to the stall angle of attack.

The influence of thickness at the two Reynolds number on key aerodynamic parameters (stall angle and $(L/D)_{max}$) are compared with available measurement data (Selig *et al.*, 1989) and shown in Fig. 7.22. Similar to the camber effect, the computed stall angle remains constant for both the airfoils with different thickness but the stall angle was found to be delayed as the Reynolds number was increased. Increasing the thickness showed a marginal increase in the $(L/D)_{max}$ at higher Re . The predicted stall angle and $(L/D)_{max}$ at $Re = 2 \times 10^5$ is found to be in a good agreement with the measurement data.

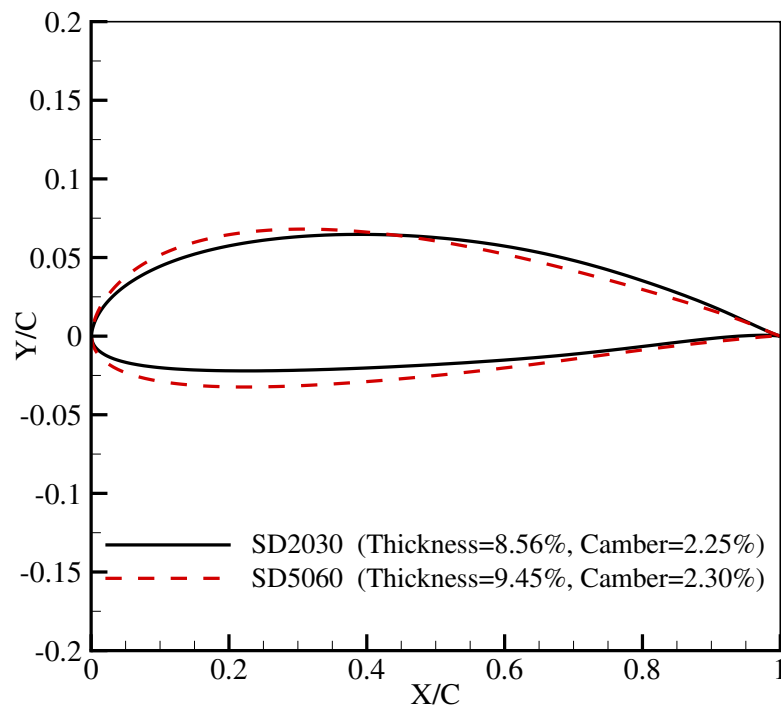
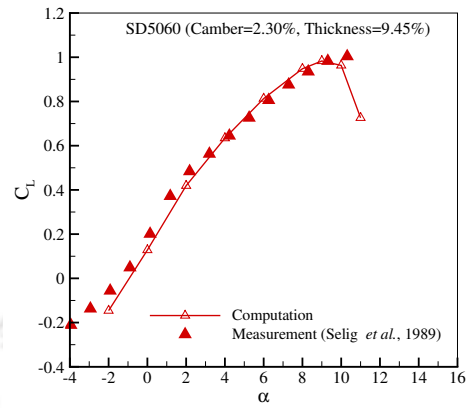
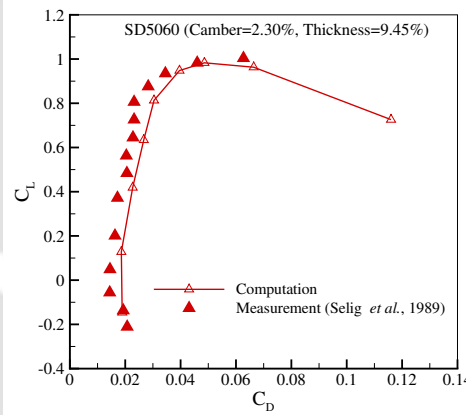


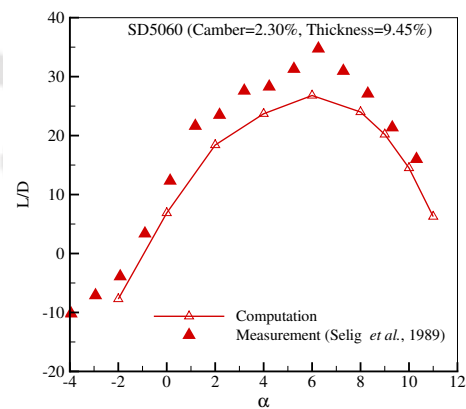
Figure 7.17: SD airfoil geometries having different thickness



(a) Lift coefficient



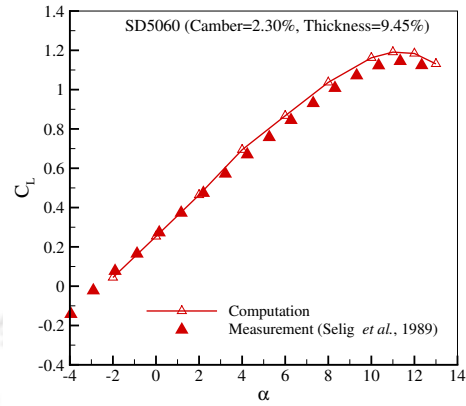
(b) Drag polar



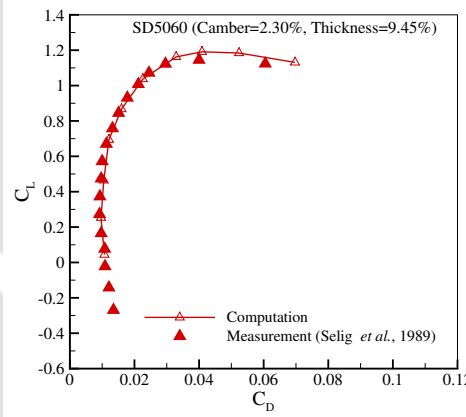
(c) Lift to drag ratio

Figure 7.18: Validation of the aerodynamic performance of SD5060 airfoil at lower Re

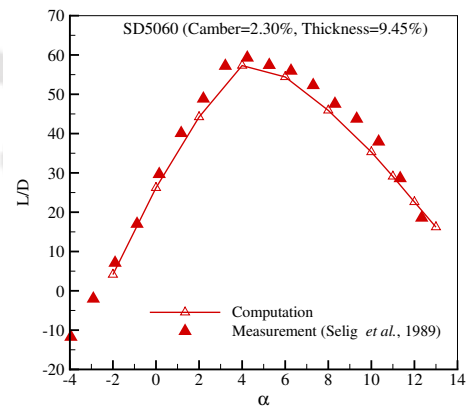
(Computation : $Re = 6 \times 10^4$, Measurement : $Re = 6 \times 10^4$)



(a) Lift coefficient



(b) Drag polar



(c) Lift to drag ratio

Figure 7.19: Validation of the aerodynamic performance of SD5060 airfoil at higher Re

(Computation : $Re = 2 \times 10^5$, Measurement : $Re = 2.025 \times 10^5$)

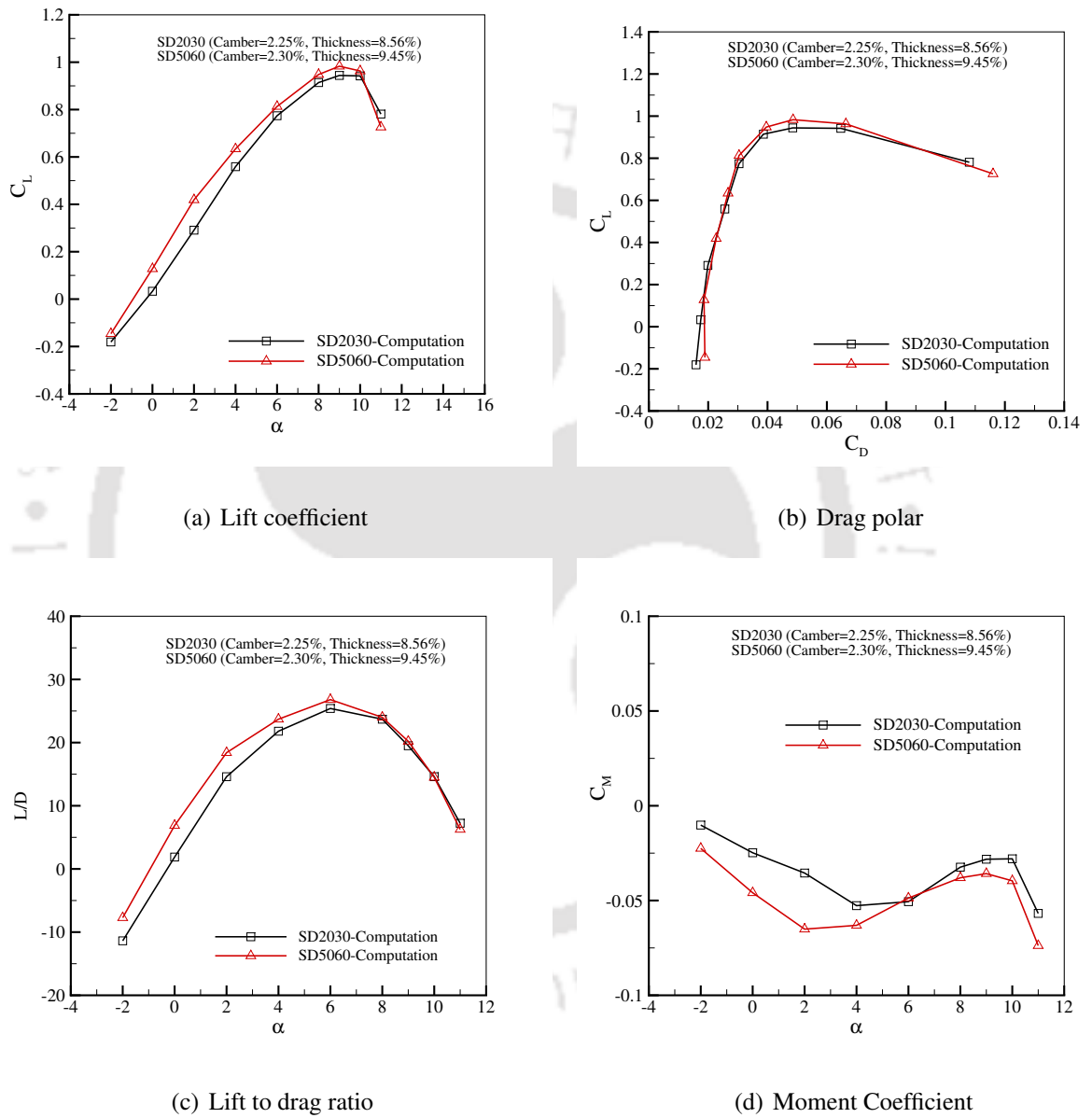


Figure 7.20: Influence of airfoil thickness on aerodynamic performance at $Re = 6 \times 10^4$

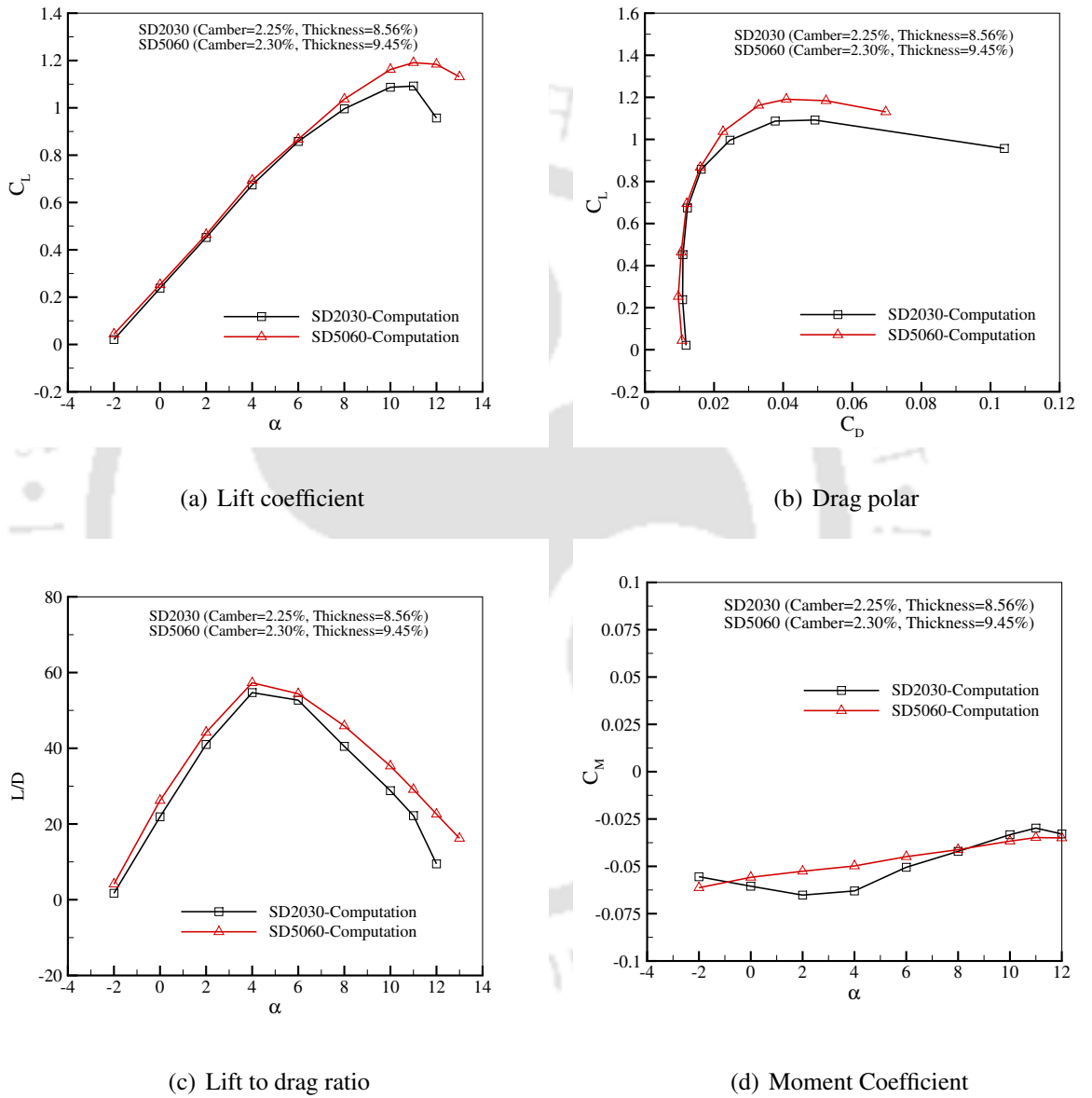
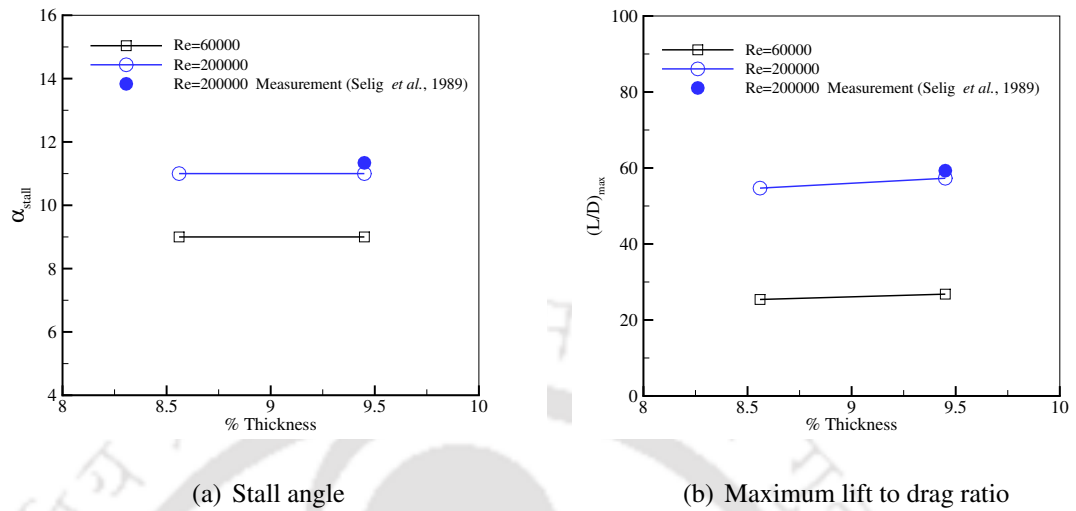


Figure 7.21: Influence of airfoil thickness on aerodynamic performance at $Re = 2 \times 10^5$

Figure 7.22: Influence of thickness on aerodynamic parameters at different Re

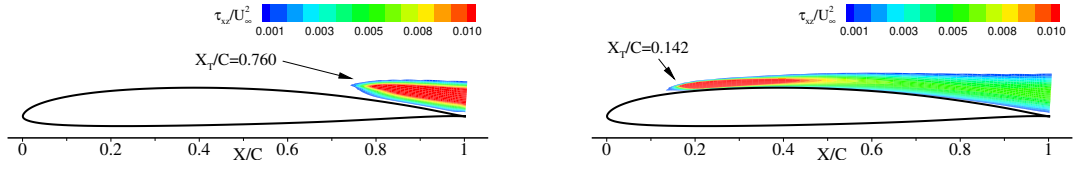
7.3.2 Transition characteristics

The effect of thickness on transition phenomenon which is characterised by transition onset location and length of laminar separation bubble (LSB) is studied by comparing the Reynolds shear stress contours and surface streamlines obtained at the two Reynolds number for two typical angles of attack. The Reynolds shear stress contours (Fig. 7.23) and surface streamlines (Fig. 7.24) obtained for $Re = 6 \times 10^4$ at angle of attack 4° and 8° clearly indicate the transition onset and the formation of the separation bubble for the two airfoils with varying thickness. It is evident from the Reynolds shear stress contours at $Re = 6 \times 10^4$ that the transition onset location shifts upstream with increase in thickness for the lower angle of attack ($\alpha = 4^\circ$). Whereas, at the higher angle of attack ($\alpha = 8^\circ$) increasing the thickness has resulted in a downstream shift of the transition onset leading to a more laminar region for the airfoil with higher thickness. The transition onset location as expected moves upstream for both the airfoils when the angle of attack is increased from 4° to 8° . The airfoil thickness is found an influence both on the laminar separation point as well as on the length of the LSB for both the Reynolds number as shown in the streamlines plot (Fig. 7.24). Similar to the effect of airfoil camber, It is clearly depicted in the streamlines plot for $\alpha = 4^\circ$ that increasing the airfoil

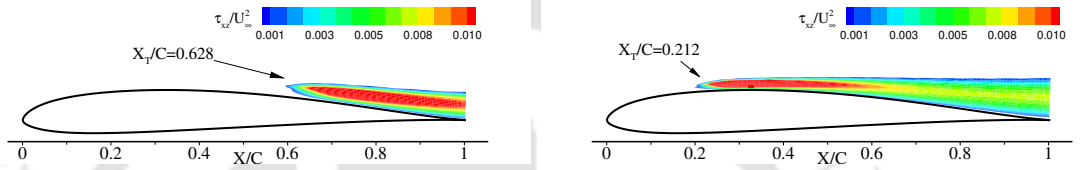
thickness has shifted the laminar separation point upstream which has lead to a increase in the length of the LSB as the turbulent reattachment point remained almost constant. It was observed that even at $\alpha = 8^\circ$ the length of LSB increased with increase in the airfoil thickness. However this increase in the length of LSB was because both the laminar separation point and turbulent reattachment moves downstream with increase in the airfoil thickness.

Even for this case it was found that at higher Reynolds number ($Re = 2 \times 10^5$) case, the turbulence level reduced which is indicated by lower level of the Reynolds shear stress for both the angles of attack (Fig. 7.25). The transition onset as expected for both the airfoils at $Re = 2 \times 10^5$ is predicted earlier than compared to the lower Re case. Similar to low Reynolds number case, the transition onset shifts upstream for $\alpha = 4^\circ$ and shifts downstream for $\alpha = 8^\circ$ with increase in airfoil thickness. The surface streamlines (Fig. 7.26) at $Re = 2 \times 10^5$ clearly point out the formation of a weaker laminar separation bubble as compared to the lower Re case. This similar features was observed in all the Reynolds number effect study carried out. Influence of airfoil thickness on the inception of the laminar separation point and turbulent reattachment point as well as the length of the separation bubble at $Re = 2 \times 10^5$ is similar to that observed for the lower Re case.

The overall influence of airfoil thickness on the transition onset location and length of LSB at $Re = 6 \times 10^4$ and $Re = 2 \times 10^5$ is shown in Fig. 7.27 and Fig. 7.28 respectively. The effect of airfoil thickness on variation of the transition onset with angles of attack indicate that increasing the thickness moves the transition onset upstream upto $\alpha = 6^\circ$ leading to more turbulent flow but beyond this angle the turbulent region for the higher thickness airfoil is lesser as the transition onset moves downstream when compared to the thinner airfoil. This trend was also observed in the Reynolds shear stress contours. Similar to the streamlines plot, the length of the LSB obtained for the thicker airfoil is found to be larger for angles of attack except $\alpha = 6^\circ$ and this difference in the length of LSB being more at lower angles of attack.

