INFORMATION TO USERS

This manuscript has been reproduced from the microfilm master. UMI films the text directly from the original or copy submitted. Thus, some thesis and dissertation copies are in typewriter face, while others may be from any type of computer printer.

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleedthrough, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send UMI a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.

Oversize materials (e.g., maps, drawings, charts) are reproduced by sectioning the original, beginning at the upper left-hand corner and continuing from left to right in equal sections with small overlaps.

Photographs included in the original manuscript have been reproduced xerographically in this copy. Higher quality 6" x 9" black and white photographic prints are available for any photographs or illustrations appearing in this copy for an additional charge. Contact UMI directly to order.

**Bell & Howell Information and Learning
300 North Zeeb Road, Ann Arbor, MI 48106-1346 USA
800-521-0600**

UMI[®]

**NUMERICAL SIMULATIONS OF DIFFUSING S-DUCT AND
VORTEX-GENERATOR-JET FLOWS**

by

Allan D. Grosvenor

B. Eng. (Mechanical)

**A thesis submitted to the Faculty of Graduate
Studies and Research in partial fulfilment
of the requirements for the degree of**

Master of Engineering

in

Aerospace Engineering

**Ottawa-Carleton Institute for Mechanical
and Aerospace Engineering**

Department of Mechanical and Aerospace Engineering

Carleton University

Ottawa, Ontario

Canada

May, 2000

Copyright © 2000

Allan D. Grosvenor

**National Library
of Canada**

**Acquisitions and
Bibliographic Services**

**395 Wellington Street
Ottawa ON K1A 0N4
Canada**

**Bibliothèque nationale
du Canada**

**Acquisitions et
services bibliographiques**

**395, rue Wellington
Ottawa ON K1A 0N4
Canada**

Your file Votre référence

Our file Notre référence

The author has granted a non-exclusive licence allowing the National Library of Canada to reproduce, loan, distribute or sell copies of this thesis in microform, paper or electronic formats.

The author retains ownership of the copyright in this thesis. Neither the thesis nor substantial extracts from it may be printed or otherwise reproduced without the author's permission.

L'auteur a accordé une licence non exclusive permettant à la Bibliothèque nationale du Canada de reproduire, prêter, distribuer ou vendre des copies de cette thèse sous la forme de microfiche/film, de reproduction sur papier ou sur format électronique.

L'auteur conserve la propriété du droit d'auteur qui protège cette thèse. Ni la thèse ni des extraits substantiels de celle-ci ne doivent être imprimés ou autrement reproduits sans son autorisation.

0-612-52390-X

Canada

The undersigned recommend to
the Faculty of Graduate Studies and Research
acceptance of the thesis

**Numerical Simulations of Diffusing S-Duct and
Vortex-Generator-Jet Flows submitted by**

Allan D. Grosvenor

B. Eng. (Mechanical)

in partial fulfilment of the requirements for
the degree of Master of Engineering
in Aerospace Engineering

Dr. M.I. Yaras

Thesis Supervisor

Dr. R. Bell

Chair, Department of Mechanical and Aerospace Engineering

Carleton University

May, 2000

Abstract

This study evaluates several turbulence models and a particular *CFD* algorithm with emphasis on the prediction of flows related to high-performance aircraft engine intakes. The intake in question is a diffusing duct with an S-shaped centerline wherein vortex-generator jets are to be used for active flow control. Although the ultimate case would be to consider the instance where the vortex generator jets are installed in the S-duct, for the purposes of a controlled study, the present effort examines the plain S-duct and vortex-generator-jet flows separately. A couple of simpler flows precede the simulations of these more complex flows to establish or verify the basic parameter settings for the solution algorithm and the turbulence models.

The objective of this study is to establish whether or not the resultant combination of solution algorithm and turbulence model yields sufficiently accurate and fast calculation of diffusing-S-duct and vortex-generator-jet flows so as to allow its use in iterative design-cycle analysis of high-performance intakes. The turbulence models considered are a low-*Re* k - ε model utilising damping functions, a two layer k - ε model, a shear-stress-transport model, a k - ω model and a one-equation eddy viscosity model. Overall, the models are noted to provide fairly similar accuracy with the exception of the low-*Re* k - ε model. The minimum requirements for spatial resolution are found to differ, and these are highlighted in the context of grid optimisation studies. The solution algorithm based on explicit pseudo-time marching with implicit residual smoothing, multigrid acceleration, and preconditioning based on the artificial compressibility concept is found to provide favourable convergence and minimal numerical dissipation in all instances, and the required parameter setting to realise such results are presented.

to my family

Table of Contents

Abstract	iii
Table of Contents	iv
List of Figures	viii
List of Tables	xxiii
Nomenclature	xxvi
1.0 Introduction	1
1.1 Flow Control in Diffusing S-Ducts with Vortex-Generator Jets	1
1.2 Research Objectives	4
1.3 Literature Survey	5
1.3.1 Modern Intake Requirements	5
1.3.1.1 Modern Fighter Design	6
1.3.1.2 S-Duct Intake / Engine Compatibility	12
1.3.2 Diffusing S-Duct Aerodynamics	14
1.3.2.1 Inflow Conditions	18
1.3.2.2 Inflow Turbulence Level	19
1.3.2.3 Inflow Velocity Profile	21
1.3.2.4 Inflow Swirl	27
1.3.2.5 Diffuser Geometry	30
1.3.3 Vortex Flow Control	35
1.3.3.1 Guidelines for Vortex Control in S-Duct Intakes	36
1.3.3.2 Impeding Distortion	38
1.3.3.3 Counteracting Distortion	43
1.3.4 Vortex-Generator Jets	44
1.3.4.1 Longitudinal Vortex Structure	47
1.3.4.2 Wake and Horseshoe Vortex Structures	49
1.3.4.3 VGJ Streamwise Vorticity Field	50
1.3.4.4 Effect of Jet / Cross-stream Velocity Ratio on Performance	55

1.3.4.5 Effect of Jet Skew Angle Performance	59
1.3.4.6 Effect of Jet Pitch Angle on Performance	61
1.3.5 Turbulence Modelling for High Performance S-Duct Intakes	62
1.3.5.1 Turbulence Modelling in Straight Diffusers	63
1.3.5.1.1 Two-Equation Turbulence Models	64
1.3.5.1.2 Zero Equation Models	74
1.3.5.1.3 One Equation Models	75
1.3.5.1.4 Zonal Two-Equation Models	78
1.3.5.2 Curved Ducts	78
1.3.5.3 Vortex-Generator Jets	87
2.0 Description of Numerical Algorithm	91
2.1 Finite Volume Discretisation	91
2.2 Computational Molecule and Control-Volume Geometry	93
2.3 Artificial Dissipation	95
2.4 Integration in Pseudo-time	97
2.5 Convergence Acceleration Techniques	99
2.6 Artificial Compressibility	101
2.7 Procuring Convergence	102
3.0 Modelling of Turbulence	103
3.1 The k - ϵ Turbulence Model	105
3.1.1 Linearisation of Source Terms	106
3.1.2 Chien's Low-Re k - ϵ Model	107
3.1.3 Two-Layer k - ϵ Model	109
3.2 The k - ω Turbulence Model	111
3.3 Menter's Shear-Stress-Transport Turbulence Model	113
3.4 The Spalart-Allmaras Turbulence Model	116
3.5 Procuring Convergence	117