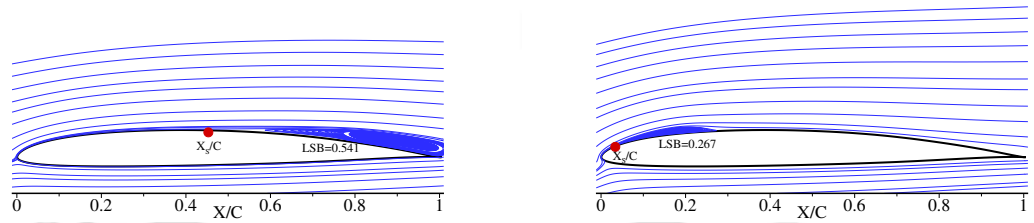


(a) SD2030 Airfoil (Thickness=8.56%, Camber=2.25%)

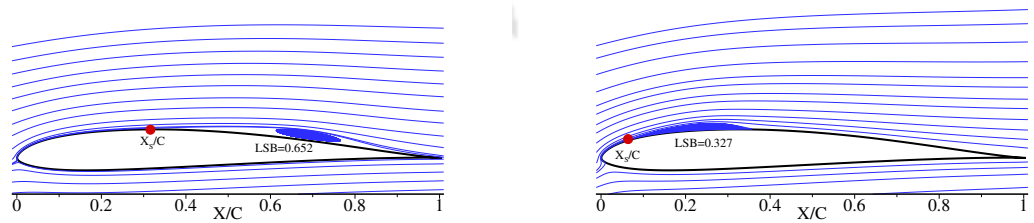


(b) SD5060 Airfoil (Thickness=9.45%, Camber=2.30%)

Figure 7.23: Influence of thickness on Reynolds shear stress at $Re = 6 \times 10^4$
(Left figure : $\alpha = 4^\circ$, Right figure : $\alpha = 8^\circ$)

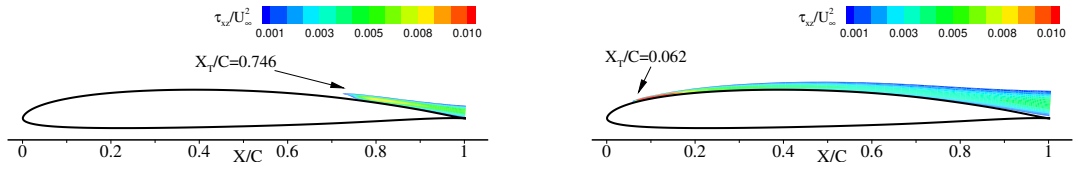


(a) SD2030 Airfoil (Thickness=8.56%, Camber=2.25%)

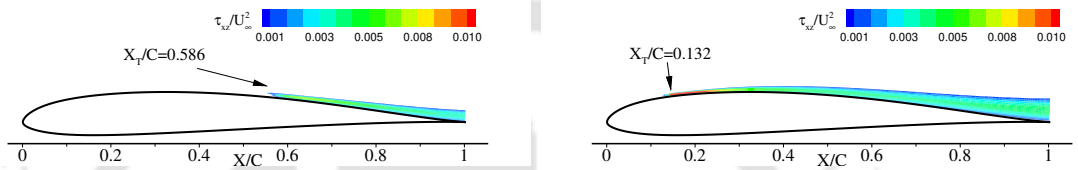


(b) SD5060 Airfoil (Thickness=9.45%, Camber=2.30%)

Figure 7.24: Influence of thickness on surface streamlines at $Re = 6 \times 10^4$
(Left figure : $\alpha = 4^\circ$, Right figure : $\alpha = 8^\circ$)

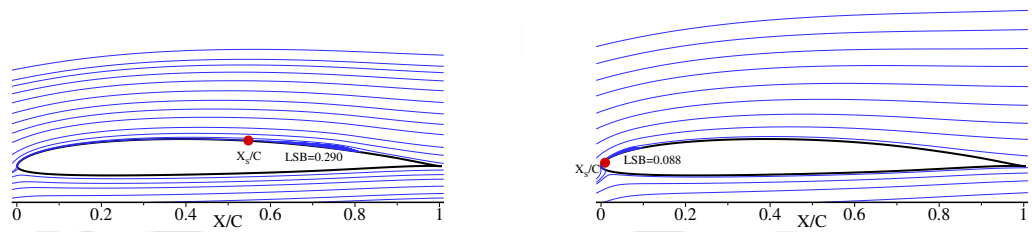


(a) SD2030 Airfoil (Thickness=8.56%, Camber=2.25%)

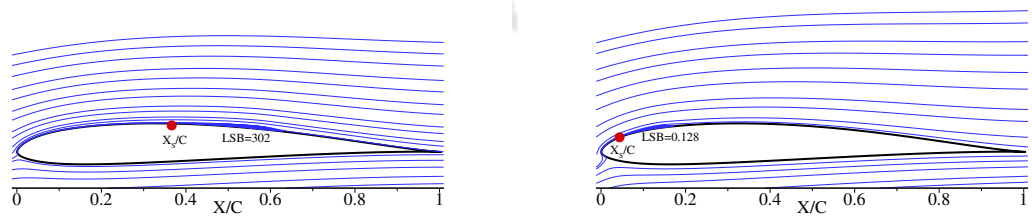


(b) SD5060 Airfoil (Thickness=9.45%, Camber=2.30%)

Figure 7.25: Influence of thickness on Reynolds shear stress at $Re = 2 \times 10^5$
(Left figure : $\alpha = 4^\circ$, Right figure : $\alpha = 8^\circ$)



(a) SD2030 Airfoil (Thickness=8.56%, Camber=2.25%)



(b) SD5060 Airfoil (Thickness=9.45%, Camber=2.30%)

Figure 7.26: Influence of thickness on surface streamlines at $Re = 2 \times 10^5$
(Left figure : $\alpha = 4^\circ$, Right figure : $\alpha = 8^\circ$)

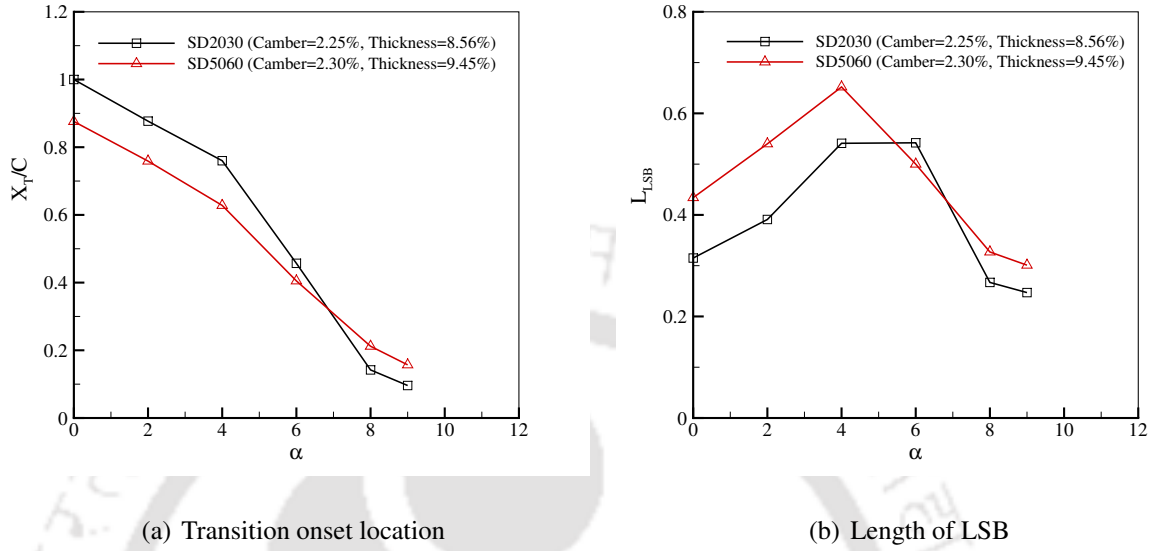


Figure 7.27: Influence of airfoil thickness on transition characteristics at $Re = 6 \times 10^4$

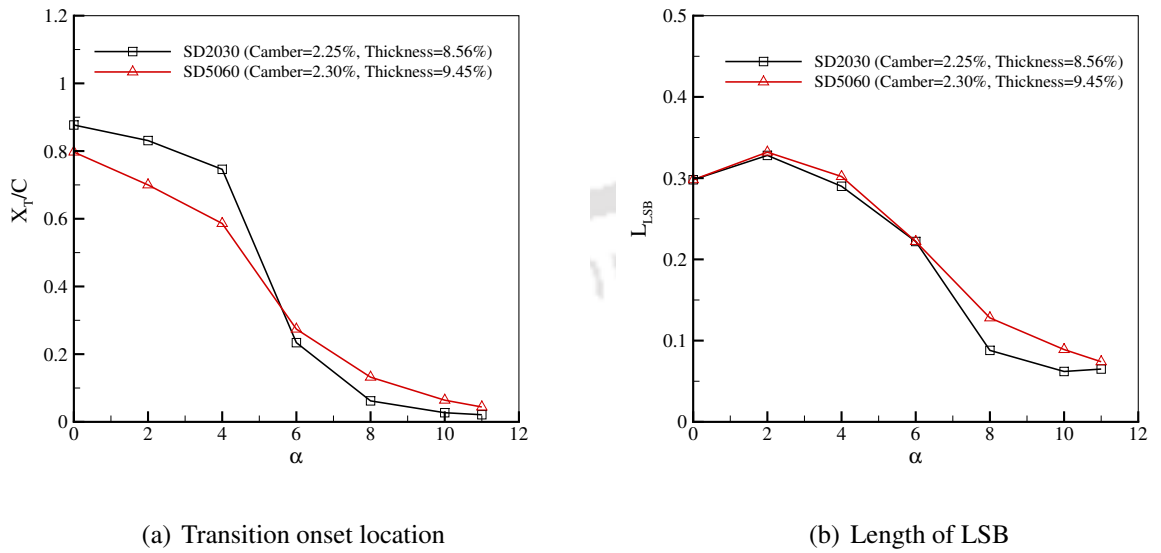


Figure 7.28: Influence of airfoil thickness on transition characteristics at $Re = 2 \times 10^5$

7.4 Summary

The effect of camber and thickness for the low Re SD airfoil series was analysed at two Reynolds number in the low Re regime and the aerodynamic data was validated against the available measurements. The aerodynamic data in general was found to be in good agreement with measurement data for most of the cases analysed. The discrepancies obtained with the measurements was found to reduce when the Reynolds number was increased. The present study showed that, increasing the camber improved the aerodynamic characteristics in terms of C_{Lmax} and $(L/D)_{max}$ however stall angle was not effected by the airfoil camber. It is further observed that at both the Reynolds number, the flow remained laminar over a longer region for the airfoil with highest camber particularly at higher angles of attack. In the low Re case, the length of LSB was not greatly influenced by the airfoil camber at lower angles of attack but beyond $\alpha = 4^\circ$ the length of the LSB was quite larger compared to the other two airfoils. On the other hand, when the Re was increased to 2×10^5 the effect of camber on the length LSB was not very significant for all angles of attack but in general it was observed that increase in camber increased the length of LSB. The airfoil camber was found to influence the transition features as well with the airfoil having the highest camber was found to have longer laminar region and larger laminar separation bubble particularly at higher angle of attack. The effect of varying the airfoil thickness on the aerodynamic coefficients was not as prominent as observed in the effect of varying camber as the difference in the thickness was small which was about 10%. In general it was found that increasing the thickness increased the lift coefficient to some extent and the lift to drag ratio. The thicker airfoil was found to have larger turbulent region at lower angles of attack whereas the thinner airfoil was found to be more turbulent at angles close to the stall angle. Increasing the Reynolds number enhanced the C_{Lmax} , $(L/D)_{max}$ and delayed the stall angle for the airfoils considered for this study.

Chapter 8

Flow Past Low Aspect Ratio Wings

8.1 Background

This chapter discusses the effect of aspect ratio on the aerodynamic and transition characteristics of the SD7003 rectangular wing at $Re = 6 \times 10^4$. This study was carried out using the $\gamma - Re_\theta$ SST transition model for three different semi aspect ratio upto the stall angle of attack (α). This chapter also discusses the effect of Reynolds number in the low Re regime on the performance of low aspect ratio wing.

8.2 Computational Details

The in-house grid generation code (Fathima *et al.*, 1994) was used to generate the quasi three-dimensional H-H grid by stacking 2D airfoil H-grid at each of the wing section along the span direction. The H-H grid topology was used since grid generation and prescription of flow boundary conditions are simple and straight forward. Similar grid topology was used by Hsiao and Pauley (Hsiao and Pauley, 1998) for simulating flow past rectangular wing. This quasi-3D grid generation approach was successfully used to simulate the flow past the Golden Hawk MAV wing using the in-house code 3D-PURLES (Subrahmanya and Rajani, 2018). The present simulations have been carried out for half the wing by specifying a symmetry boundary condition. The H-H grid topology

having a grid size of $590 \times 93 \times 178$, $590 \times 109 \times 178$ and $590 \times 129 \times 178$ was generated for the wing with semi aspect ratio (sAR) of 0.25, 0.5 and 1 respectively as shown in Fig 8.1. The grids are stretched appropriately towards the wing surface in order to obtain the near wall y^+ to be less than 1. The 590 grid points along the streamwise direction is distributed such that the wing upper and lower surface are covered by 200 grid points with the resolution near the leading and trailing edge to be around $2 \times 10^{-3}C$. The resolution at the tip was around $10^{-4}C$. The distance from the wing tip to the farfield is assumed to be 4 times the wing sAR leading to a different number of grid points along span direction for the three wings. The computational domain and boundary conditions used for the present simulations are shown in Fig. 8.2. The computational domain was divided into 12 blocks so as to run the flow solver in parallel. The flow parameters like the discretisation scheme, freestream turbulence intensity, and eddy viscosity ratio used for this 3D computations are same as that used for the airfoil simulations 5.2.

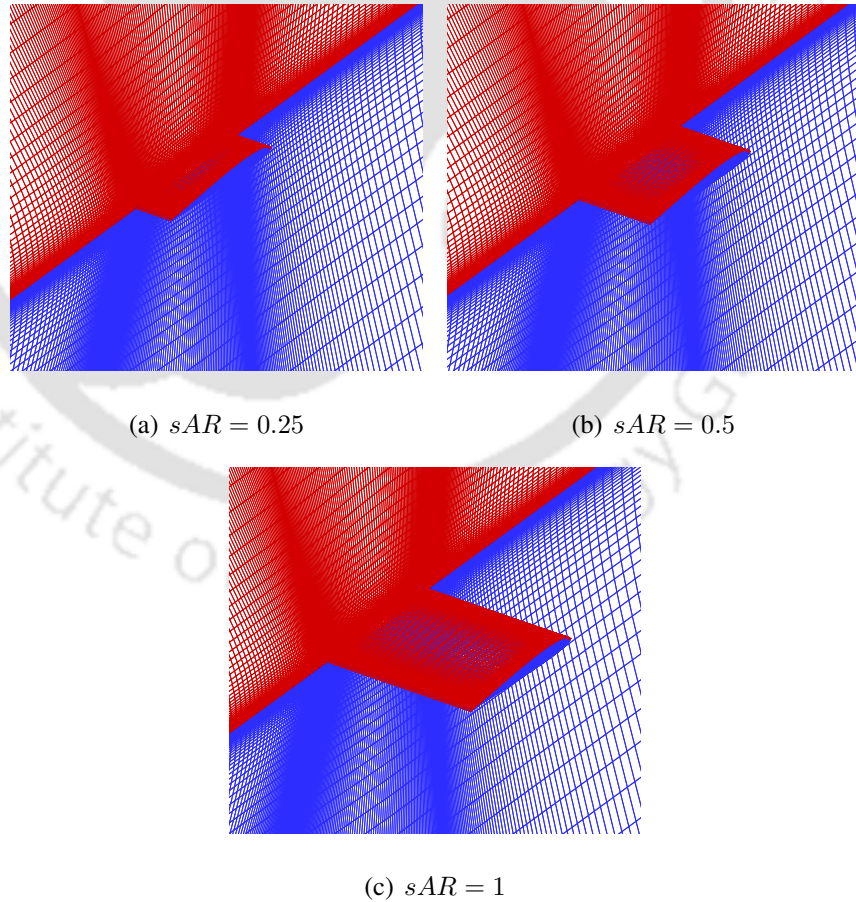


Figure 8.1: H-H Grid showing the symmetry plane and wing surface

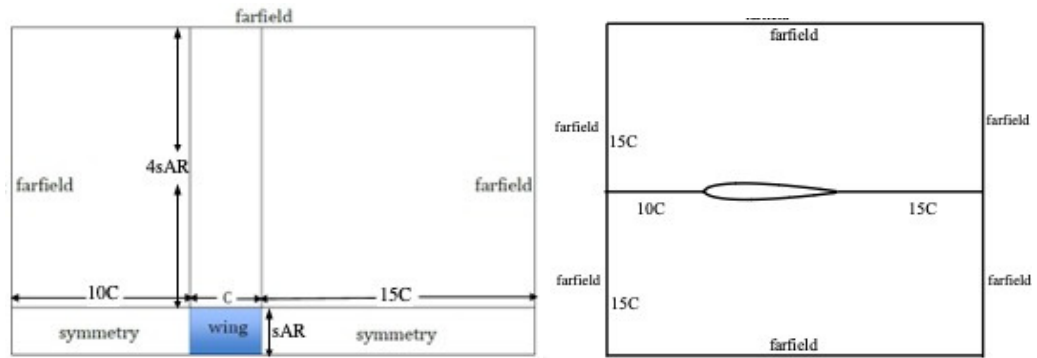


Figure 8.2: Boundary conditions used in the present simulations

8.3 Effect of Wing Aspect Ratio on the Flow Characteristics

Numerical simulations using the $\gamma - Re_\theta$ SST transition model were carried out at $Re = 6 \times 10^4$ in order to understand the effect of wing aspect ratio on the transition and aerodynamic characteristics. These simulations were carried for three rectangular wings with different semi aspect ratios ($sAR = 0.25, 0.5$ and 1) whose cross-section is the SD7003 airfoil. In this study, the transition onset location is obtained based on the shear stress threshold criterion as suggested by Ol *et al.* (2005). The Reynolds shear stress contours on the wing surface along with the sectional view of the wing surface are shown in Fig. 8.3 to Fig. 8.6 at four angles of attack. The two dimensional sectional view of the contours at the root, mid and tip sections are shown to obtain a clear view of distribution of the contours and also to determine the transition onset location. These figures clearly shows that the transition onset along the wing span is almost constant from the root to the region close to the tip section. The contours at the tip section are observed to be quite different which may be due to the formation of the tip vortices. The transition onset location (X_T/C) at the tip is observed to be much delayed (occurring downstream) compared to the root and mid section especially at lower angles of attack. At $\alpha = 0^\circ$ (Fig. 8.3), the flow is almost laminar (quantified by very negligible value of the Reynolds shear stress) at the wing tip for all the three wings and further it is observed that the Reynolds shear stress contours are almost similar. The effect of the wing aspect ratio on the variation of the Reynolds shear stress contours along the wing span is observed to be more prominent with increase in angle of attack. At all the angles of attack, it is observed that the

level of the turbulent intensity (quantified by the value of Reynolds shear stress) is much less at wing tip as compared to the root section for all the three wings. The wing with lowest aspect ratio is found to be more laminar compared to the other two wings. In other words, the transition onset moves upstream leading to a lesser laminar region with increase in the wing aspect ratio. It is further observed that the length of transition zone is also found to reduce (*i.e.* the flow become turbulent faster which is indicated by higher value of Reynolds shear stress contour) with increase in sAR . At $\alpha = 16^\circ$ (Fig. 8.6) the flow is almost fully turbulent for $sAR=0.5$ and 1 with the transition onset occurring very close to the leading edge and having a very high turbulence level. The transition onset, as expected move upstream with angle of attack for the three wing making the flow to be more turbulent at higher angles of attack. The influence of the wing span on the transition onset location at different angles of attack is shown in Fig. 8.7. The wing with lowest aspect ratio ($sAR = 0.25$) is found to have a delayed transition onset (longer laminar region) at all the angles of attack as compared to other two wings. The volume streamlines for the wings at four different angles of attack are shown from Fig. 8.8 to Fig. 8.11. The streamlines shows the formation of the tip vortex for all the three wings. These figures clearly depict that the formation of the tip vortices in terms of its strength and onset location is not greatly influenced by the wing aspect ratio. However the strength of the tip vortices is found to increase with angle of attack with its onset location moving upstream which is an expected phenomenon.

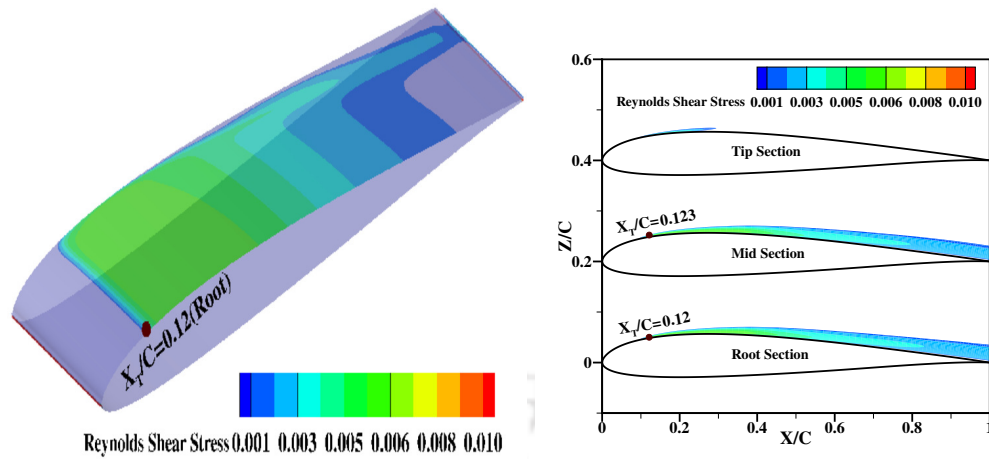
The effect of the wing aspect ratio on the aerodynamic characteristics is shown in Fig. 8.12. The influence of the wing span on the aerodynamic coefficients is observed to be quite insignificant for $\alpha \leq 0^\circ$. The difference especially in the lift coefficient obtained by the three wings are found to be enhanced with angle of attack. The lift coefficient (C_L) curve for the wing having the lowest sAR (Fig. 8.12(a)) is observed to be nonlinear especially at the lower angles of attack and this nonlinearity disappears with increase in aspect ratio. The wing with the lowest aspect ratio though has lesser C_L is observed to have a soft and delayed stall angle around $\alpha = 42^\circ$. On the other hand, for the wing with higher aspect ratio an early and sharp stall is observed with the stall angle reducing with increase in aspect ratio. The stall around $\alpha = 22^\circ$ and $\alpha = 16^\circ$ are obtained for the wing with aspect ratio 0.5 and 1 respectively. It is further observed that the maximum C_L reduces with increase in the wing aspect ratio but the C_L before the stall angle is highest for the wing having

highest aspect ratio. The variation of drag coefficient C_D (Fig. 8.12(b)) for the three wing with different sAR are almost identical with the wing with lowest aspect ratio having slightly lower drag especially beyond $\alpha = 4^\circ$. This lower drag obtained by this wing may be attributed to reason that the flow over this wing has a longer laminar region compared to the other two wings. The lift to drag ratio L/D (Fig. 8.12(c)) is observed to reduce with the reduction in the wing aspect ratio however the maximum L/D is found to occurs at the same angle of attack ($\alpha = 8^\circ$) for all the three wings. The variation of pitching moment coefficient (Fig. 8.12(d)) with angle of attack for the lowest sAR is observed to be nonlinear and having a negative slope. It is further evident from this figure that increasing the wing aspect ratio has resulted in an almost linear C_M curve with its slope tending to become positive. The wing with the lowest aspect ratio ($sAR = 0.25$) though having lower L/D ratio may be preferred for MAV applications as this wing has much delayed stall angle, higher maximum C_L , lower drag coefficient and negative C_M slope.