3.6 Turbulence Boundary Conditions	118
4.0 Verification of Turbulence Models	120
4.1 Simulations of a Flat Plate Boundary Layer	120
4.1.1 Sensitivity to Freestream Turbulence Parameters	122
4.1.2 Sensitivity to Artificial Compressibility	132
4.1.3 Sensitivity to Spatial Resolution, Artificial Dissipation and Wall Boundary Conditions	133
4.1.4 Flat-Plate Boundary Layer with Nonuniform Inflow Boundary Conditions	142
4.2 Simulations of Separating Flow in an Annular Diffuser	149
5.0 Simulations of Vortex-Generator-Jet Flow	159
5.1 Description of the Test Case	159
5.1.1 Boundary and Initial Conditions	163
5.2 Description of the Flow Physics	170
5.3 Grid Optimisation Studies	173
5.3.1 Classification of Grids	176
5.3.2 Results of the Grid Sensitivity Study	178
5.4 Effect of Turbulence Models on Simulation Results	187
5.5 Alternative Treatments of Jet-Orifice Boundary Conditions	192
6.0 Simulations of the M2129 Diffusing S-duct Flow	199
6.1 Description of the Test Case	199
6.1.1 Boundary and Initial Conditions	202
6.2 Description of the Flow Physics	206
6.3 Spatial Resolution	210
6.4 Effect of Turbulence Models on Simulation Results	217

7.0 Conclusions	223
8.0 Recommendations for Future Studies	227
Acknowledgments	231
Appendix A	232
A.1 References	232
A.2 Pseudo Computer Codes for Turbulence Models	252
Pseudo Code Nomenclature	253
A.2.1 The k-ϵ Turbulence Model	256
A.2.2 The k-ω Turbulence Model	262
A.2.3 Menter's SST Turbulence Model	267
A.2.4 The SA Turbulence Model	272
A.3 Grid Details	276

List of Figures

	Page
Figure 1.3.1.1.1 Defensive Counter Air (<i>DCA</i>) and Battlefield Air Interdiction (<i>BAI</i>) missions, reproduced from Bingaman (1992)	7
Figure 1.3.1.1.2 Future <i>MRF PST</i> vs. current F-16 transonic manoeuvring envelope, reproduced from Bingaman (1992)	9
Figure 1.3.1.1.3 Defensive <i>PST</i> manoeuvre, reproduced from Tamrat (1988)	9
Figure 1.3.1.1.4 Serpentine S-duct intake, a) Three view schematic, b) Installation in a modern concept <i>MRF</i> , reproduced from Bingaman (1992)	11
Figure 1.3.1.2.1 Effect of circumferential distortion on axial compressor stability, (nine stage, single spool), reproduced from Greitzer (1981)	13
Figure 1.3.1.2.2 Effect of circumferential distortion on compressor exit static pressure, a) number of sectors, b) sector angle, reproduced from Hall et al. (1996)	14
Figure 1.3.2.1 Flow-regime map for a 2-D plane-wall diffuser, reproduced from Sovran and Klomp (1967)	16
Figure 1.3.2.2 Performance map for a 2-D plane-wall diffuser, reproduced from Sovran and Klomp (1967)	17
Figure 1.3.2.2.1 Increase in diffuser pressure recovery with inflow turbulence intensity, reproduced from Hoffmann (1981)	20
Figure 1.3.2.3.1 Types of inflow velocity profiles	22
Figure 1.3.2.3.2 C_p Variation with B_1 for $AR=4$, reproduced from Klein (1981)	24
Figure 1.3.2.3.3 Two different inlet profiles with similar values of B_1 , a) step profile created by obstacle, b) thick turbulent boundary layer created by long inlet section	25

Figure 1.3.2.4.1 Flow regimes in a conical diffuser with increasing inlet swirl vorticity, reproduced from So (1967)	28
Figure 1.3.2.4.2 Observed pressure recovery performance enhancement with inlet swirl, a) no swirl b) with inlet swirl, reproduced from McDonald et al. (1971)	28
Figure 1.3.2.4.3 Ingested vortex trajectory in a F/A-18 inlet duct, reproduced from Anderson (1991)	29
Figure 1.3.2.5.1 Advanced fighter / attack aircraft serpentine S-duct intake, a) geometry, b) internal flow, reproduced from Mayer et al. (1998)	30
Figure 1.3.2.5.2 Secondary flow due to streamline curvature, reproduced from Tindell (1987)	31
Figure 1.3.2.5.3 Location of first appreciable stall in 2- <i>D</i> curved diffusers, reproduced from Fox and Kline (1962)	32
Figure 1.3.2.5.4 Spiral node indicating vortex lift-off, U. Tennessee oil flow- visualisation, reproduced from Anderson (1991)	34
Figure 1.3.3.1 Schematic of a vane <i>VG</i> , reproduced from Kunik (1986)	36
Figure 1.3.3.2.1 M2129 <i>VG</i> 170 vortex generator configuration, a) co-rotating <i>VG</i> geometry, b) <i>VG</i> spacing, c) S-duct, reproduced from Anderson and Gibb (1996)	39
Figure 1.3.3.2.2 Axial location of vortex generator sectors tested on the M2129, reproduced from Anderson and Gibb (1993)	40
Figure 1.3.3.2.3 Comparison of numerical and experimental M2129 performance results, a) total-pressure, b) engine face flow distortion, reproduced from Anderson and Gibb (1993)	41
Figure 1.3.3.2.4 Effect of <i>VG</i> vane height and axial location on engine face distortion, a) <i>VG</i> vane height, b) <i>VG</i> sector location, reproduced from Anderson and Gibb (1993)	41

Figure 1.3.3.2.5	Effect of <i>VG</i> sector and spacing angles on total pressure, a) <i>VG</i> sector angle, b) <i>VG</i> spacing angle, reproduced from Anderson and Gibb (1993)	43
Figure 1.3.3.3.1	Counteraction of inflow distortion using <i>VG</i> 's a) uncontrolled, b) non-optimum control, c) optimum control, reproduced from Anabtawi et al. (1999), (DC(60) normalised by inlet dynamic pressure)	44
Figure 1.3.4.1	The vortex-generator jet, reproduced from Pearcey, (1961)	46
Figure 1.3.4.2	Structure of a jet / crossflow interaction, reproduced from Chiu et al. (1993)	47
Figure 1.3.4.1.1	Pressure distribution on a cylinder in crossflow, reproduced from Panton (1996)	48
Figure 1.3.4.3.1	Schematic of a vortex-generator jet, reproduced from Barberopoulos and Garry (1998)	51
Figure 1.3.4.3.2	Observed <i>VGJ</i> vortex sense of rotation (looking downstream) by: a)Compton and Johnston (1992), b) Johnston and Nishi (1990)	51
Figure 1.3.4.3.3	Composite 3-D jet / crossflow interaction streamline pattern, reproduced from Kelso et al. (1996)	53
Figure 1.3.4.3.4	3-D flow field of a jet pitched at 45° and zero skew angle, reproduced from Barberopoulos and Garry (1998)	54
Figure 1.3.4.3.5	Surface flow topology of a jet pitched at 45° a) 0° Skew, $VR=1.2$, b) 60° Skew, $VR=1.4$, reproduced from Barberopoulos and Garry (1998)	55
Figure 1.3.4.4.1	Variation of dominant <i>VGJ</i> vortex path with VR for 45° pitch and 45° skew, reproduced from Barberopoulos and Garry (1998)	56
Figure 1.3.4.4.2	Variation in <i>VGJ</i> vortex normalised circulation with VR , 45° pitch and 45° skew, reproduced from Zhang, (1995)	56