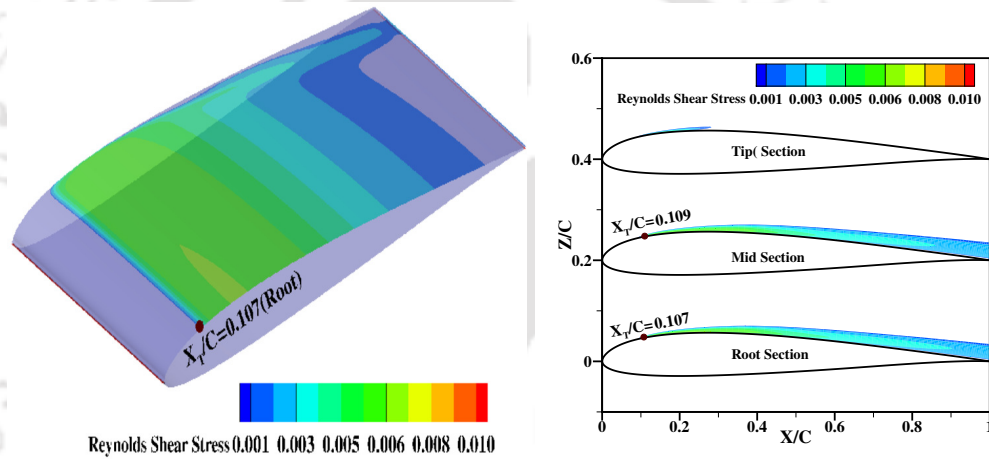
In order to further quantify the results obtained, the surface pressure and skin-friction coefficients obtained at the root and tip sections of the wings are compared and shown in Fig. 8.13 to Fig. 8.16 for different angles of attack. The C_p and C_f curves obtained at tip section are significantly different from that obtained at the root section. The C_p and C_f plot at the mid section is not shown here as it was found to be identical to the root section. The variation of C_p and C_f obtained at $\alpha = 0^\circ$ for the three wings are almost identical both at the root and the tip sections. The difference in the C_p and C_f obtained by wings is found to increase with angle of attack. This nature is also reflected in the aerodynamic characteristics of the wings. The area under the C_p curves quantifies the coefficient of lift. These C_p plots at different angles of attack indicates that the wing with higher aspect ratio is found to have higher lift coefficient. The surface skin-friction coefficient curve also depicts the transition nature which the flow undergoes. The C_f plot is widely used to quantify the transition behaviour of flow over a flat plate (Coupland, 1990; Kulkarni *et al.*, 2019). The initial reduction observed in the C_f plot indicates the laminar region on the wing surface, the point from where the C_f is found to increase denotes the transition onset location, the region of the increase in C_f indicates the transition zone. The transition onset indicated by the C_f curve is quite similar to that obtained by the Reynolds shear stress contours for all the three wings. The C_f obtained at tip section for $\alpha = 0^\circ$ is observed to be almost laminar of all the three wings which corroborate

the findings of the Reynolds shear stress contours. Similar to that observed in the Reynolds shear stress contours, the transition zone is found reduce with increase in the wing aspect ratio and this reduction is found to enhance with angle of attack. These C_f plots at different angles of attack clearly indicates the shift in the transition onset location upstream and reduction in the length of the transition zone for all the three wings.

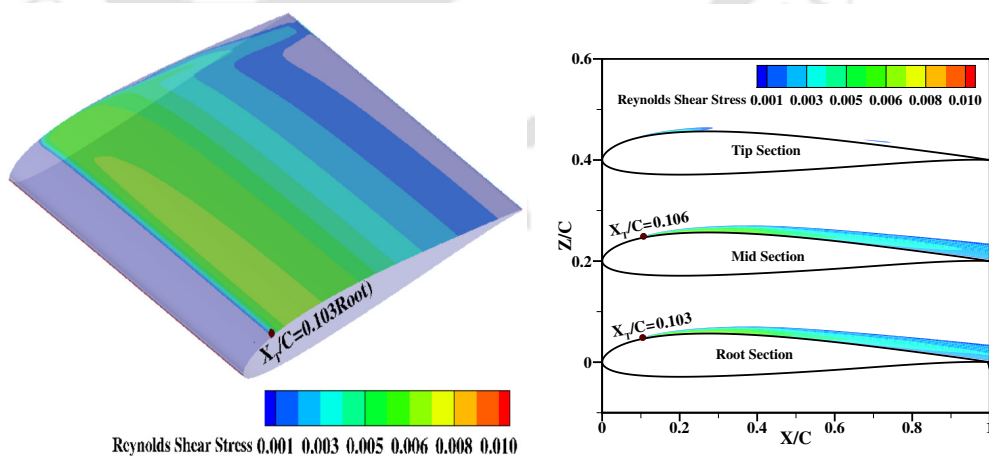




(a) $sAR = 0.25$

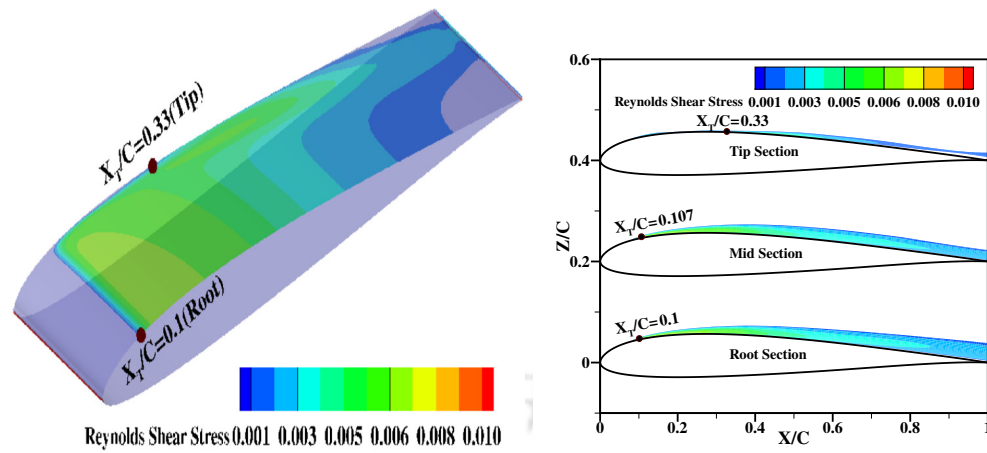


(b) $sAR = 0.5$

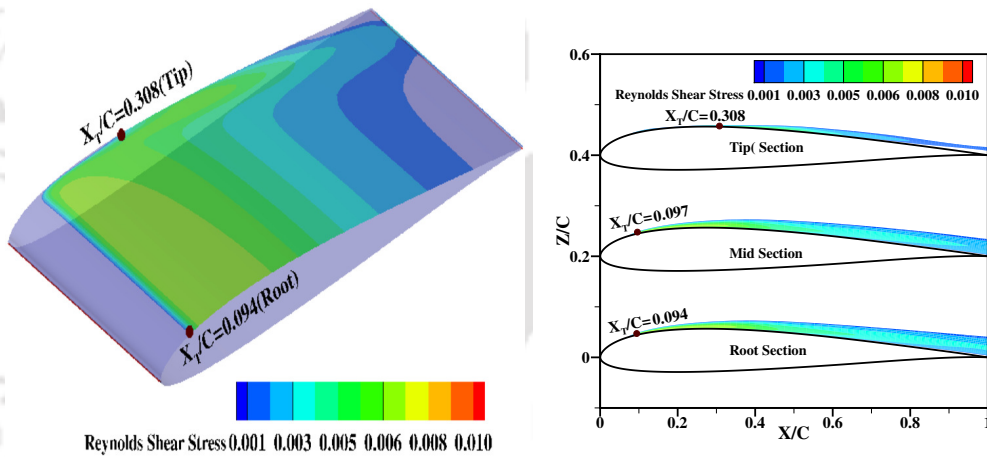


(c) $sAR = 1$

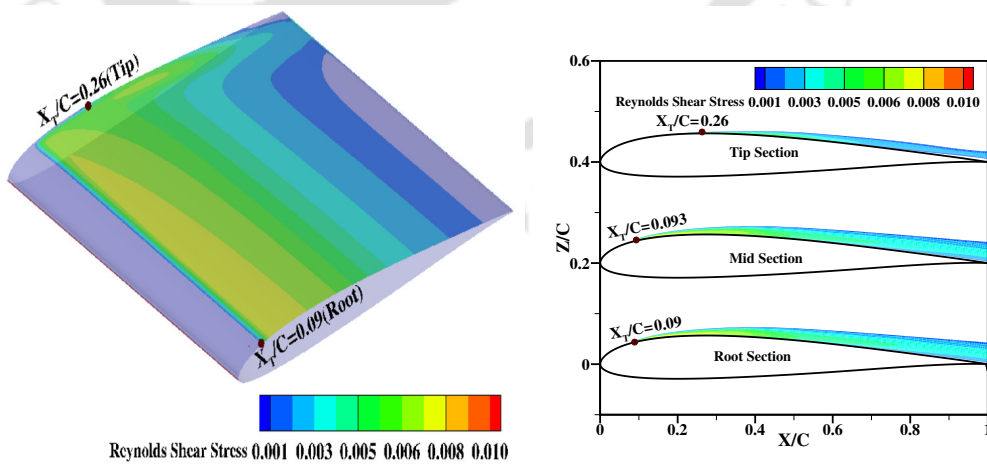
Figure 8.3: Reynolds shear stress contours for different wing spans at $\alpha = 0^\circ$
(Left figure : Surface contours, Right figure : Contours on the 2D sections)



(a) $sAR = 0.25$

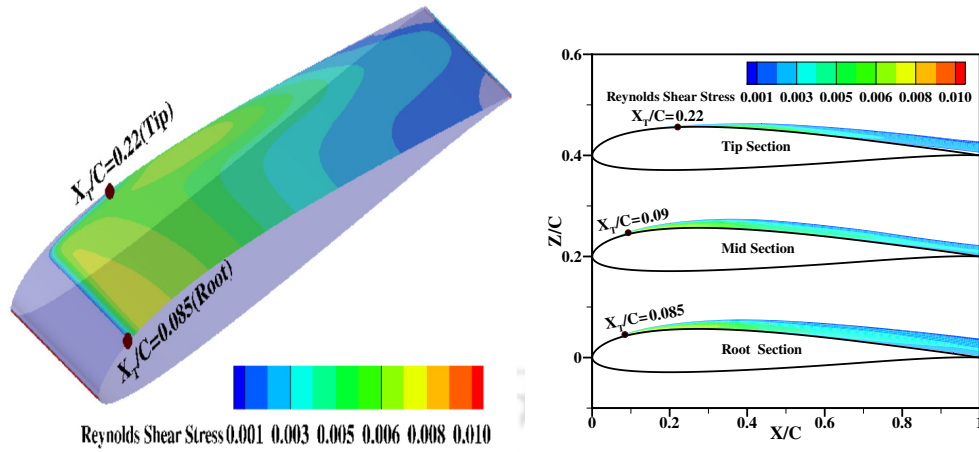


(b) $sAR = 0.5$

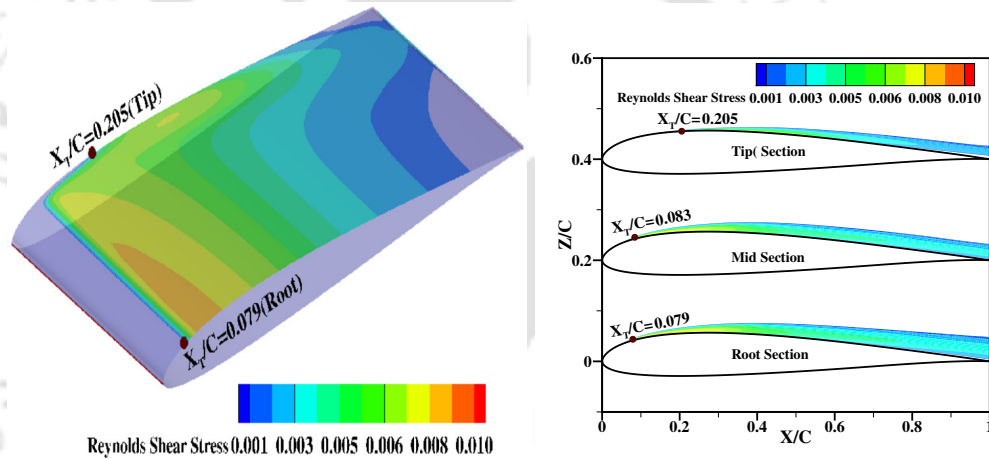


(c) $sAR = 1$

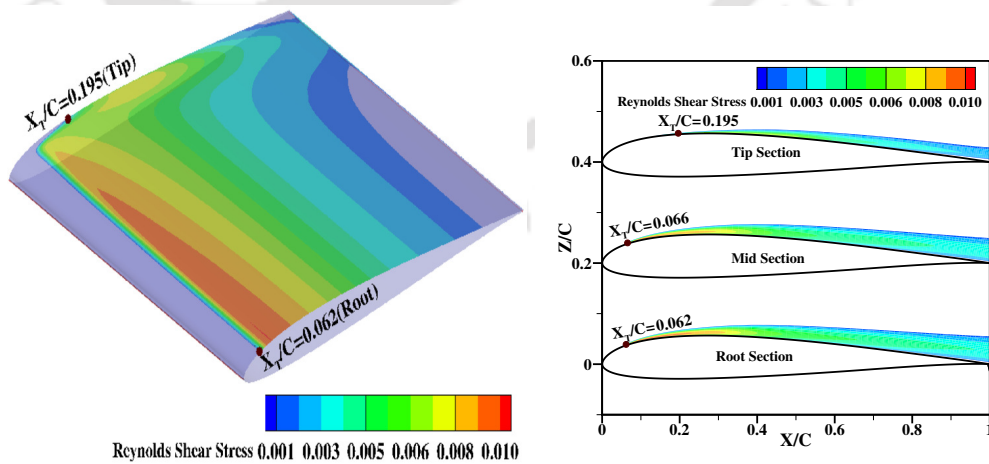
Figure 8.4: Reynolds shear stress contours for different wing spans at $\alpha = 4^\circ$
(Left figure : Surface contours, Right figure : Contours on the 2D sections)



(a) $sAR = 0.25$

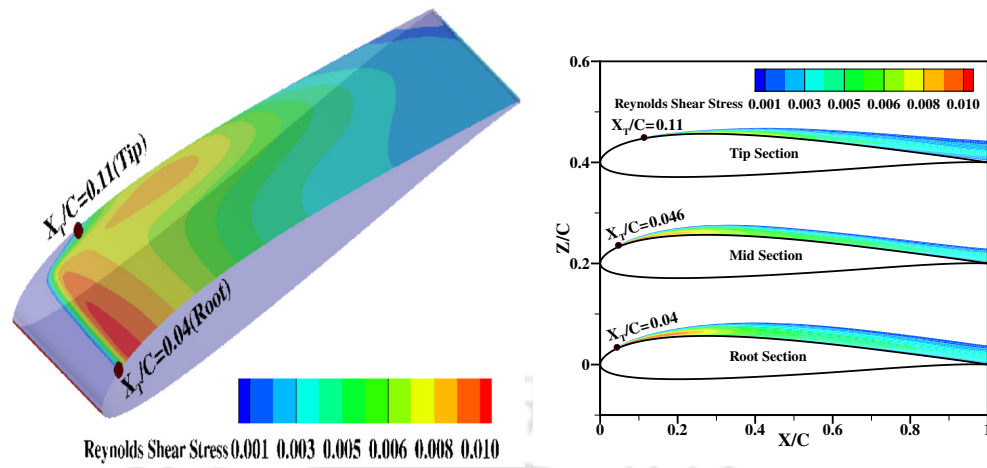


(b) $sAR = 0.5$

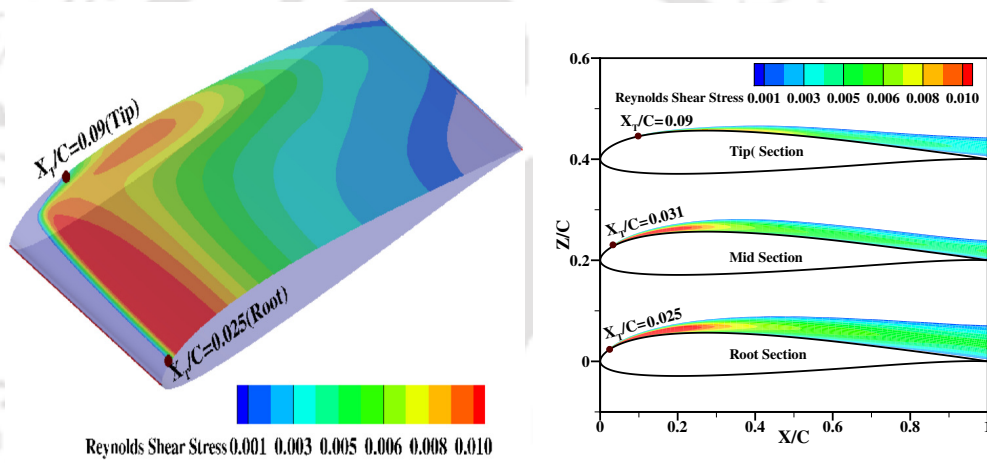


(c) $sAR = 1$

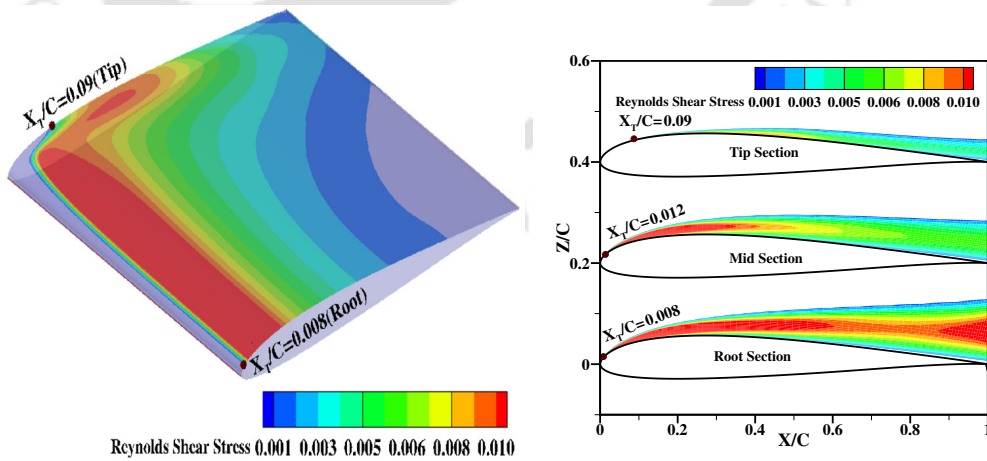
Figure 8.5: Reynolds shear stress contours for different wing spans at $\alpha = 8^\circ$
(Left figure : Surface contours, Right figure : Contours on the 2D sections)



(a) $sAR = 0.25$



(b) $sAR = 0.5$



(c) $sAR = 1$

Figure 8.6: Reynolds shear stress contours for different wing spans at $\alpha = 16^\circ$
(Left figure : Surface contours, Right figure : Contours on the 2D sections)

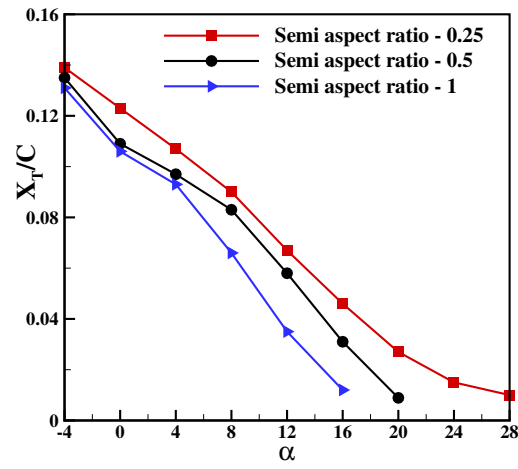


Figure 8.7: Effect of wing-span on the transition onset location at $Re = 6 \times 10^4$

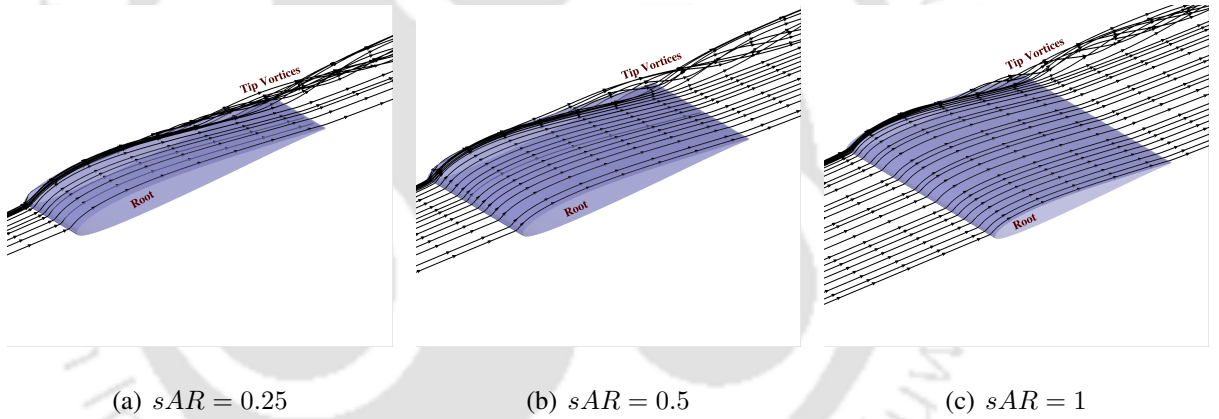


Figure 8.8: Volume streamlines for different wing span at $Re = 6 \times 10^4$ and $\alpha = 0^\circ$

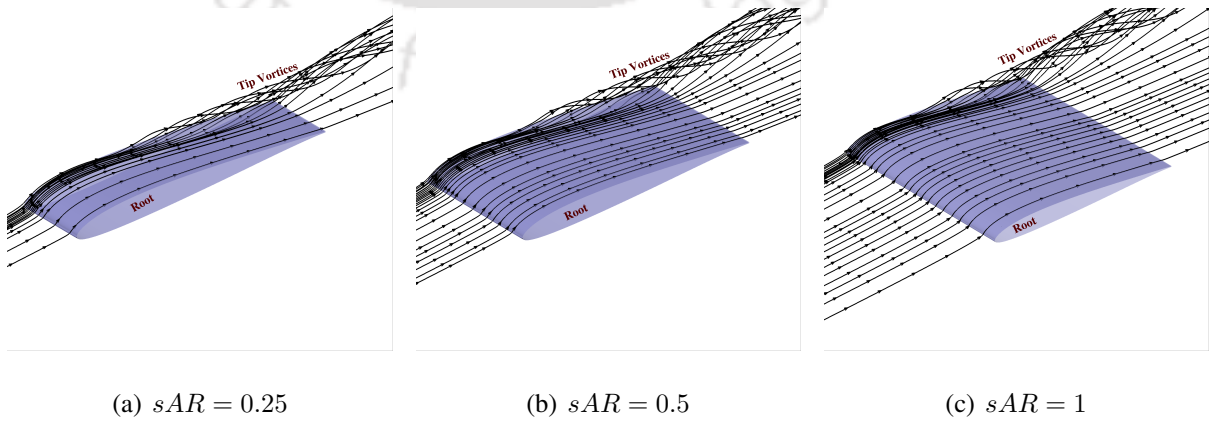
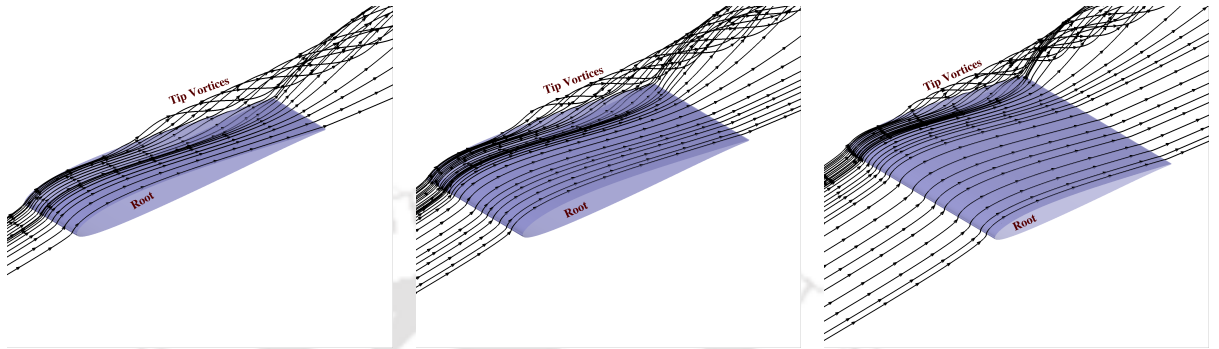


Figure 8.9: Volume streamlines for different wing span at $Re = 6 \times 10^4$ and $\alpha = 4^\circ$

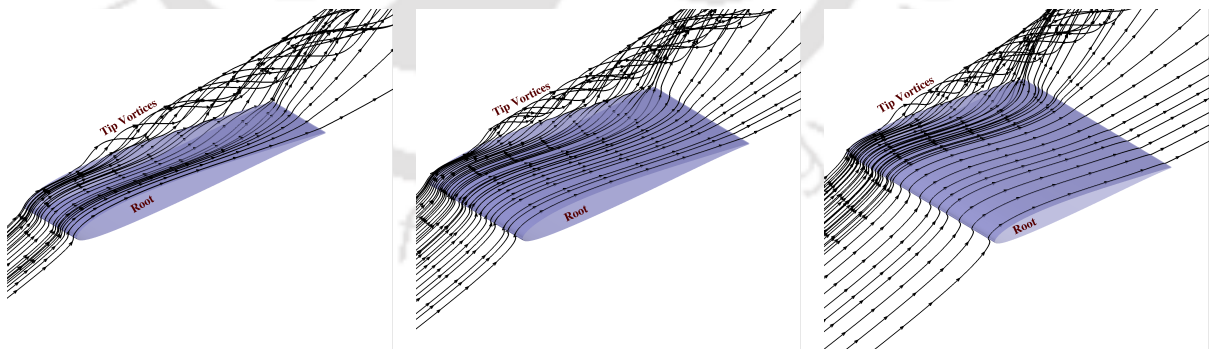


(a) $sAR = 0.25$

(b) $sAR = 0.5$

(c) $sAR = 1$

Figure 8.10: Volume streamlines for different wing span at $Re = 6 \times 10^4$ and $\alpha = 8^\circ$

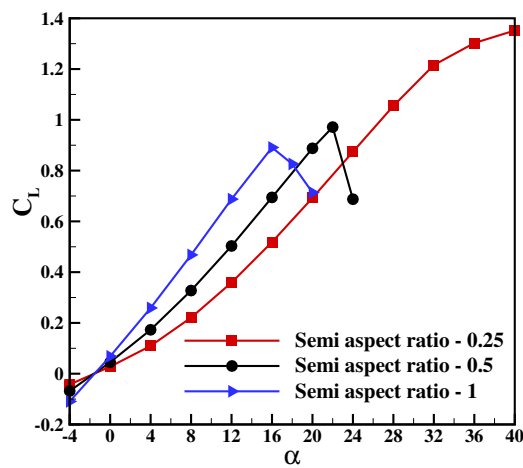


(a) $sAR = 0.25$

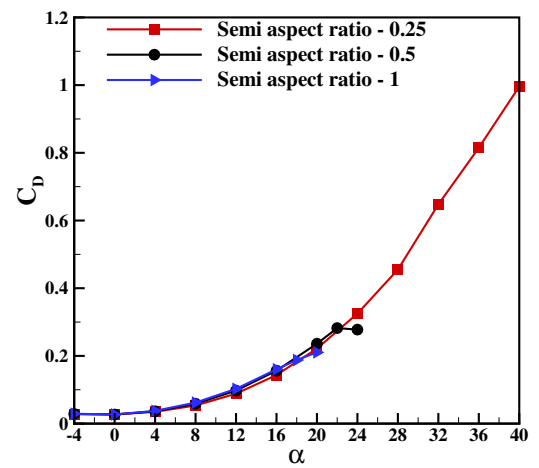
(b) $sAR = 0.5$

(c) $sAR = 1$

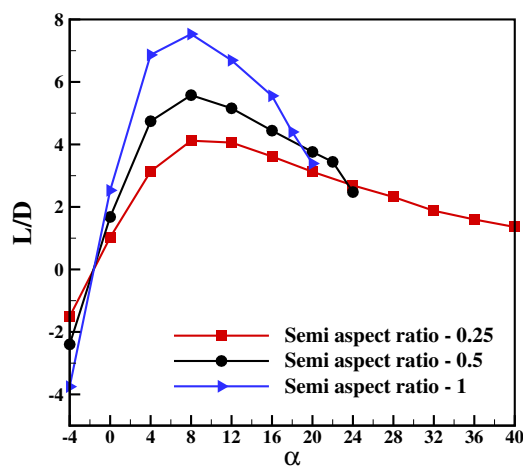
Figure 8.11: Volume streamlines for different wing span at $Re = 6 \times 10^4$ and $\alpha = 16^\circ$



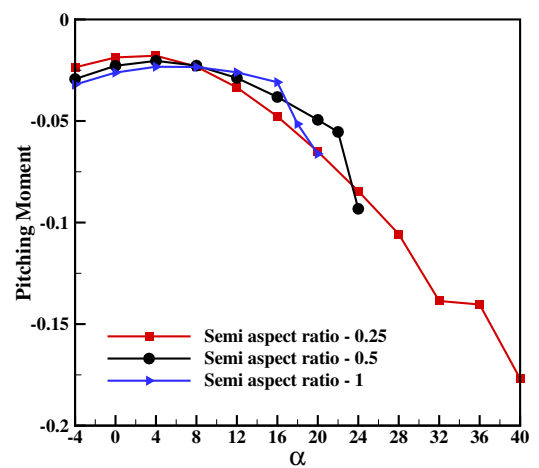
(a) Coefficient Lift



(b) Coefficient Drag

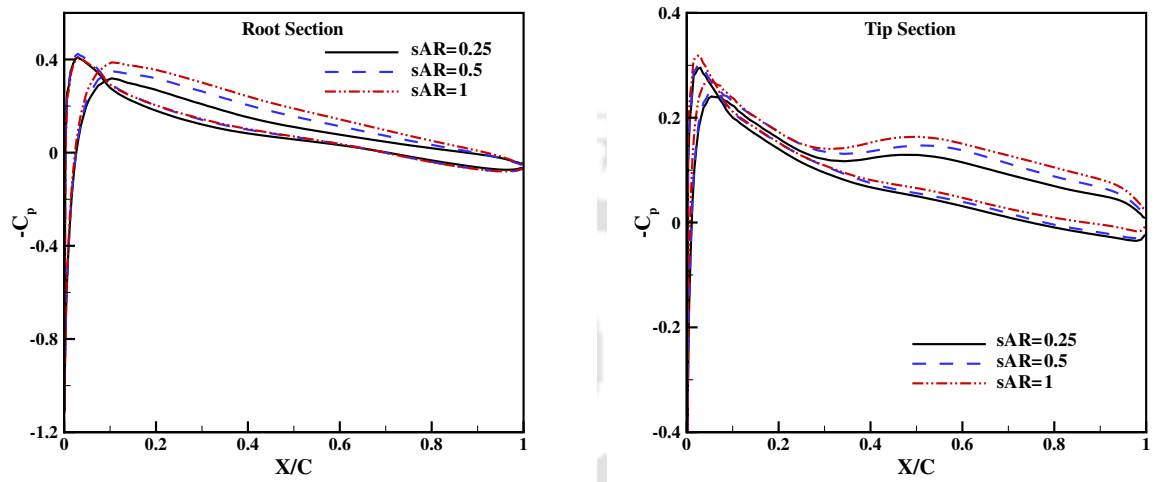


(c) Lift to Drag Ratio

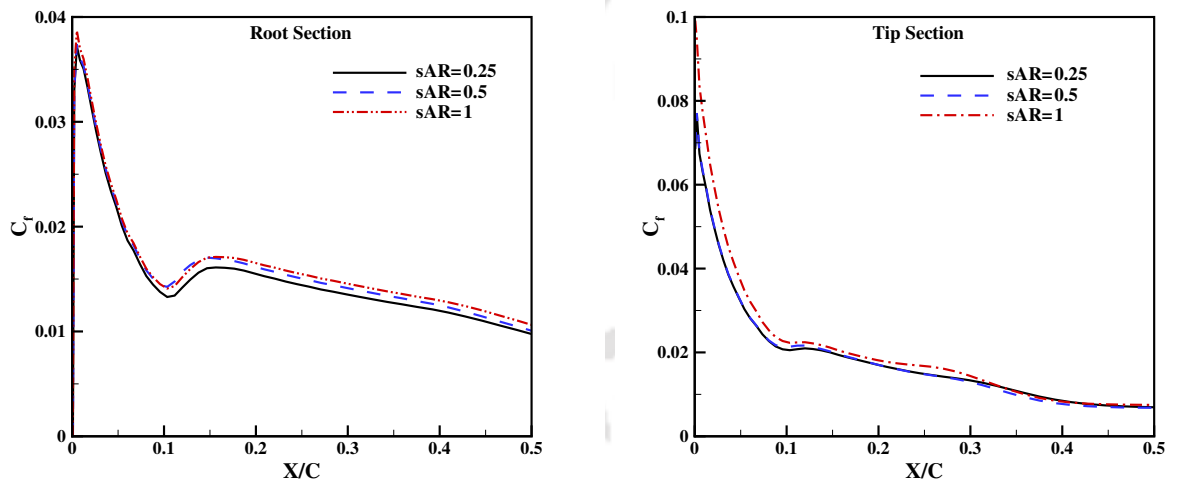


(d) Coefficient of Moment

Figure 8.12: Effect of wing-span on the aerodynamic coefficient

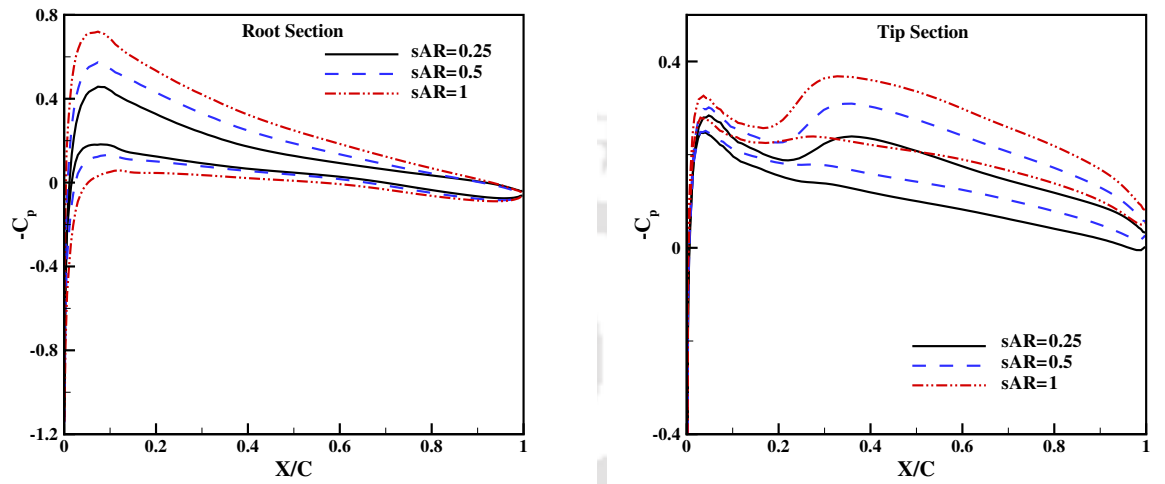


(a) Surface pressure coefficient (C_p)

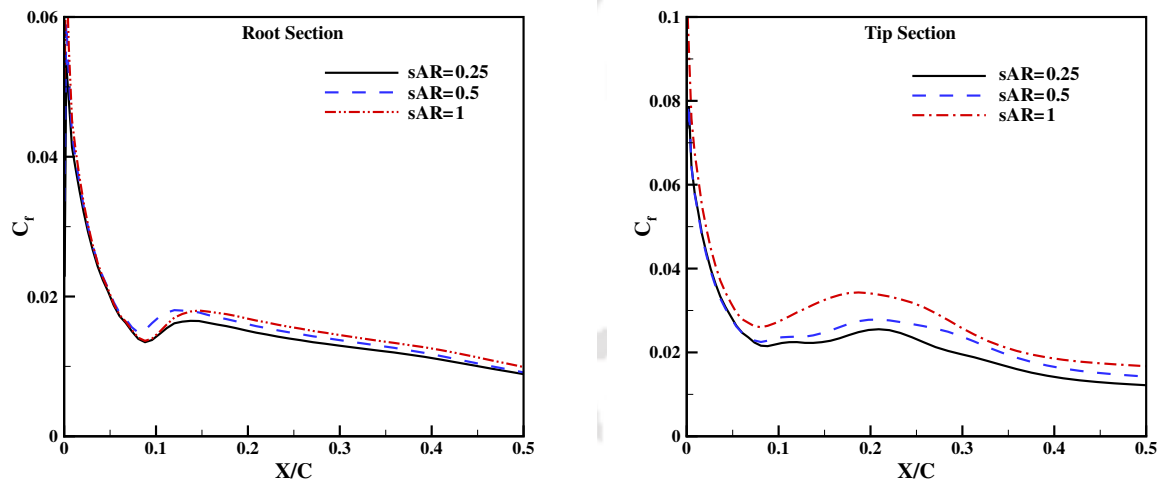


(b) Skin-friction coefficient (C_f)

Figure 8.13: Comparison of C_p and C_f for the wing root and tip sections at $\alpha = 0^\circ$

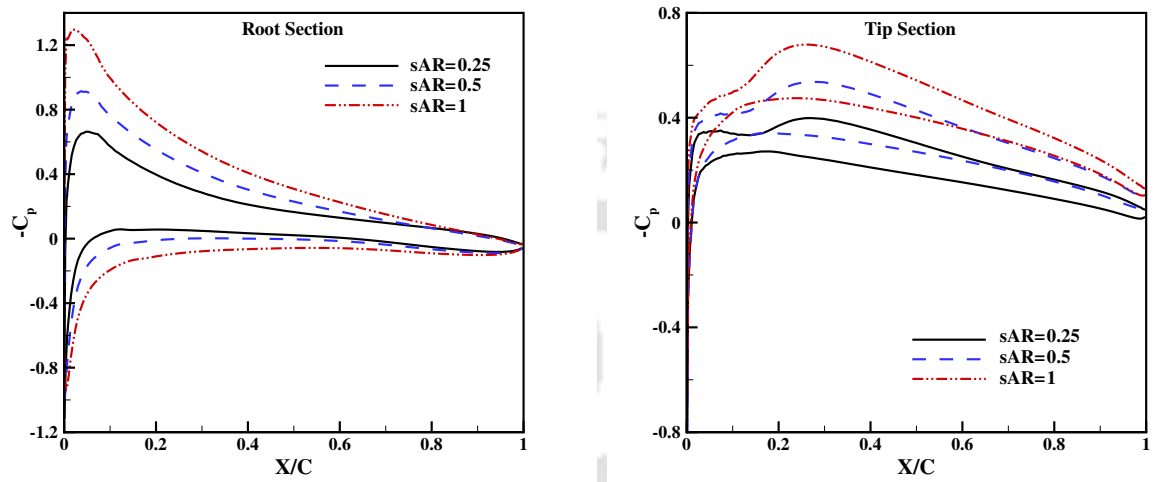


(a) Surface pressure coefficient (C_p)

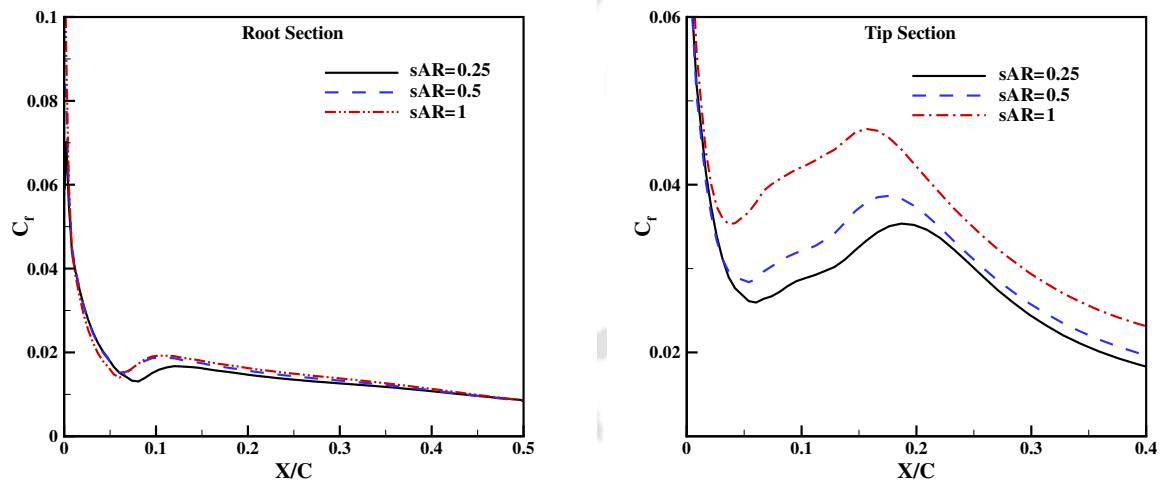


(b) Skin-friction coefficient (C_f)

Figure 8.14: Comparison of C_p and C_f for the wing root and tip sections at $\alpha = 4^\circ$

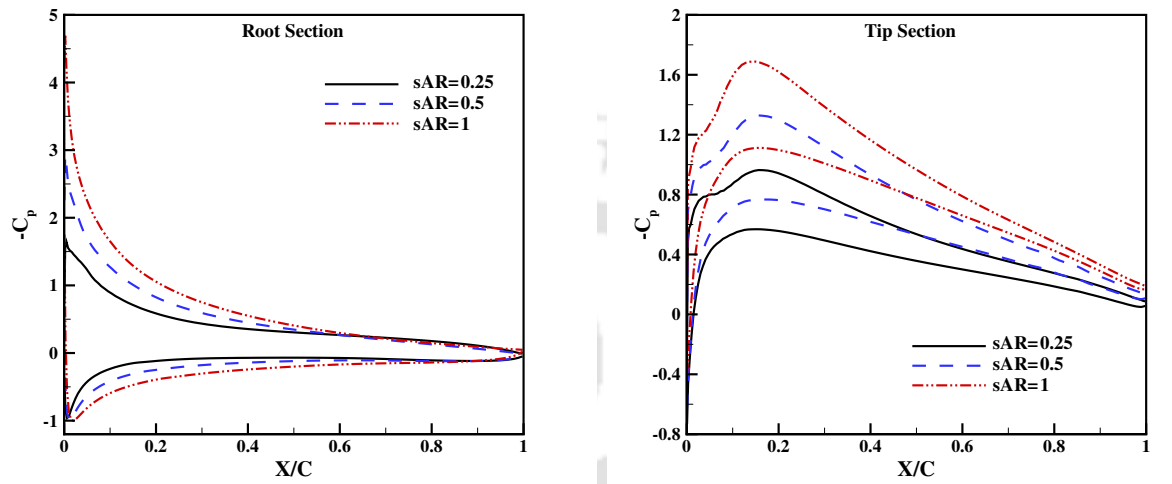


(a) Surface pressure coefficient (C_p)

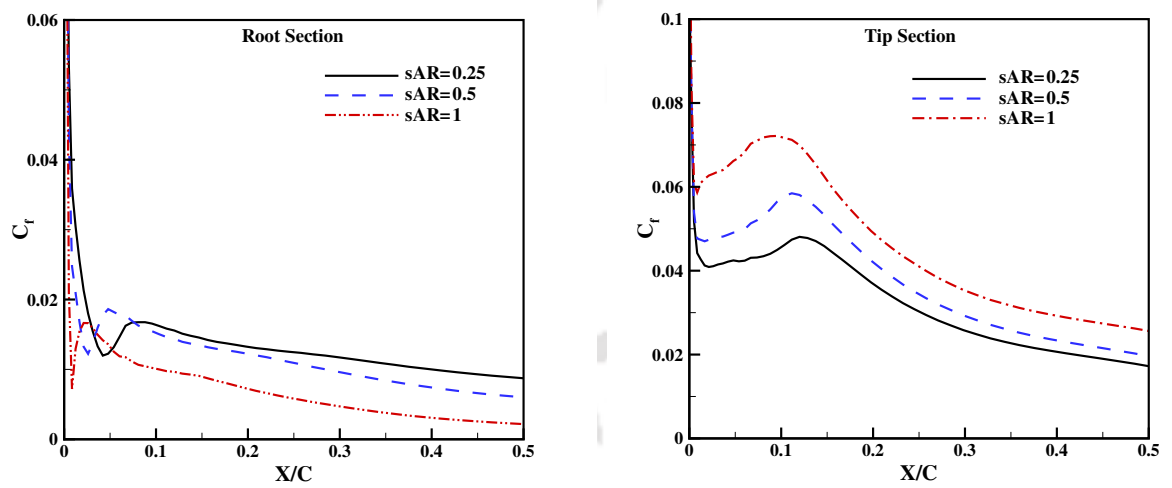


(b) Skin-friction coefficient (C_f)

Figure 8.15: Comparison of C_p and C_f for the wing root and tip sections at $\alpha = 8^\circ$



(a) Surface pressure coefficient (C_p)



(b) Skin-friction coefficient (C_f)

Figure 8.16: Comparison of C_p and C_f for the wing root and tip sections at $\alpha = 16^\circ$

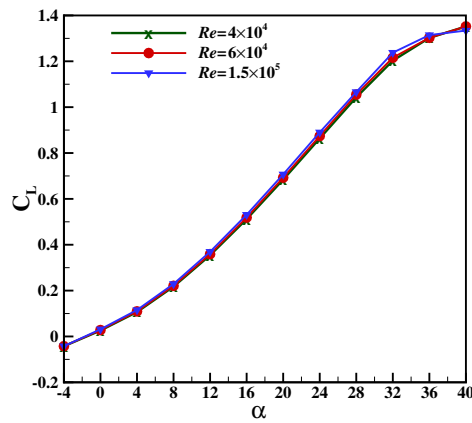
8.4 Effect of Reynolds Number on the Flow Characteristics

The influence of Reynolds number in the low Re regime on the performance of the SD7003 rectangular wing with 0.25 a semi aspect ratio are discussed in this section. In this study, the simulations are carried out for three Reynolds number (4×10^4 , 6×10^4 and 1.5×10^5) using the $\gamma - Re_\theta$ transition model and the effect of the Reynolds number on the transition onset, flow characteristics and aerodynamic coefficients are analysed.

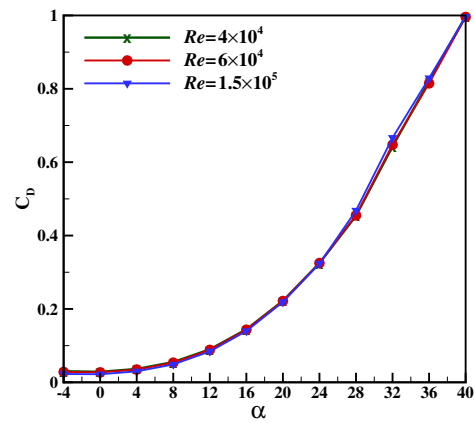
The aerodynamic coefficients obtained at the three Reynolds number are shown in Fig. 8.17. It is evident from this figure that increasing the Reynolds number from 4×10^4 to 1.5×10^5 has not significantly influence the variation of the lift, drag and pitching moment coefficients for the SD7003 rectangular wing with $sAR = 0.25$. However a close examination of these aerodynamic coefficients reveal that increasing the Reynolds number has produced higher lift coefficient, lower drag coefficient and lower moment coefficient. This higher C_L and lower C_D at higher Re has resulted in the higher lift to drag ratio.

In order to examine the influence of varying the Reynolds number on the transition onset and flow structure, the Reynolds shear stress contours on the wing surface along with the contours and streamlines on the 2D sections of the wing obtained for the three Reynolds number is shown for $\alpha = 8^\circ$, $\alpha = 16^\circ$ and $\alpha = 32^\circ$ in Fig. 8.18 to Fig. 8.20 respectively. The contours of the Reynolds shear stress on the wing surface and 2D sections clearly indicate that the level of the Reynolds shear stress which depict the turbulence intensity reduces with increase in Re this similar feature was also observed in our 2D simulations. It is further noticed that the level of the turbulence intensity obtained for the 3D rectangular wing is lesser than the 2D airfoil. This feature of reduction in the turbulence level with increase in Re and for geometry change from 3D to 2D was also observed in the experimental study on the 2D and 3D large scale wavy wall carried out by Hamed *et al.* (2015). The transition onset location for $\alpha = 8^\circ$ and $\alpha = 16^\circ$ as indicated in the Reynolds shear stress contours is observed to move upstream as the Reynolds number is increased for the wing root section as well as for the tip section. It is further noticed that the transition onset location move upstream with increase in angle of attack resulting in the reduction of the laminar region. Eventually beyond $\alpha = 28^\circ$, the flow over this wing is found to become fully turbulent as shown for $\alpha = 32^\circ$

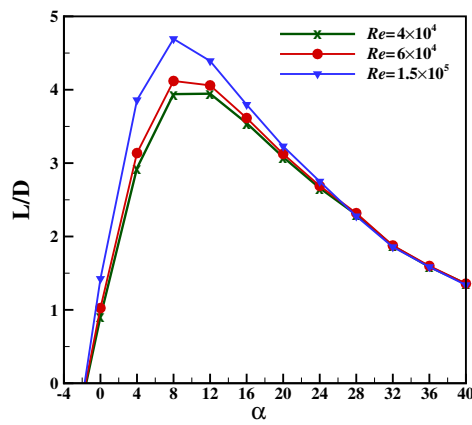
in Fig. 8.20 where the level of Reynolds shear stress is very high from the leading edge itself. At this angle of attack for all the three Reynolds number, negative Reynolds shear stress contours is obtained in the region covering almost half the wing from the wing leading edge to half the chord and from root section to almost mid span. The streamline shown on the root and mid sections of the wing clearly indicates that flow remains attached at lower angles of attack unlike the SD7003 airfoil where a laminar separation bubble is observed both in experiments (Ol *et al.*, 2005; Radespiel *et al.*, 2006) and in present computations (Sec 6.3) and other computations (Yuan *et al.*, 2005; Galbraith and Visbal, 2008; Uranga *et al.*, 2009) for these Reynolds number. The 2D streamline at tip section of the wing are observed to be quite different from the other two section which can be attributed to the formation of the tip vortices. The streamlines on the root section of the wing at $\alpha = 32^\circ$ for the three Reynolds number indicates flow separation and the formation of the fully turbulent bubble covering from leading edge to the mid chord of this wing section. The separation bubble length and height at the root section is almost identical for the three Reynolds number. However, at the mid section a prominent separation bubble is observed only at the highest Reynolds number analysed. The formation of the separation bubble at this angle of attack is the manifestation of the negative Reynolds shear stress contours obtained in this region. The computed volume streamlines obtained for the three Reynolds number is shown in Fig. 8.21 for three angles of attack which clearly reflect the formation of the wing tip vortices for all the three Reynolds numbers. It is quite evident from this figure that increasing the Reynolds has not greatly influenced the strength and inception of the tip vortices. However it can be noticed that the inception of tip vortices on the wing surface to move upstream with its strength increasing as angle of attack is increased causing the formation of tip vortices close to the leading edge at $\alpha = 32^\circ$. Similar to the 2D streamlines on the wing section at this angle of attack, the volume streamlines also indicate flow separation on the wing surface.



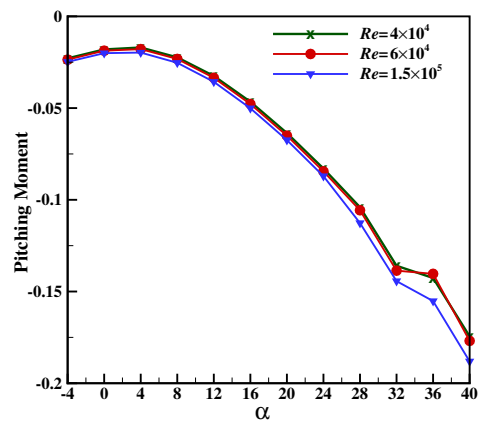
(a) Coefficient Lift



(b) Coefficient Drag

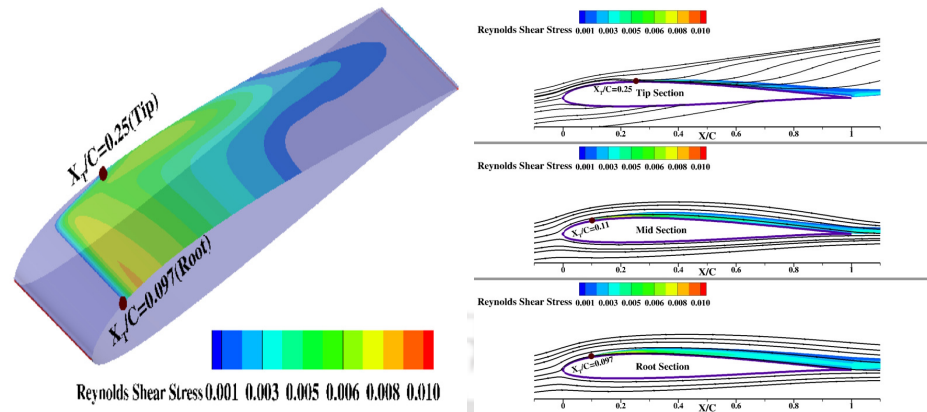


(c) Lift to Drag Ratio

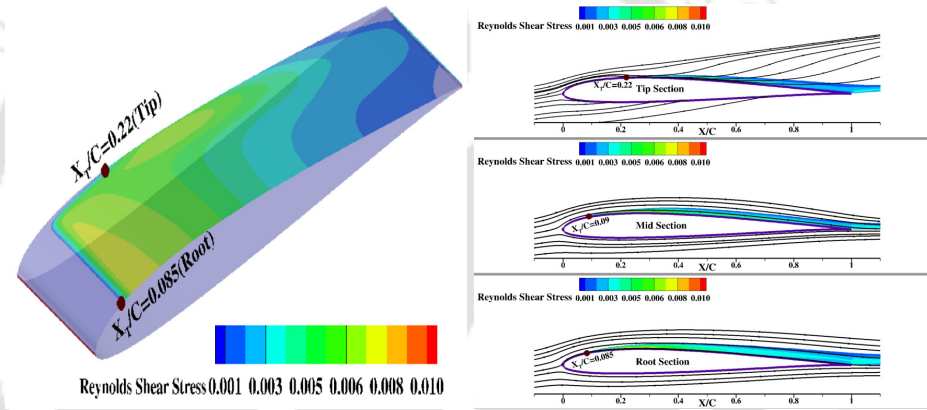


(d) Coefficient of Moment

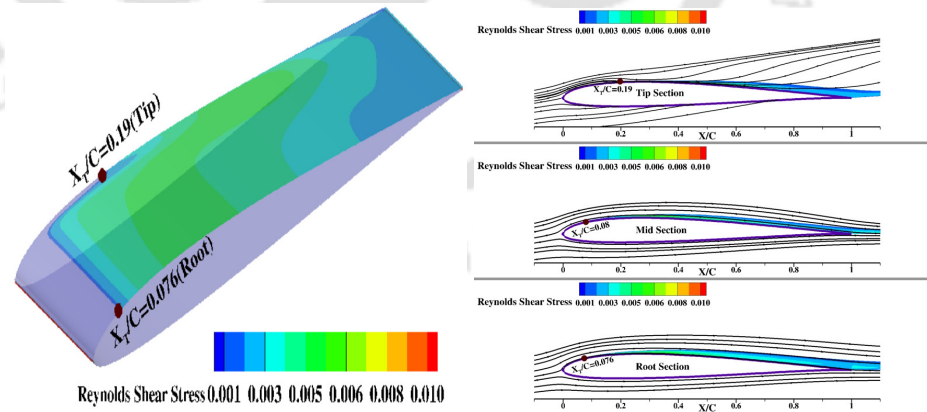
Figure 8.17: Effect of Reynolds number on the aerodynamic coefficient



(a) $Re = 4 \times 10^4$



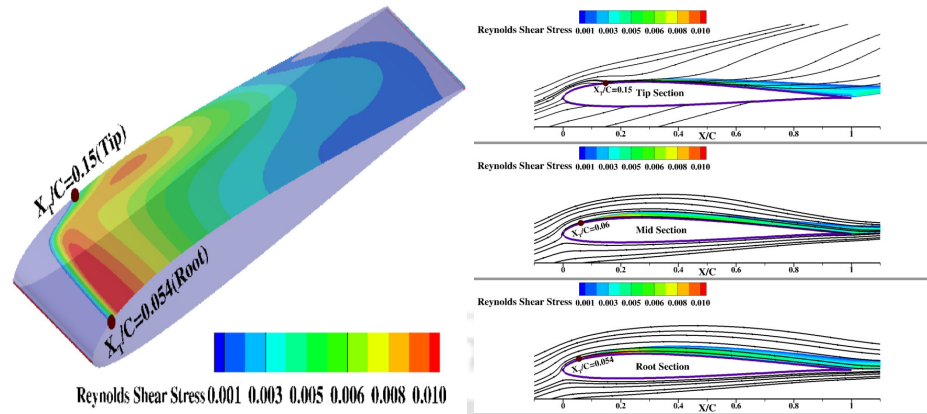
(b) $Re = 6 \times 10^4$



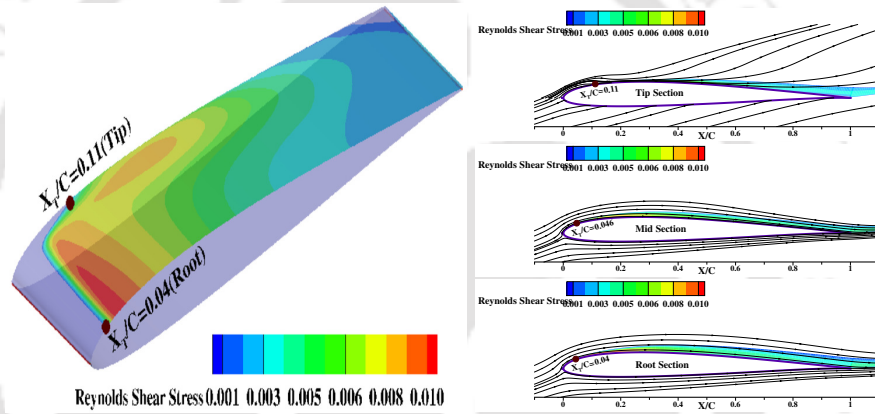
(c) $Re = 1.5 \times 10^5$

Figure 8.18: Reynolds shear stress contours at different Re for $sAR = 0.25$ and $\alpha = 8^\circ$

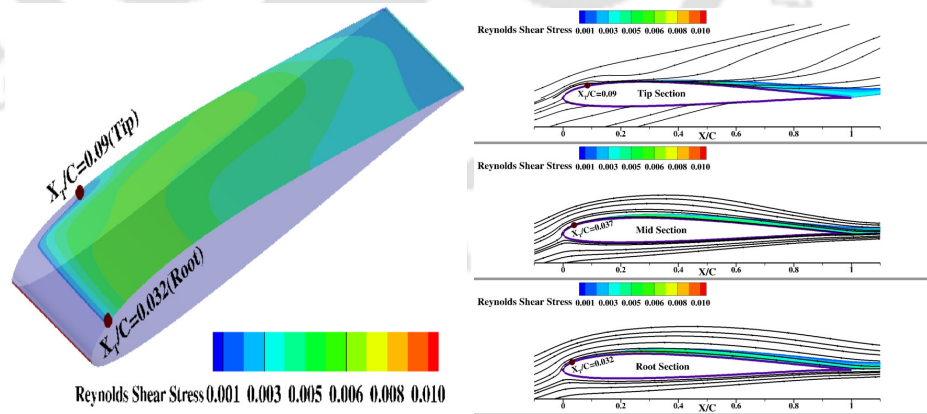
(Left figure : Surface contours, Right figure : Contours on the 2D sections)



(a) $Re = 4 \times 10^4$



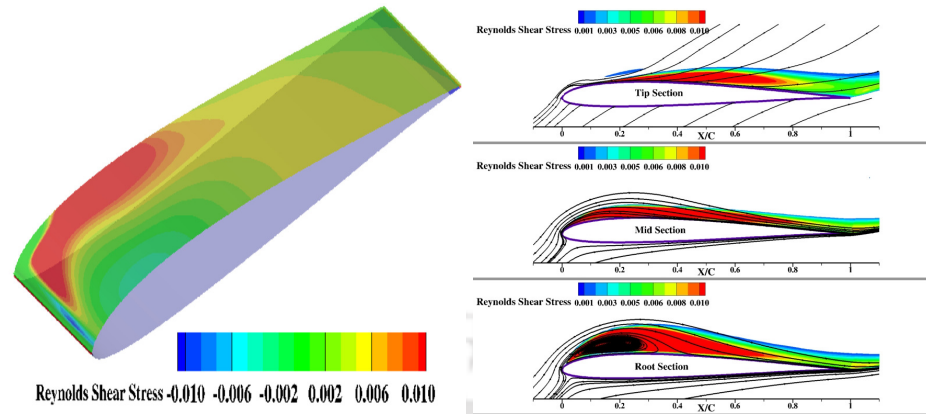
(b) $Re = 6 \times 10^4$



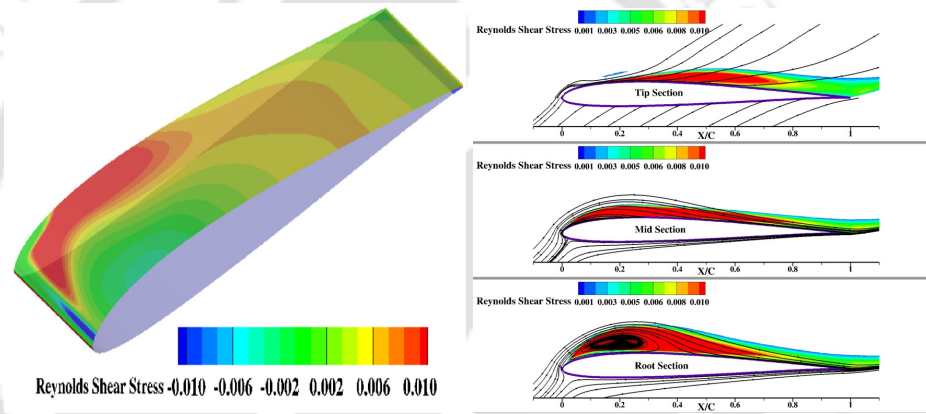
(c) $Re = 1.5 \times 10^5$

Figure 8.19: Reynolds shear stress contours at different Re for $sAR = 0.25$ and $\alpha = 16^\circ$

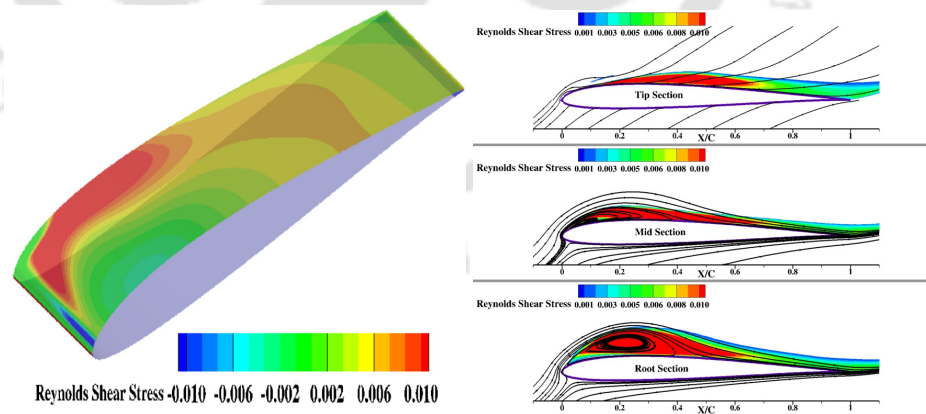
(Left figure : Surface contours, Right figure : Contours on the 2D sections)



(a) $Re = 4 \times 10^4$



(b) $Re = 6 \times 10^4$



(c) $Re = 1.5 \times 10^5$

Figure 8.20: Reynolds shear stress contours at different Re for $sAR = 0.25$ and $\alpha = 32^\circ$

(Left figure : Surface contours, Right figure : Contours on the 2D sections)

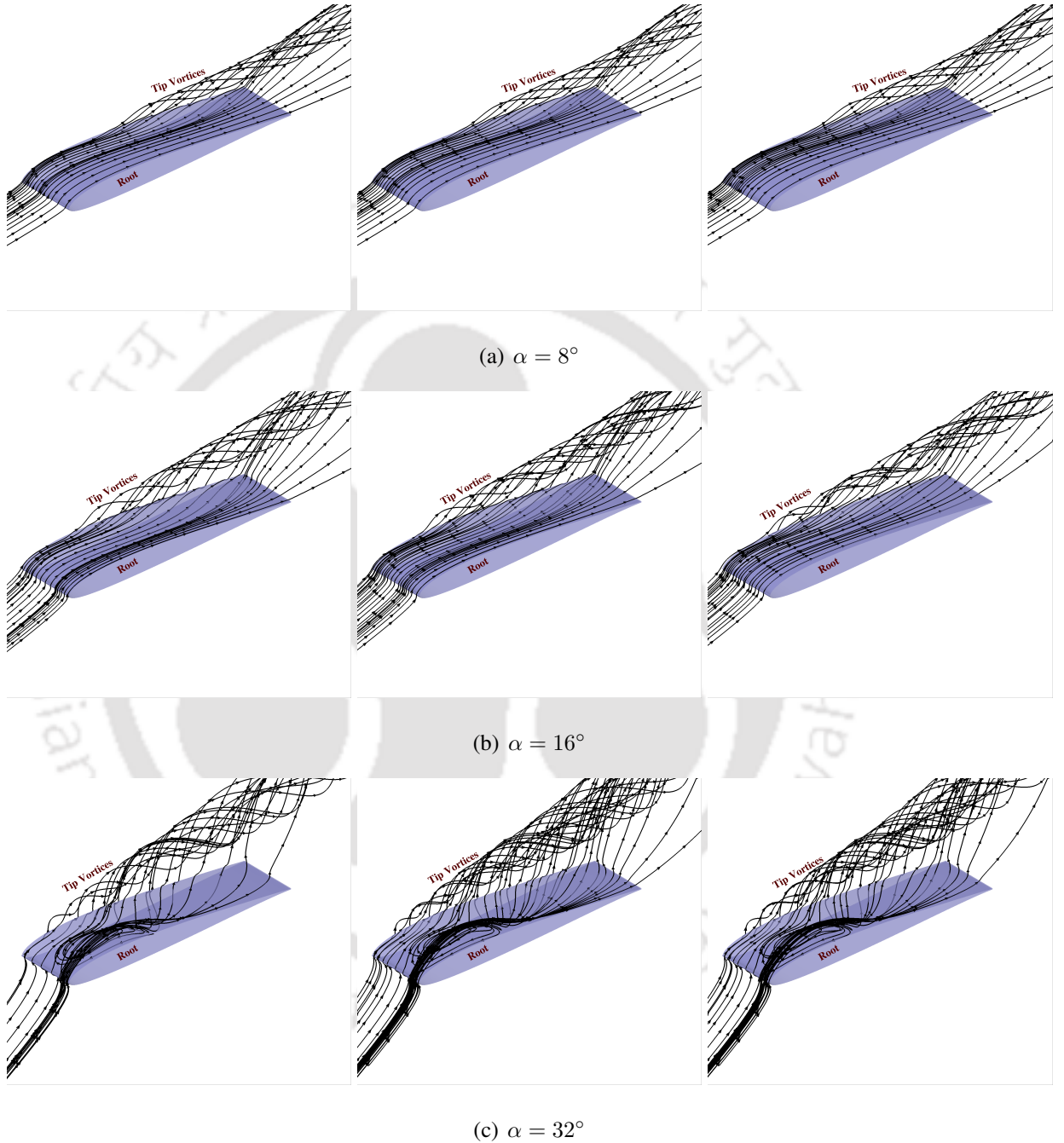


Figure 8.21: Effect of Re on tip vortex formation at different α for wing $sAR = 0.25$
 (Left figure : $Re = 4 \times 10^4$, Center figure: $Re = 6 \times 10^4$ Right figure : $Re = 1.5 \times 10^5$)

8.5 Summary

The $\gamma - Re_\theta$ SST transition model was successfully used to simulate the effect of wing aspect ratio at $Re = 6 \times 10^4$ for SD7003 rectangular wing with three semi aspect ratio (sAR) viz. 0.25, 0.5 and 1. The wing aspect ratio was found to significantly influence both the transition and aerodynamic characteristics especially beyond $\alpha = 0^\circ$ at this Reynolds number. The Reynolds shear stress contours as well as the skin-friction coefficient plots indicated that (i) the transition onset is delayed and length of the transition zone is longer for the wing with the lowest aspect ratio ($sAR = 0.25$) as compared to other two wings (ii) The length of the laminar and transition region is found to reduce with angle of attack for all the three wings. (iii) transition onset location is observed to be almost constant from the root section to the section close to tip region along the wing planform. The volume streamlines indicated the formation of the tip vortices which were observed to be almost identical for the three wings analysed. The strength of the vortices was found to increase as angle of attack increased with its onset location moving upstream. The stall angle obtained for wing with the lowest aspect ratio was much delayed compared to other two wings leading to a higher maximum C_L . However at angles of attack before the stall angle of the other two wings, the wing with highest aspect ratio was found to have the highest lift coefficient. The drag coefficient was not significantly influenced by the wing aspect ratio but it was found that the wing with the lowest aspect ratio had the lowest drag coefficient as it was found to have a longer laminar region. The L/D was found to increase with increase in angle of attack with the $(L/D)_{max}$ occurring at 8° for all the three wings. The wing with the lowest aspect ratio ($sAR = 0.25$) though having lower L/D may be considered as a suitable candidate for MAV application at this low Reynolds number because of its better aerodynamic and transition performance. The simulations indicated that aerodynamic, transition and flow characteristics of this wing with $sAR = 0.25$ was not greatly influenced when the Reynolds number was increased from $Re = 4 \times 10^4$ to 1.5×10^5 . However, the aerodynamic performance in terms of the lift to drag ratio and moment coefficient was found to slightly better at $Re = 1.5 \times 10^5$. The flow found to be fully turbulent beyond $\alpha = 28^\circ$ at all the three Reynolds number where the flow separation on the wing surface was obtained due to the negative Reynolds stress in this region.

Chapter 9

Conclusions and Scope for Future Work

9.1 Conclusions

Numerical simulations have been carried out past the SD low Re airfoil series and SD7003 rectangular wing using the RANS based transition models available in the in-house incompressible flow solution code 3D-PURLES. These simulations were carried out in the low Re flow regime (4×10^4 to 2×10^5) and the predicted aerodynamic and transition features for the low Re SD airfoils have been compared with the measurement data as well as other computation results. The validation study carried out indicated that the lift coefficient and the transition onset obtained by the present computation matched well with both the experimental data and LES/DNS computational data of others. On the other hand, when compared to the experiments, the length of the LSB obtained by the computations in general were observed to be over predicted which resulted in a enhancement of the drag coefficient. The influence of different grid quality parameters and flow solution control parameters on the the accuracy in predicting the flow characteristics of SD7003 airfoil at $Re = 6 \times 10^4$ using 3D-PURLES were studied. The parameters considered in the grid generation procedure which are likely to influence the flow solution accuracy are namely farfield distance, grid spacing near the leading and trailing edge of the airfoil, near-wall distance and different grid topologies. Whereas for the flow solution procedure, the different parameters such as spatial discretisation schemes, freestream turbulence intensity and inlet eddy viscosity ratio were considered to study

their influence on the accuracy of flow solution. Based upon this study, C-Grid with farfield placed at a distance of $15C$ having a leading and trailing edge grid resolution of $(\Delta S) = 2.4 \times 10^{-3}C$ and grid near-wall distance of $\Delta Y_{nw} = 9.8 \times 10^{-5}C$ were found to be good enough for this low Reynolds number. Whereas for the flow solution code using third order accurate QUICK spatial discretisation scheme with freestream turbulent intensity of 0.1% and inlet eddy viscosity ratio of one was found to be sufficient to obtain reasonable accurate solution for SD7003 airfoil at $Re = 6 \times 10^4$. Based on this study, the above parameters were fixed for all other computations carried out in this work.

The $k_T - k_L - \omega$ and $\gamma - Re_\theta$ SST transition models were successfully used to predict the transition and aerodynamic characteristics of the SD7003 airfoil at $Re = 6 \times 10^4$ and their relative performance was studied. The two transition models could capture the transition onset location as well as the laminar separation bubble. The transition onset location obtained by $\gamma - Re_\theta$ SST transition model was found to be in good agreement with the measurement and other computations at all angles of attack but $k_T - k_L - \omega$ model predicted the location upstream of the airfoil which deviated by 15% on the average with the $\gamma - Re_\theta$ SST model. It was further noted that the present computation as well as the other computations in general have predicted the separation point upstream on the airfoil surface as compared to the measurement data. The separation location obtained by $\gamma - Re_\theta$ SST model was found to be in close agreement with that predicted by ILES data of Galbraith and Visbal (2008). The $k_T - k_L - \omega$ model similar to the transition location had predicted the separation location upstream leading to a larger discrepancy with the measurement data. However, both the transition models have predicted the reattachment downstream of the airfoil surface when compared to measurements as well as other LES/DNS computations with the discrepancy being higher for $k_T - k_L - \omega$ model. The delayed in the reattachment point obtained by the present computation has resulted in the longer LSB. This delay in the reattachment point may be attributed to the lower level of turbulence predicted by these transition models and their inability to rapidly produce enough turbulence kinetic energy in the shear layer to induce early flow reattachment beyond the transition point. The longer LSB obtained by the $\gamma - Re_\theta$ SST model has lead to an early stall when compared to the ILES data. However this early stall was observed in the experiments conducted by Rival and Tropea (2010). On the other hand, the $k_T - k_L - \omega$ model failed to predict stall angle as this model in the present formulation is not designed to handle separated flow transition. It was

further noticed that the computations in general predicted a slightly higher drag compared to the measurements with $\gamma - Re_\theta$ SST model following the ILES data closely. The drag predicted by the $k_T - k_L - \omega$ model was much higher as the LSB obtained by this model was much larger both in terms of length and height when compared to $\gamma - Re_\theta$ SST model. The flow pattern obtained by the $k_T - k_L - \omega$ model showed the formation of a large bubble attached to the airfoil surface with length of the bubble remaining constant beyond $\alpha = 8^\circ$ but the height of the bubble was found to increase with increase in angle of attack resulting in a continuous increase in the lift coefficient and due to this the stall angle was not predicted even at $\alpha = 16^\circ$. The transition and aerodynamic performance of the SD7003 airfoil were found to be influenced by changing the Reynolds number in the low Re regime (4×10^4 to 1.5×10^5). The aerodynamic performance was observed to improve with increase in Reynolds number from $Re = 4 \times 10^4$ to $Re = 1.5 \times 10^5$ which is mainly because the strength of the laminar separation bubble was weaker. The soft stall and nonlinearity in lift coefficient curve features expected at low Re flows were captured by the present simulation at the lowest Reynolds number ($Re = 4 \times 10^4$).

The effect of camber and thickness for the low Re SD airfoil series were analysed at two Reynolds number in the low Re regime and the aerodynamic data was validated against the available measurements. The aerodynamic data in general was found to be in good agreement with measurement data for most of the cases analysed. The discrepancies obtained with the measurements was found to reduce when the Re was increased. The present study showed that, increasing the camber improved the aerodynamic characteristics in terms of C_{Lmax} and $(L/D)_{max}$ however stall angle was not effected by the airfoil camber. It is further observed that at both the Reynolds number, the flow remained laminar over a longer region for the airfoil with highest camber particularly at higher angles of attack. In the low Re case, the length of LSB was not greatly influenced by the airfoil camber at lower angles of attack but beyond $\alpha = 4^\circ$ the length of the LSB was quite larger compared to the other two airfoils. On the other hand, when the Re was increased to 2×10^5 the effect of camber on the length LSB was not very significant for all angles of attack but in general it was observed that increase in camber increased the length of LSB. The airfoil camber was found to influence the transition features as well with the airfoil having the highest camber was found to have longer laminar region and larger laminar separation bubble particularly at higher angle of attack. The effect

of varying the airfoil thickness on the aerodynamic coefficients was not as prominent as observed in the effect of varying camber as the difference in the thickness was small which was about 10%. In general it was found that increasing the thickness increased the lift coefficient to some extent and the lift to drag ratio. The thicker airfoil was found to have larger turbulent region at lower angles of attack whereas the thinner airfoil was found to be more turbulent at angles close to the stall angle. The length of LSB obtained for the thicker airfoil in general was found to be longer than that obtained by the thinner airfoil. Increasing the Re enhanced the C_{Lmax} , $(L/D)_{max}$ and delayed the stall angle for the airfoils with different camber and thickness considered for this study.

The $\gamma - Re_\theta$ SST transition model was successfully used to simulate the effect of wing aspect ratio at $Re = 6 \times 10^4$ for SD7003 rectangular wing with three semi aspect ratio (sAR) viz. 0.25, 0.5 and 1. The wing aspect ratio was found to significantly influence both the transition and aerodynamic characteristics especially beyond $\alpha = 0^\circ$ at this Reynolds number. The Reynolds stress contours as well as the skin-friction coefficient plots indicated that (i) the transition onset was delayed and length of the transition zone was longer for the wing with the lowest aspect ratio ($sAR = 0.25$) as compared to other two wings (ii) The length of the laminar and transition region was found to reduce with angle of attack for all the three wings. (iii) transition onset location was observed to be almost constant from the root section to the section close to tip region along the wing planform. The volume streamlines indicated the formation of the tip vortices which were observed to be almost identical for the three wings analysed. The strength of the vortices was found to increase as angle of attack increased with its onset location moving upstream. The stall angle obtained for wing with the lowest aspect ratio was much delayed compared to other two wings leading to a higher maximum C_L . However at angles of attack before the stall angle of the other two wings, the wing with highest aspect ratio was found to have the highest lift coefficient. The drag coefficient was not significantly influenced by the wing aspect ratio but it was found that the wing with the lowest aspect ratio had the lowest drag coefficient as it was found to have a longer laminar region. The L/D was found to increase with increase in angle of attack with the $(L/D)_{max}$ occurring at 8° for all the three wings. The wing with the lowest aspect ratio ($sAR = 0.25$) though having lower L/D may be considered as a suitable candidate for MAV application at this low Reynolds number because of its better aerodynamic and transition performance. The simulations indicated

that aerodynamic, transition and flow characteristics of this wing with $sAR = 0.25$ was not greatly influenced when the Reynolds number was increased from $Re = 4 \times 10^4$ to 1.5×10^5 . However, the aerodynamic performance in terms of the lift to drag ratio and moment coefficient was found to slightly better at $Re = 1.5 \times 10^5$. The flow found to be fully turbulent beyond $\alpha = 28^\circ$ at all the three Reynolds number where the flow separation on the wing surface was obtained due to the negative Reynolds shear stress in this region.

Importance of this study in the design of MAVs: The study of influence of airfoil geometrical parameters indicated that, increasing the airfoil camber had a more significant influence on the lift coefficient and lift to drag ratio as compared to the thickness. In this direction, the design team can only vary airfoil camber to improve the aerodynamic performance by retaining the same thickness. This change in camber will not hamper the weight of the configuration which is a major concern in the MAV design. The influence of the wing aspect ratio on the aerodynamic and transition characteristics indicated that the wing with lowest aspect ratio may be more suitable and efficient for the design of MAV configuration since it has delayed stall (the wing can sustain larger angle of attack) angle and maximum lift coefficient (more endurance, more manoeuvrability), lower drag coefficient (lesser battery power consumption) and a better moment coefficient (more stability).

9.2 Scope for Future Work

The future work planned are as follows :

- (1) Incorporation of Detached Eddy Simulation (DES) in the existing code for analysing airfoil/wings at high angles of attack and at high Reynolds number.
- (2) Relative assessment of DES and transition models for Natural laminar flow (NLF) airfoil/wing at high Reynolds number.
- (3) Implementation or improvement of the transition models in the open source CFD codes based on this research work for our in-house aircraft programs.

Bibliography

- Aarunvinthan, S. and Pillai, S. N. (2019). Aerodynamic characteristics of unsymmetrical aerofoil at various turbulence intensities. *Chinese Journal of Aeronautics*, 32, 2395–2407.
- Ananda, G. K., Sukumar, P. P., and Selig, M. S. (2012). Low to moderate aspect ratio wings tested at low reynolds number. In *30th AIAA Applied Aerodynamics Conference, New Orleans, Louisiana*.
- Ananda, G. K., Sukumar, P. P., and Selig, M. S. (2015). Measured aerodynamic characteristics of wings at low Reynolds number. *Aerospace Science and Technology*, 42, 392–406.
- Anyojiu, M., Liu, T., Nonomura, T., Oyama, A., and Fujji, K. (2003). Effect of planform on aerodynamic characteristics at low Reynolds numbers using a low density wind tunnel. In *Fluid Dynamics and Co-located Conference*.
- Aranda, S. M., Gercia-Gonzalez, A. L., Parras, L., Velazquez-Navarro, J. F., and Pino, C. D. (2016). Comparison of the aerodynamic characteristics of the NACA0012 airfoil at low-to-moderate Reynolds numbers for any aspect ratio. *International Journal of Aerospace Sciences*, 4, 1–8.
- Arnal, D., Casalis, G., and Juillen, J. C. (1996). A survey of the transition prediction methods: from analytical criteria to PSE and DNS. In *Turbulence and Transition modeling, Lecture Notes from the ERCOFTAC/IUTAM Summer school held in Stockholm*. Kluwer academic publishers, Dordrecht.
- Bastedo, W. G. and Mueller, T. J. (1986). Spanwise variation of laminar separation bubbles on wings at low Reynolds number. *Journal of Aircraft*, 23, 687–694.

- Boom, P. D. and Zingg, D. W. (2013). Time-accurate flow simulations using an efficient Newton-Krylov-Schur approach with high-order temporal and spatial discretization. In *51th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, Grapevine, Texas*, Number 2013-0383.
- Burgmann, S., Brucker, C., and Schroder, W. (2006). PIV measurements of a laminar separation bubble. *Experiments in fluids*, 41, 319–326.
- Carmichael, B. (1981). Low Reynolds number airfoil survey. Technical Report 165803(1), NASA Contractor.
- CartondeWiert, C. and Hillewaert, K. (2012). DNS and ILES of transitional flows around a SD7003 using a high order discontinuous Galerkin method. In *Seventh International Conference on Computational Fluid Dynamics, Big Island, Hawaii*, Number ICCFD7-3604.
- Castonguay, P., Liang, C., and Jameson, A. (2010). Simulation of transitional flow over airfoils using the spectral difference method. In *40th AIAA Fluid Dynamics Conference and Exhibit, Chicago, Illinois*, Number 2010-4626, pp. 1–17.
- Catalano, P. and Tognaccini, R. (2011). RANS analysis of the low-Reynolds number flow around the SD7003 airfoil. *Aerospace Science and Tech.*, 15, 615–626.
- Chen, Z. J. and Qin, N. (2013). Planform effects for low-Reynolds-number thin wings with positive and reflex cambers. *Journal of Aircraft* 3), 952–964.
- Chien, K. Y. (1982). Predictions of channel and boundary layer flows with a low Reynolds no. turbulence model. *AIAA Journal*, 20, 321–339.
- Cole, G. M. and Mueller, T. J. (1990). Experimental measurement of the laminar separation bubble on an Eppler-387 airfoil at low Reynolds number. *UNDAS-1419-FR*.
- Cosyn, P. and Vierendeels, J. (2006). Numerical investigation of low-aspect-ratio wings at low Reynolds numbers. *Journal of Aircraft*, 43, 713–722.

- Counsil, J. N. N. and Boulama, K. G. (2011). Validating the URANS shear stress transport $\gamma - Re_\theta$ model for low-Reynolds-number external aerodynamics. *International Journal For Numerical Methods In Fluids*, 69, 1411–1432.
- Coupland, J. (1990). Flat plate transitional boundary layers, ERCOFTAC classic database. <http://cfd.mace.manchester.ac.uk/ercoftac>.
- Demir, H., Ozden, M., Genc, M. S., and Cagdas, M. (2016). Numerical investigation of flow on NACA4412 aerofoil with different aspect ratios. In *EPJ Web of Conferences*, Number 114.
- Deng, S., Jiang, L., and Liu, C. (2007). DNS for separation control around an airfoil by pulsed jets. *Computers and Fluids*, 36, 1040–1060.
- Derla, M. (2002). XFOIL Users Guide, Version 6.94. MIT Aero. and Astro. Department.
- Durbin, P. A. (1995). Separated flows computation with the $k - \epsilon - \overline{v^2}$ model. *AIAA Journal*, 33, 659–644.
- Elimelch, Y., Arieli, R., and Iosilevskii, G. (2005). On the onset of transition at low reynolds number flow over airfoils. In *4th AIAA Theoretical Fluid Mechanics Meeting, Toronto, Canada*.
- Fathima, A., Baldawa, N. S., Pal, S., and Majumdar, S. (1994). Grid generation for arbitrary 2D configurations using a differential algebraic hybrid method. Technical Report PD CF 9461, National Aerospace Laboratories, Bangalore.
- Furst, J., Prihoda, J., and Straka, P. (2013). Numerical simulation of transitional flows. *Computing*, 95, 163–182.
- Galbraith, M. C. and Visbal, M. R. (2008). Implicit large-eddy simulation of low Reynolds number flow past the SD7003 airfoil. In *46th AIAA Aerospace Sciences Meeting and Exhibit, Reno, Nevada*, Number 2008-225.
- Germann, D. J. and Visbal, M. R. (2012). High-order solutions of transitional flow over the SD7003 airfoil using compact finite-difference and filtering. In *AIAA 1st International Workshop on High-Order CFD Methods*.

- Gordnier, I. E. and Visbal, M. R. (2009). Implicit LES computations with applications to micro air vehicles. In *High performance computing modernization program users group conference*.
- Grameyer, J. M. and Keennon, M. T. (2001). Development of the black micro air vehicle. AIAA Paper, 2001-0127.
- Grundy, T. M., Keefe, G. P., and Lowson, M. V. (2000). Effect of acoustic disturbances on low Re airfoils. In *In Proc. Conf. on Fixed, Flapping and Rotary wing vehicles at very low Reynolds numbers*, pp. 91–113.
- Hamburg, S. D., Napolillo, Z., Gautam, M., and Huebsch, W. W. (2011). Computational investigation of aspect ratio and taper ratio effects on wings operating in the transitional low Reynolds number flight regime. In *29th AIAA Applied Aerodynamics Conference, Honolulu, Hawaii*.
- Hamed, A. M., Kamdar, A., Castillo, L., and Chamorro, L. P. (2015). Turbulent boundary layer over 2D and 3D large-scale wavy walls. *Physics of fluids*, 27, 1–14.
- Hein, B. R. (2005). Hover performance of a micro air vehicle: Rotors at low Reynolds number. AHS Lichten Award Competition.
- Hoarau, Y., Faghani, D., Braza, M., Perrin, R., Anne-archard, and Ruiz (2003). Direct numerical simulation of the three-dimensional transition to turbulence in the incompressible flow around a wing. *Flow, Turbulence and Combustion*, 71, 119–132.
- Horton, H. P. (1968). Laminar separation bubbles in two and three dimensional incompressible flow. Ph. D. thesis, Thesis, University of London.
- Hsiao, C. T. and Pauley, L. L. (1998). Numerical study of the steady state tip vortex flow over a finite-span hydrofoil. *Journal of Fluids Engineering*, 120, 345–353.
- Hu, H. and Yang, Z. (2008). An experimental study of the laminar flow separation on a low-reynolds number airfoil. *Journal of Fluid Engineering*, 130.
- Istavan, M. S., Kurelek, J. W., and Yarusevych, S. (2018). Turbulence intensity effects on laminar separation bubbles formed over an airfoil. *AIAA journal*, 56, 1335–1347.

- Istavan, M. S. and Yarusevych, S. (2018). Effects of free-stream turbulence intensity on transition in a laminar separation bubble formed over an airfoil . *Experiments in Fluids*, 59-52.
- Jian, T. and Ke-Qin, Z. (2004). Numerical and experimental study of flow structure of low-aspect-ratio wing . *Journal of aircraft*, 41, 1196–1201.
- Jones, G. W., Bradshaw, C. J., Papadopoulos, J., and Paltzer, M. F. (2004). Improved performance and control of flapping-wing propelled micro air vehicles. *AIAA Paper*, 2004-0399.
- Karasu, L., Sardar, M., and Acikel, H. H. (2013). Numerical study on low Reynolds number flows over an aerofoil . *Journal of Applied Mechanics Engineering*, 2.
- Kellogg, M. I. and Bowman, W. J. (2004). Parametric design study of the thickness of airfoils at Reynolds numbers from 60,000-150,000. In *42nd AIAA Aerospace Sciences and Exhibit, Reno, Nevada*.
- Khosla, P. K. and Rubin, S. G. (1974). A diagonally dominant second-order accurate implicit scheme. *Computers and Fluids*, 2, 207–209.
- Kondo, K., Aono, H., Oyama, T. N. A., Fujii, K., and Yamamoto, M. (2013). Large eddy simulation of owl-like wing under low Reynolds number conditions. In *Proceedings of the ASME 2013 Fluids Engineering Division Summer Meeting*.
- Kulkarni, D. S., Rajani, B. N., and Kulkarni, V. (2018). Numerical simulation of low Reynolds number airfoils using correlation based $\gamma - Re_\theta$ SST model. In *7th International and 45th National conference on fluid mehcnaics and fluid power*. IIT-Bombay, Maharastra, India.
- Kulkarni, D. S., Rajani, B. N., and Kulkarni, V. (2021). Transitional flow past low Re SD airfoils with varying camber. In *48th National conference on fluid mehcnaics and fluid power*. BITS-Pilani, Rajasthan, India.
- Kulkarni, D. S., Rajani, B. N., and Kulkarni, V. (2023). Transitional flow past low *Re* SD airfoils with varying camber. In *Lecture Notes in Mechanical Engineering*, Volume 2, pp. 333–338. Springer Nature Singapore Pvt. Ltd.

- Kulkarni, D. S., Rajani, B. N., and Majumdar, S. (2001). Studies on temporal and spatial discretisation schemes used in a pressure-based RANS algorithm for incompressible flow. NAL PD CF 0108.
- Kulkarni, D. S., Subrahmanya, M. B., and Rajani, B. N. (2019). Validation of the RANS based transition models for incompressible flows. Technical Report PD-CTFD/2019/1017, National Aerospace Laboratories, Bangalore.
- Laitone, E. V. (1997). Wind tunnel tests of wings at Reynolds numbers below 70000 . Experiments in fluids, 23, 405–409.
- Langtry, R. and Menter, F. R. (2009). Correlation based transition modeling for unstructured parallelized computational fluid dynamics codes. AIAA J., 47, 2894–2906.
- Lauder, B. E. and Spalding, D. B. (1974). The numerical computation of turbulent flows. Computer Methods and Applied Mechanics and Engg., 3, 269.
- Leonard, B. P. (1979). A stable and accurate convective modelling procedure based on quadratic interpolation. Computer Methods in Applied Mechanics and Engg., 19, 59–98.
- Lian, Y. and Shyy, W. (2006). Laminar-turbulent transition of low Reynolds number rigid or flexible airfoil. In *36th AIAA Fluid Dynamics Conference and Exhibit, San Francisco, California, AIAA 2006-3051*.
- Lian, Y. and Shyy, W. (2007). Laminar-turbulent transition of low Reynolds number rigid or flexible airfoil. AIAA Journal, 45(7), 1501–1513.
- Lin, J. C. M. and Pauley, L. L. (1996). Low Reynolds number separation on an airfoil. AIAA Journal, 34, 1570–1576.
- Lowson, M. V. (1999). Aerodynamics of airfoils at low Reynolds numbers. In *UAVs-14th Unmanned Air Vehicle Systems International Conference, United Kingdom*.
- Majumdar, S. (1988). Role of underrelaxation in momentum interpolation for calculation of flow with non-staggered grids. Numerical Heat Transfer, 13, 125–132.

- Majumdar, S., Rodi, W., and Zhu, J. (1992). Three dimensional finite volume method for incompressible flows with complex boundaries. *Journal of Fluid Engineering*, ASME, 496–503.
- Malan, P., Suluksna, K., and Juntasaro, E. (2009). Calibrating the $\gamma - Re_\theta$ transition model for commercial CFD. In *47th AIAA Aerospace Sciences Meeting, Orlando, Florida*, Number 2009-1142.
- Manu, Y. C., Rajesh, A., Subrahmanya, M. B., Kulkarni, D. S., and Rajani, B. N. (2014). Numerical simulation of low Reynolds number airfoils using laminar kinetic energy based transition model. In *3rd International Conference on Recent Advances in Design, Development and Operation of Micro Air Vehicles*, pp. 169–174. JNTU Hyderabad, India.
- McGhee, R. J., Walker, B. S., and Millard, B. F. (1988). Experimental results for the Eppler 387 airfoil at low Reynolds number in the Langley low turbulence pressure tunnel. NASA Technical Memorandum 4062.
- Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32, 1598–1605.
- Menter, F. R., Langtry, R., and Volker, S. (2006). Transition modelling for general purpose CFD code. *Flow Turbulence and Combustion*, 77, 277–303.
- Mohan, S., Rajani, B. N., and Majumdar, S. (2000). Parallelisation of an implicit pressure-based RANS code for naval hydrodynamics application. Technical Report PD CF 0018, National Aerospace Laboratories, Bangalore.
- Mueller, T. J. (1999). Aerodynamic measurements at low Reynolds numbers for fixed wing micro-air vehicles. In *RTO AVT/VKI special course, Rhode-St-Genèse, Belgium*.
- Mueller, T. J. (2001). Fixed and flapping wing aerodynamics for micro air vehicle applications. *Progress in Astronautics and Aeronautics*, 195.
- Mueller, T. J. and Batill, S. M. (1982). Experimental studies of separation on a two-dimensional airfoil at low Reynolds numbers. *AIAA Journal*, 20, 457–463.

- Noughabi, A. K. and Tadjfar, M. (2013). Cross wind influence on low aspect ratio wings at low Reynolds number. In *Proceedings of the ASME 2013 Fluids Engineering Division, Summer Meeting, Nevada, USA*.
- Null, W. and Shkarayev, S. (2005). Effect of camber on the aerodynamics of adaptive-wing micro air vehicles. *Journal of Aircraft*, 42, 1537–1542.
- Ol, M. V., McAuliffe, B. R., Hanff, E. S., Scholz, U., and Kalher, C. (2005). Comparison of laminar separation bubble measurements on a low Reynolds number airfoil in three facilities. In *35th AIAA Fluid Dynmaics Conf. and Exhibit, Toronto, Ontario Canada*, Number 2005-5149.
- O'Meara, M. M. and Mueller, T. J. (1987). Laminar separation bubble characteristics on an airfoil at low Reynolds number. *AIAA Journal*, 25, 1033–1041.
- Pal, K., Lalitha, J., Rajani, B. N., and Majumdar, S. (1995). Three dimensional grid generation using elliptic equations with direct control of mesh spacing and orthogonality near boundaries. Technical Report PD CF 9511, National Aerospace Laboratories, Bangalore.
- Patankar, S. V. (1980). *Numerical heat transfer and fluid flow*, Hemisphere Pub. Co.
- Patankar, S. V. and Spalding, D. B. (1972). A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flows. *International Journal of Heat and Mass Transfer*, 15, 1787–1806.
- Pelletier, A. and Mueller, T. J. (2000). Low Reynolds number aerodynamics of low-aspect-ratio thin/flat/cambered-plate wings. *Journal of Aircraft*, 37, 825–832.
- Peric, M. (1985). A finite volume method for the prediction of three-dimensional fluid flow in complex ducts. Ph. D. thesis, Imperial College, London University.
- Radespiel, R., Windte, J., and Scholz, U. (2006). Numerical and experimental flow Analysis of moving airfoils with laminar separation bubbles. In *44th AIAA Aerospace Sciences Meeting, Reno, NV*, Number AIAA Paper 2006-0501.

- Reid, M. R. and Kozak, J. (2006). Thin/cambered/reflexed airfoil development for micro air vehicle applications at Reynolds numbers of 60,000 to 100,000. In *AIAA Atmospheric Flight Mechanics Conference and Exhibit, Keystone, Colorado*, Number 2006-6832.
- Rival, D., Sharman, M., and Tropea, C. (2010). Numerical and experimental investigation of stall-onset and deep-stall conditions at low Reynolds numbers . In *AIAA Atmospheric FLight Mechanics Conference, Toronto, Ontario Canada, AIAA 2010-8125*.
- Rival, D. and Tropea, C. (2010). Characteristics of pitching and plunging airfoils under dynamic stall conditions. *Journal of aircraft* 1), 80–86.
- Rodriguez, I., Lehmkuhl, O., Borerell, R., and Oliva, A. (2013). Direct numerical simulation of a NACA0012 in full stall . *International journal of heat and fluid flow*, 43, 194–203.
- Sahin, M., Hall, J., and Mohseni, K. (2008). Direct numerical simulation of separated low-Reynolds number flows around an Eppler-387 airfoil. In *46th AIAA Aerospace Sciences Meeting and Exhibit*.
- Sathaye, S., Yuan, J., and Olinger, D. J. (2004). Lift distribution on low-aspect ratio wings at low Reynolds number for MAV applications . In *22nd AIAA Applied Aerodynamics Conference, Providence, RI*.
- Schmitz, F. W. (1989). *Aerodynamik des flugmodells* . C.J.E. Volckmann Nachf. E. Wette, Berlin-CHarlottenburg.
- Selig, Denovan, and Fraser (1989). *Airfoils at low speeds*, H A Stokeley publisher, Virginia Beach, Virginia.
- Selig, M. S., Guglielmo, J. J., Broeren, A. P., and Giguere, P. (1995). *Summary of low-speed airfoil data - Volume 1*, SoarTech Publications, Virginia Beach, Virginia.
- Selig, M. S. and McGranahan, B. D. (2004). Wind tunnel aerodynamic tests of six airfoils for use on small wind turbines . *Transactions of the ASME*, 126, 986–1001.

- Shetty, P., Subrahmanya, M. B., Kulkarni, D. S., and Rajani, B. N. (2013). CFD simulation of flow past MAV wings. *International J. of Aerospace Innovations*, 5(1), 19–27.
- Shome, B. and Radle, M. (2013). Assessment of transitional model for prediction of aerodynamic performance of airfoils at low Reynolds number regime. In *Conference paper in SAE Technical Papers*, doi:10.4271/2013-01-2314.
- Shyy, W., Berg, M., and Ljungqvist, D. (1999). Flapping and flexible wings for biological and micro air vehicles. *Progress in Aerospace Sciences*, 35(5), 455–505.
- Shyy, W., Klevebring, F., M.Nilsson, Sloan, J., Carroll, B., and Fuentes, C. (1999). Rigid and flexible low Reynolds number airfoils . *Journal of aircraft*, 36, 523–529.
- Singh, A. P. (2000). Computational study on airfoils at low Reynolds numbers. In *Preceedings of the ASME Fluids Engineering Division, FED-253,ASME*, pp. 405–411.
- Spalart, P. R. and Allamaras, S. R. (1992). A one-equation turbulence model for aerodynamic flows. *AIAA paper*, 92-0439.
- Stone, H. L. (1968). Iterative solution of implicit approximations of multidimensional partial differential equations. *SIAM Journal of Numerical Analysis*, 5, 530–530.
- Subrahmanya, M. B. and Rajani, B. N. (2017). Turbulent flow simulation of a 6:1 prolate spheroid . Technical Report NAL PD CF 1004, National Aerospace Laboratories, Bangalore.
- Subrahmanya, M. B. and Rajani, B. N. (2018). Transition prediction for flow over a MAV using the correlation based model. In 8th *Symposium on Applied Aerodynamics and Design of Aerospace Vehicles*.
- Subrahmanya, M. B., Rajesh, A., Kulkarni, D. S., and Rajani, B. N. (2015). Transitional flow past low *Re* airfoil. In 17th *annual AeSI CFD symposium*. NAL, Bangalore, India.
- Swift, K. M. (2009). An experimental analysis of the laminar separation bubble at low Reynolds numbers. Master's thesis, Thesis, University of Tennessee-Knoxville.

- Taira, K. and Colonius, T. (2009). Three-dimensional flows around low-aspect-ratio flat plate wings at low Reynolds numbers. *Journal of Fluid Mechanics*, 623, 187–207.
- Tang, L. (2006). RANS simulation of low-Reynolds-number airfoil aerodynamics. In *44th AIAA Aerospace Sciences Meeting*.
- Thomas, P. D. and Middlecoff, J. F. (1979). Direct control of the grid point distribution in meshes generated by elliptic equations. *AIAA Journal*, 18 (6), 652–656.
- Thompson, J. F., Thames, F. C., and Martin, C. W. (1974). Automatic numerical generation of body-fitted curvilinear coordinate system for fields containing any number of arbitrary two-dimensional bodies. *Journal of Computational Physics*, 15, 299–319.
- Torres, G. E., , and Mueller, T. J. (2004). Low-aspect-ratio wing aerodynamics at low Reynolds numbers. *AIAA Journal*, 42, 865–873.
- Torres, G. E. and Mueller, T. J. (2004). Low-aspect-ratio wing aerodynamics at low Reynolds numbers. *AIAA Journal*, 42, 865–873.
- Turner, C. (2012). Laminar kinetic energy modelling for improved laminar-turbulent transition prediction. Ph. D. thesis, University of Manchester.
- Uranga, A., P. O., P. M. D., and Peraire, J. (2009). Implicit large eddy simulation of transitional flows over airfoils and wings. In *19th AIAA Computational Fluid Dynamics, San Antonio, Texas*, Number 2009-4131, pp. 1–17.
- Vandoormal, J. P. and Raithby, G. D. (1984). Enhancement of the SIMPLE method for predicting incompressible fluid flows. *Numerical Heat Transfer*, 7, 147–163.
- Walters, D. K. and Cokljat, D. (2008). A three-equation eddy-viscosity model for Reynolds-Averaged Navier Stokes simulations of transitional flow. *J. of Fluids Engg.*, 130.
- Walters, D. K. and Leylek, J. H. (2002). A new model for boundary layer transition using a single-point RANS approach. *J. of Turbomachinery*, 126, 193–202.

- Wilcox, D. (1994). Simulation of transition with a two-equation turbulence model. *AIAA Journal*, 32, 247–255.
- Windte, J., Scholz, U., and Radespiel, R. (2006). Validation of the RANS-simulation of laminar separation bubbles on airfoils. *Aerospace Science and technology*, 10, 484–494.
- Winslow, J., Otsuka, H., Govindarajan, B., and Chopra, I. (2018). Basic understanding of airfoil characteristics at low Reynolds numbers ($10^4 - 10^5$). *Journal of Aircraft*., 55, 1050–1061.
- Xia, T., Dong, H., Yang, L., Liu, S., and Jin, Z. (2021). Investigation on flow structure and aerodynamic characteristics over an airfoil at low Reynolds number-A review. *AIP advances*, 11, 1–13.
- Yuan, W., Kalid, M., Windte, J., Scholz, U., and Radespiel, R. (2005). An investigation of low Reynolds number flows past airfoils. In *23rd AIAA Applied Aerodynamics Conference Toronto, Ontario Canada*, Number 2005-4607, pp. 1–19.
- Yuan, W., Xu, H., Khalid, M., and Radespiel, R. (2006). A parametric study of LES on laminar turbulent transitional flows past an airfoil. *International Journal of Computational Fluid Dynamics*, 20(1), 45–54.
- Zhou, Y. and Wang, Z. J. (2010). Implicit large eddy simulation of transitional flow over a SD7003 wing using high-order spectral difference method. In *40th Fluid Dynamics Conference and Exhibit, Chicago, Illinois*, Number 2010-4442.
- Zhu, J. (1990). A hybrid differential-algebraic method for three dimensional grid generation. *International Journal of Numerical Methods in Engineering*, 29, 1271–1279.
- Zhu, J. (1991). A low diffusive and oscillation-free convection scheme. *Communications in Applied Numerical Method*, 7, 225–232.
- Zilli, J., Sutton, D., and Lavoie, P. (2017). Effect of freestream turbulence on laminar separation bubbles and flow transition on an SD7003 airfoil at low Reynolds numbers. In *55th AIAA Aerospace Sciences Meeting*.

- Zuo, L. X. and Wang, J. J. (2010). Planform and flexibility on lift characteristics for flow over low-aspect ratio wings. *Journal of Aerospace Engineering*, 23, 55–61.



Appendix A

Study of Freestream Turbulence Intensity Influence at Low Re

A.1 Background

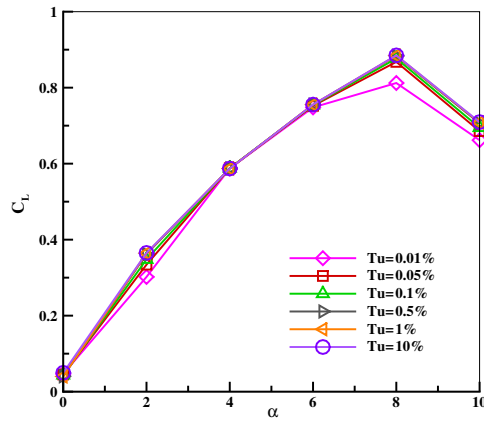
This chapter discusses the numerical study carried out in order to predict and understand the effects of freestream turbulence intensity (Tu) on the aerodynamic and transition characteristics of low Re SD7003 airfoil at Reynolds numbers 6×10^4 and 1.5×10^5 for a range of angles of attack. This study has been carried out using $\gamma - Re_\theta$ SST transition model by considering six different turbulence intensities ranging from 0.01% to 10%.

A.2 Effect of Tu on Aerodynamic Characteristics

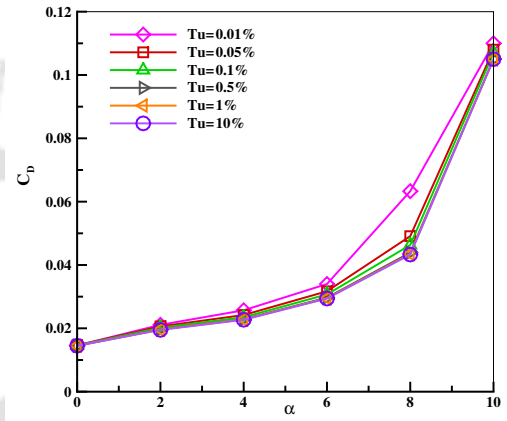
The variation of the aerodynamic characteristics in terms of lift coefficient (C_L), drag coefficient (C_D), lift to drag (L/D) ratio and moment coefficient (C_M) with angle of attack at the $Re = 6 \times 10^4$ and $Re = 1.5 \times 10^5$ for different turbulence intensities are shown in Fig. A.1 and Fig. A.2 respectively. It is observed (Fig. A.1) at lower Reynolds number ($Re = 6 \times 10^4$), lowering the freestream turbulence intensity is found to influence the aerodynamic characteristics. At this Reynolds number, maximum difference of about 9% in C_L and 29% in C_D is observed when Tu is reduced from from

0.05% to 0.01%. This difference is mainly found at lower angle of attack (between 2° to 4°) and close to stall angle ($\alpha = 8^\circ$). This difference is found to reduce when the Tu was further increased indicating that the aerodynamic characteristics are insensitive to the change in freestream turbulence intensity beyond 0.05%. The difference in variation C_L and C_D with Tu is highlighted in the lift to drag ratio. However when the Reynolds number was increased to $Re = 1.5 \times 10^5$ it is clearly evident (Fig. A.2) that the aerodynamic characteristics are not influenced by the freestream Tu .

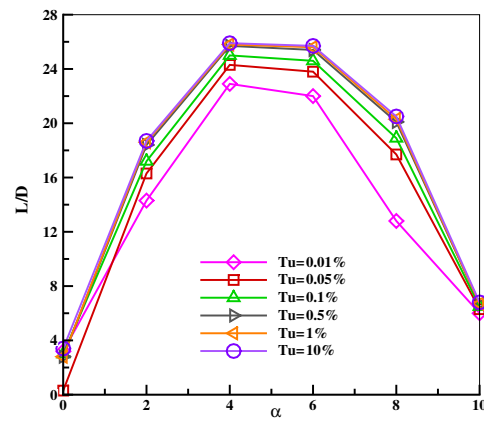
The influence of turbulence intensity on the surface pressure distribution at four different angles of attack for the two different Reynolds number are shown in Fig. A.3 and Fig. A.4 respectively. At lower Reynolds number ($Re = 6 \times 10^4$) similar to the the aerodynamic characteristics, enhanced difference in the pressure distribution curve was observed when the freestream turbulence intensity reduced below 0.05%. The difference in pressure distribution curve is mainly observed in the separation region on the airfoil surface which is close to trailing edge at lower angles of attack and shifts towards leading edge close to the stall angle. This difference in the separation region with angle of attack is further discussed in details in the following section. At higher Reynolds number, as expected, the freestream turbulence intensity is found have insignificant effect on the pressure distribution curve.



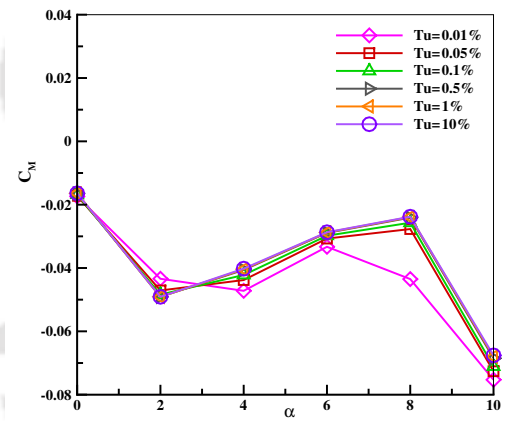
(a) Lift coefficient



(b) Drag coefficient

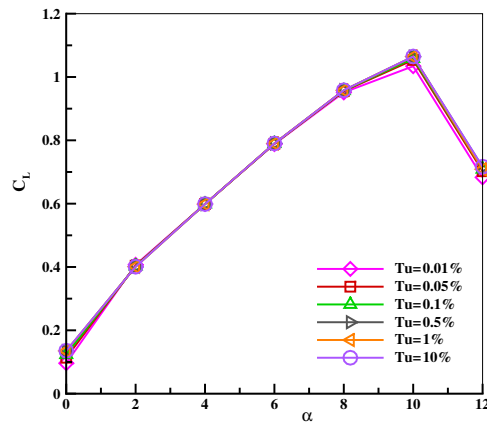


(c) Lift to Drag Ratio

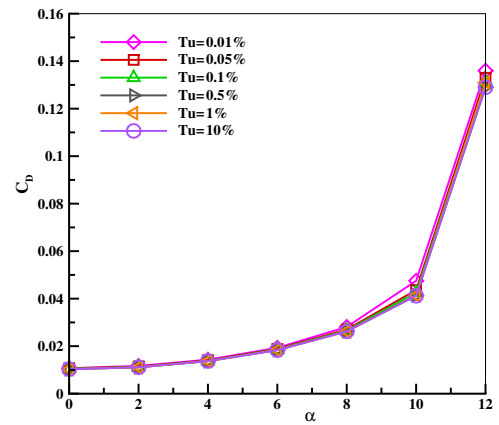


(d) Moment coefficient

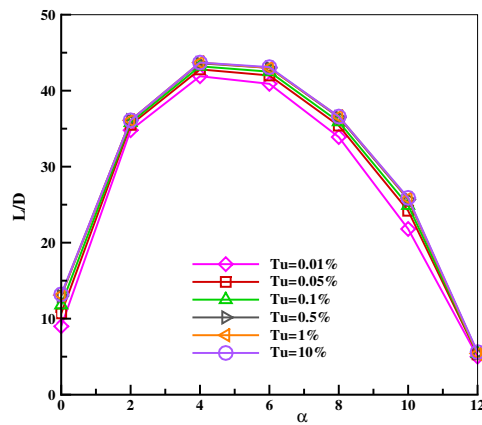
Figure A.1: Influence of Tu on aerodynamic characteristics at $Re = 6 \times 10^4$



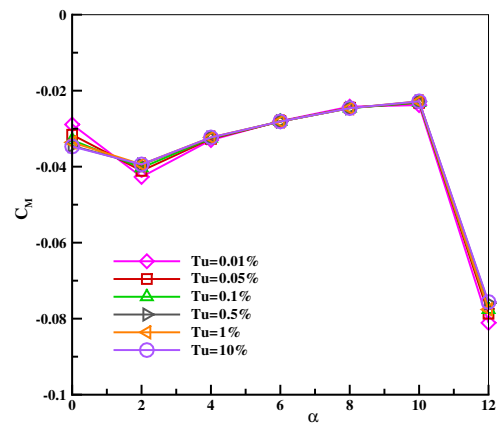
(a) Lift coefficient



(b) Drag coefficient

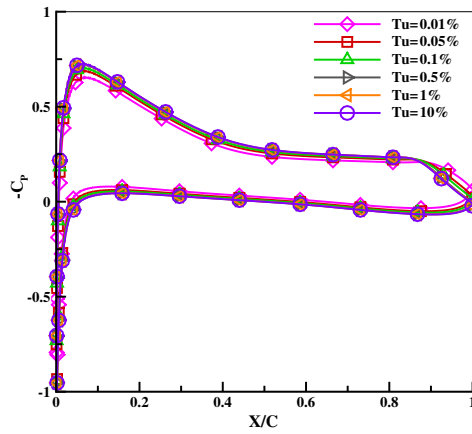


(c) Lift to Drag Ratio

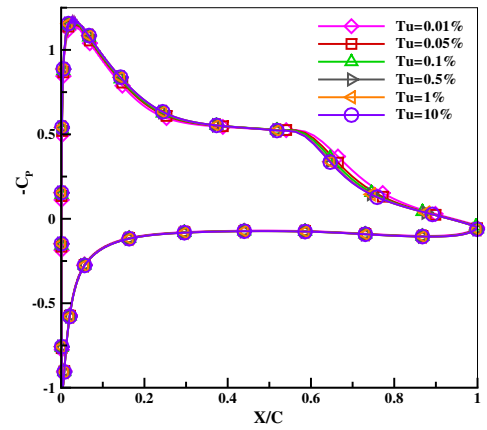


(d) Moment coefficient

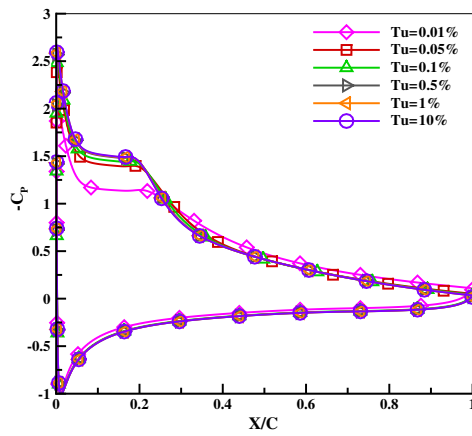
Figure A.2: Influence of Tu on aerodynamic characteristics at $Re = 1.5 \times 10^5$



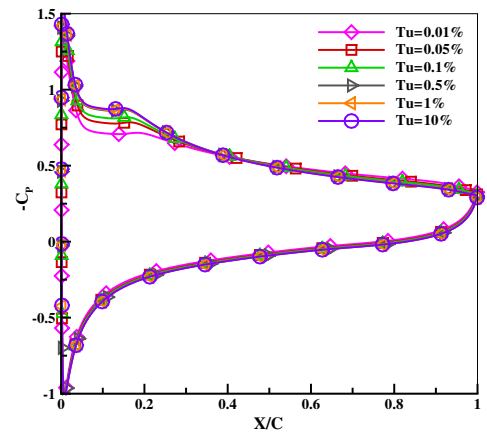
(a) $\alpha = 2^\circ$



(b) $\alpha = 4^\circ$

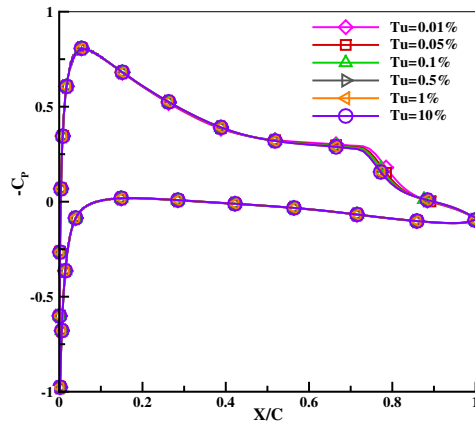


(c) $\alpha = 8^\circ$

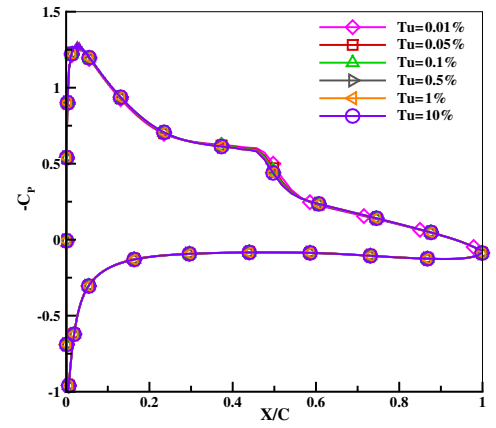


(d) $\alpha = 10^\circ$

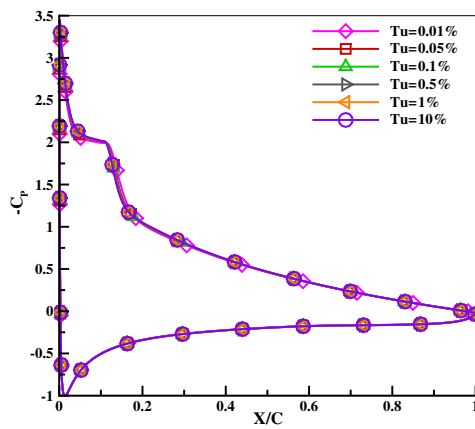
Figure A.3: Influence of Tu on C_p at different angle of attack for $Re = 6 \times 10^4$



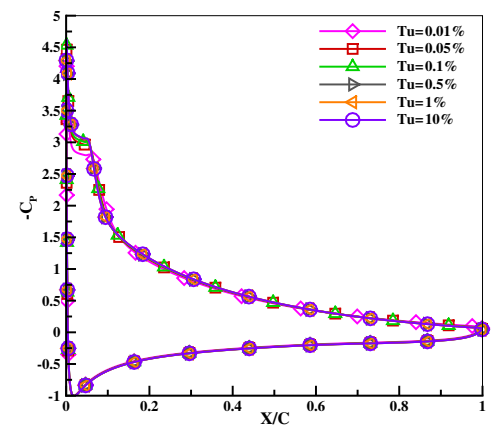
(a) $\alpha = 2^\circ$



(b) $\alpha = 4^\circ$



(c) $\alpha = 8^\circ$



(d) $\alpha = 10^\circ$

Figure A.4: Influence of Tu on C_p at different angle of attack for $Re = 1.5 \times 10^5$

A.3 Effect of Tu on Transition Characteristics

The influence of turbulence intensity on the transition onset location (X_T/C) at different angles of attack for $Re = 6 \times 10^4$ and $Re = 1.5 \times 10^5$ are shown in Fig. A.5. This figure indicates that the transition onset location is almost independent of freestream turbulence intensity for both the Reynolds number. However, for $Re = 1.5 \times 10^5$ at $\alpha = 0^\circ$ a slight difference is observed because at the lowest turbulence intensity the flow was found to be fully laminar unlike the other turbulence intensities analysed. On the other hand, for $Re = 6 \times 10^4$ the flow was observed to be fully laminar at $\alpha = 0^\circ$ for all the turbulence intensities considered. It is further observed that the transition onset location moves upstream of the airfoil with increase in angle of attack. As expected, the flow is found to be more laminar at the lower Reynolds number ($Re = 6 \times 10^4$) compared to the higher Reynolds number ($Re = 1.5 \times 10^5$). The transition onset at lower Reynolds number was delayed by about 10% of the chord on the airfoil surface when compared to the higher Reynolds number.

The effect of Tu on the formation of the laminar separation bubble (LSB) in terms of length of separation bubble, separation point (X_S/C) and reattachment point (X_R/C) at the two Reynolds number are shown in Fig. A.6 and Fig. A.7 respectively. From these figures it is clearly evident that lowering the turbulence intensity below 0.5% effected mainly the length of the LSB and reattachment point for both the Reynolds number with the difference being more prominent at the lower Re . The influence of the turbulence intensity was found to increase with angle of attack beyond $\alpha = 2^\circ$ especially at the lower Reynolds number. The maximum difference in length of LSB and reattachment point was about 11% at $\alpha = 8^\circ$ when Tu was changed from 0.01% to 0.05% and this difference reduced to 13% when Tu was changed from 0.1% to 0.5%.

The computed surface streamlines for $Tu = 0.1\%$ at four typical angles of attack for both the Reynolds number are shown in Fig. A.8 and Fig. A.9 respectively. For $Re = 6 \times 10^4$ the flow streamlines indicate that flow is fully laminar on the airfoil surface at $\alpha = 0^\circ$. This is observed for all the Tu analysed as indicated by the transition onset curve (Fig. A.5(a)). where the transition onset for $\alpha = 0^\circ$ is obtained beyond the trailing edge of the airfoil. The length of this fully laminar bubble was observed to be smaller compared to bubble formed by separated flow transition at $\alpha = 2^\circ$, beyond this angle of attack the length of LSB reduces as indicated in Fig. A.6(a). Both the

streamlines (Fig. A.8) and the curves of transition onset, separation point, and reattachment point (Fig. A.6) clearly indicates that beyond $\alpha = 0^\circ$ the flow undergoes separated flow transition as the transition onset occurs inside separation bubble before the reattachment point. The length of LSB decreases and the bubble moves upstream of the airfoil with angle of attack. Beyond $\alpha = 8^\circ$, the flow is almost fully turbulent as the transition occurs very close to the leading edge where the airfoil is observed to stall. The decrease in the length of the LSB with angles of attack beyond $\alpha = 0^\circ$ is because the transition point in the bubble shift upstream making the flow to be more turbulent leading to an early reattachment.

When the Reynolds number was increased to $Re = 1.5 \times 10^5$, both the streamlines and the transition onset location curve indicate that the separated flow transition occurs from $\alpha = 0^\circ$ itself except for $Tu = 0.01\%$. Similar to the lower Re , the transition onset location, length of the LSB and location of the the LSB moves upstream with angle of attack. However at $Re = 1.5 \times 10^5$, the strength of the separation bubble which is quantified by length and height of the LSB is lesser than that obtained at the lower Re . This reduction in strength of the LSB at the higher Re has resulted in lesser drag coefficient for all the turbulence intensities. Typically for $Tu = 0.1\%$, it was observed that the drag coefficient reduced by about 41 counts at $\alpha = 0^\circ$ and about 197 counts at $\alpha = 8^\circ$ compared to $Re = 6 \times 10^4$. It was further observed that the reduction in length of the LSB with angle of attack was steeper at $Re = 1.5 \times 10^5$ which is mainly due the faster flow reattachment on the airfoil. At this Re the flow obtained was more turbulent leading to a delay in the stall angle as compared to $Re = 6 \times 10^4$.

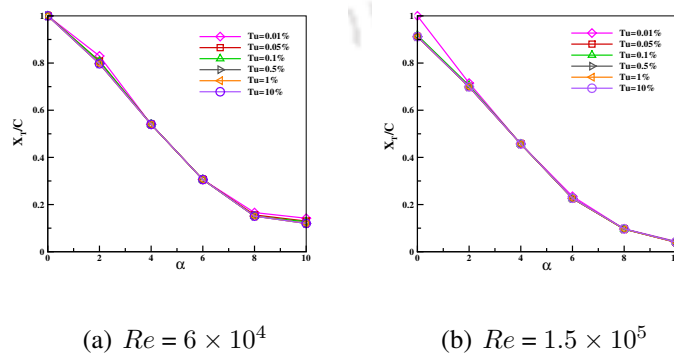
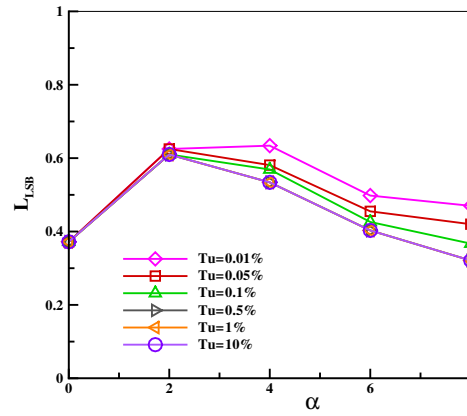
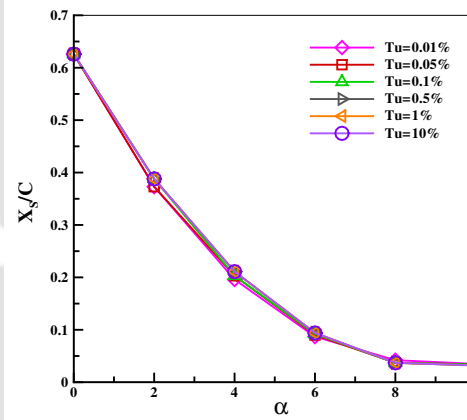


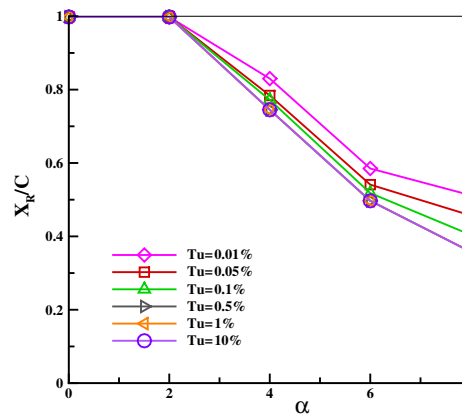
Figure A.5: Influence of Tu on transition onset location at different Reynolds number



(a) Length of LSB

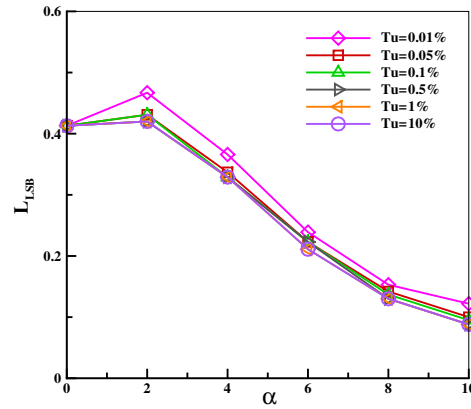


(b) Separation point

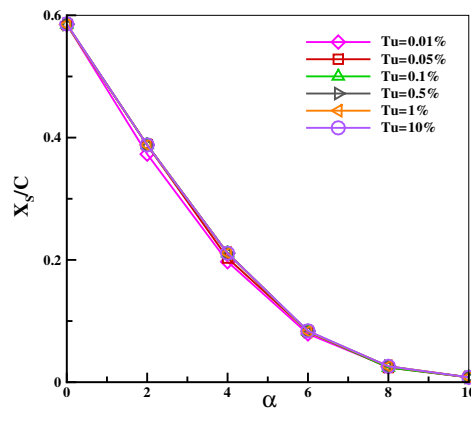


(c) Reattachment point

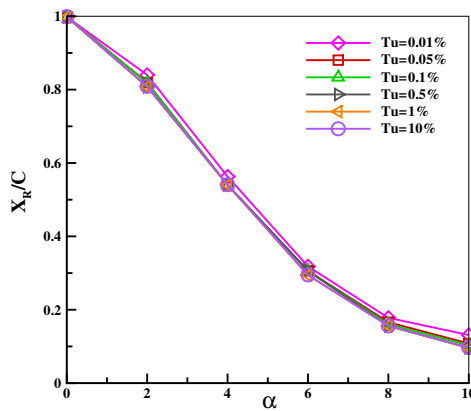
Figure A.6: Influence of Tu on LSB features at $Re = 6 \times 10^4$



(a) Length of LSB

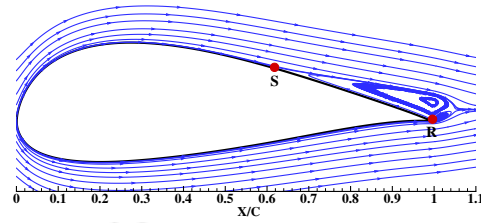


(b) Separation point

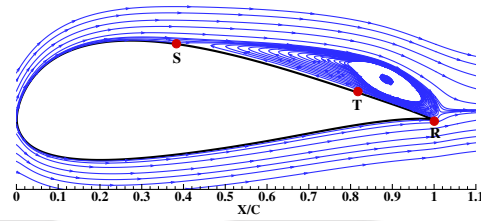


(c) Reattachment point

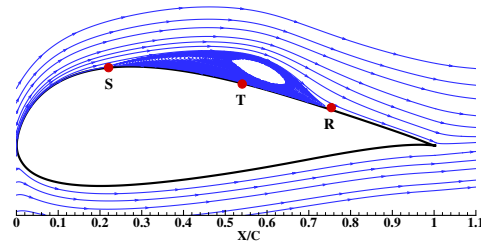
Figure A.7: Influence of Tu on LSB features at $Re = 1.5 \times 10^5$



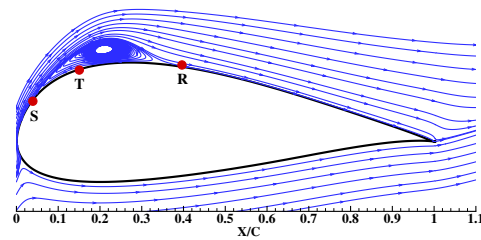
(a) $\alpha = 0^\circ$



(b) $\alpha = 2^\circ$

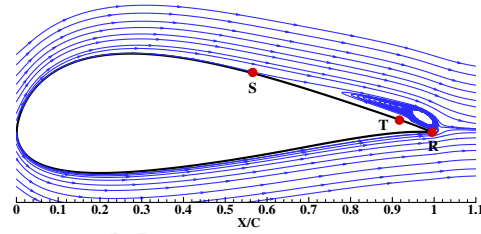


(c) $\alpha = 4^\circ$

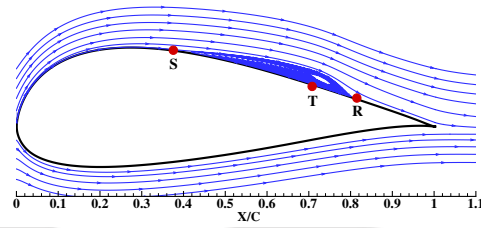


(d) $\alpha = 8^\circ$

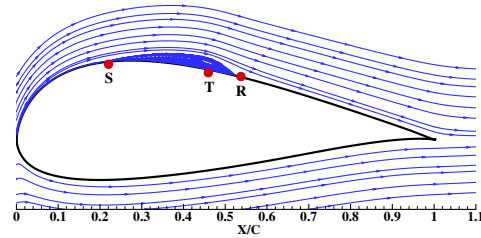
Figure A.8: Surface streamlines around SD7003 airfoil for $Tu = 0.1\%$ at $Re = 6 \times 10^4$
(S : Laminar separation point, R : Turbulent reattachment point, T : Transition onset location)



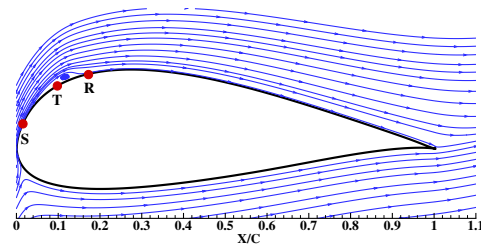
(a) $\alpha = 0^\circ$



(b) $\alpha = 2^\circ$



(c) $\alpha = 4^\circ$



(d) $\alpha = 8^\circ$

Figure A.9: Surface streamlines around SD7003 airfoil for $Tu = 0.1\%$ at $Re = 1.5 \times 10^5$
(S : Laminar separation point, R : Turbulent reattachment point, T : Transition onset location)

A.4 Summary

The effect of freestream turbulence intensity for flow past SD7003 airfoil were investigated for chord based Reynolds numbers of $Re = 6 \times 10^4$ and $Re = 1.5 \times 10^5$ at six different freestream turbulence intensities varying from 0.01% to 10%. These numerical investigations were performed using the $\gamma - Re_\theta$ SST transition model available in the in-house incompressible flow code 3D-PURLES. It was found that at $Re = 6 \times 10^4$, lowering the freestream turbulence intensity below 0.5% influenced the aerodynamic coefficients especially the drag coefficient at lower angles of attack. Further, it was noticed that at higher Reynolds number ($Re = 1.5 \times 10^5$), the aerodynamic coefficients were found to be insensitive to the change in the freestream turbulence intensity. The present study indicated that the transition onset location on the airfoil surface was insensitive to the change turbulence intensity for both the Reynolds number analysed. At all the angles of attack, it was observed that the transition onset was delayed at lower Reynolds number indicating the flow to be more laminar compared to higher Re . Similar to the transition onset location, the the separation point on the airfoil surface was not influenced by freestream Tu for both the Reynolds number considered. However, the freestream turbulence intensity below 0.5% was found to influence the length of LSB and reattachment point on the airfoil surface for both the Reynolds number with difference being more enhanced at lower Reynolds number.

Publications

Referred Books/Chapters

1. Dattatraya S. Kulkarni, B.N. Rajani, Vinayak Kulkarni, *Transitional flow past low Re SD airfoils with varying camber*. Fluid Mechanics and Fluid Power (Vol.2). Lecture notes in Mechanical Engineering, Springer Nature Singapore Pte Ltd. pp 333-338. April, 2023 (https://doi.org/10.1007/978-981-19-6970-6_56)

Referred Journals

1. Dattatraya S. Kulkarni, B. N. Rajani, Vinayak Kulkarni, *Numerical simulations of low aspect ratio wings at low Reynolds number*. Journal of Aircraft (Submitted)
2. Dattatraya S. Kulkarni, B. N. Rajani, Vinayak Kulkarni, *Performance of the transition models at low Reynolds number*. (To be submitted to Journal of Aerospace Engineering)

Referred National/Inter-National Conferences

1. Dattatraya S. Kulkarni, B. N. Rajani, Vinayak Kulkarni, *Numerical study of the flow characteristics of wings with low aspect ratio at low Reynolds number*, 10th International and 50th National conference on fluid mechanics and fluid power , FMFP-2023, Department of Mechanical Engineering, Indian Institute of Technology (IIT), Jodhpur, Paper No, FMFP2023-09-126, December 20-22, 2023
2. Dattatraya S. Kulkarni, B.N. Rajani, Vinayak Kulkarni, *Transitional flow past low Re SD airfoils with varying camber*, Proceedings, 48th National conference on fluid mechanics and

fluid power, FMFP-2021, Birla Institute of Technology and Science (BITS), Pilani, Paper No. FMFP2021-09-163, December 27-29, 2021

3. Dattatraya S. Kulkarni, B.N.Rajani, Vinayak Kulkarni, *Numerical simulation of transitional flow past SD7003 airfoil over a range of Reynolds number*, Proceedings, 21st Annual CFD Symposium, CFD Division of Aeronautical Society of India, CSIR-NAL, Bangalore, pp. 185-188, pp. 173-175, August 8-9, 2019
4. D. S. Kulkarni, B.N. Rajani, Vinayak Kulkarni, *Numerical simulation of low Reynolds number airfoils using correlation based $\gamma - Re_\theta$ SST model*, Proceedings, 7th International & 45th National conference on fluid mechanics and fluid power, FMFP-2018, Department of Mechanical Engineering, Indian Institute of Technology (IIT), Bombay, Paper No. 264, December 10-12, 2018

Publications as NAL Internal Documents

1. Dattatraya S Kulkarni, B. N. Rajani, Vinayak Kulkarni, *Effect of grid quality and flow solution control parameters on low Reynolds number airfoil*, PD/CTFD/2023/1007, April, 2023
2. Dattatraya S Kulkarni, B. N. Rajani., Vinayak Kulkarni, *Effect of Freestream turbulence intensity on the low Reynolds number airfoil SD7003*, PD-CTFD/2023/1001, February, 2023
3. Dattatraya S Kulkarni, B.N. Rajani, *Numerical study of low Reynolds number SD airfoil series with different camber and thickness*, PD-CTFD/2021/1026, September, 2021
4. Dattatraya S Kulkarni, M.B. Subrahmanya, B.N. Rajani, *Validation of the RANS based transition models for incompressible flows*, PD-CTFD/2019/1017, November, 2019