Figure 1.3.4.4.3 Computed variation in <i>VGJ</i> skin friction coefficient distribution with <i>VR</i> for 45° Pitch and 45° Skew, a) <i>VR</i> =0.7, b) <i>VR</i> =2.0, c) <i>VR</i> =3.0, reproduced from Zhang and Collins (1993)	57
Figure 1.3.4.4.4 Vertical migration of vortex centreline with <i>VR</i> for 45° pitch and 45° skew, a) <i>VR</i> =0.7, b) <i>VR</i> =2.0, c) <i>VR</i> =3.0, reproduced from Zhang and Collins (1993)	58
Figure 1.3.4.4.5 Five <i>VGJ</i> configurations, reproduced from Johnston and Nishi (1990)	58
Figure 1.3.4.4.6 Percent jet fluid mass flow variation for five <i>VGJ</i> configurations, reproduced from Johnston and Nishi (1990)	59
Figure 1.3.4.5.1 Computed skin friction coefficient variation with skew angle, 45° Pitch and <i>VR</i> =1.0, a) 0°, b) 45°, c) 90°, reproduced from Zhang and Collins (1993)	60
Figure 1.3.4.6.1 Cross-stream skin friction variation with pitch angle for 60° skew and <i>VR</i> =1.0, reproduced from Henry and Pearcey (1994)	61
Figure 1.3.5.1.1.1 Comparison of skin friction predictions in zero ($m=0.0$) and adverse ($m=-0.20$) pressure gradient flow, ○ experiment for $m=0.0$, ● experiment for $m=-0.20$, x standard $k-\epsilon$ ($m=-0.20$ only), — low-Re $k-\epsilon$, -- One-equation, reproduced from Rodi et al. (1986)	66
Figure 1.3.5.1.1.2 Turbulent velocity profile in adverse pressure gradient, reproduced from White (1991)	67
Figure 1.3.5.1.1.3 Computed and measured C_f in strong adverse pressure gradient, ○ Expt., <i>JL</i> Jones & Launder low Re $k-\epsilon$, <i>LS</i> Launder & Sharma low Re $k-\epsilon$, <i>LB</i> Lam & Bremhorst low Re $k-\epsilon$, <i>CH</i> Chien low Re $k-\epsilon$, <i>YS</i> Yang & Shih low Re $k-\epsilon$, <i>FL</i> Fan, Lakshminarayana and Barnett low Re $k-\epsilon$, <i>WH</i> Wilcox $k-\omega$, <i>WL</i> Wilcox low Re $k-\omega$, reproduced from Wilcox (1993)	69

Figure 1.3.5.1.1.4 Conical diffuser velocity profile predictions, reproduced from Dudek (1996)	70
Figure 1.3.5.1.1.5 Conical diffuser predictions a) displacement thickness, b) momentum thickness, c) shape factor, d) skin friction coefficient, e) static pressure coefficient, reproduced from Dudek (1996)	71
Figure 1.3.5.1.1.6 Separating adverse-pressure-gradient flow (C.S0) of Driver and Johnston, <i>BL</i> =Baldwin-Lomax, <i>JK</i> =Johnson-King, <i>BB</i> =Baldwin- Barth a) velocity profiles, b) Reynolds Stresses, c) skin friction coefficient, C_f , d) static pressure coefficient, reproduced from Menter (1992b)	74
Figures 1.3.5.1.3.1 Separating adverse pressure gradient flow C.S0 of Driver and Johnston, a) velocity, b) skin friction, c) static pressure reproduced from Menter (1996)	77
Figure 1.3.5.2.1 Prediction of 2-D flow in a 180° duct, $\theta=30^\circ$, a), b), c) Reynolds stresses, d) streamwise velocity (H =duct height), reproduced from Luo and Lakshminarayana (1996)	80
Figure 1.3.5.2.2 Prediction of 90° rectangular duct, $\theta=45^\circ$, a), b), c) Reynolds stresses, d) streamwise velocity (H =duct height), reproduced from Luo and Lakshminarayana (1996)	81
Figure 1.3.5.2.3 Prediction of 90° rectangular duct, 1/2 duct height (H) downstream from bend, a), b), c) Reynolds stresses, d) streamwise velocity, reproduced from Luo and Lakshminarayana (1996)	81

Figure 1.3.5.2.4 Prediction of 90° rectangular duct, a) inner (convex) wall skin friction coefficient, b) outer (concave) wall skin friction coefficient (H =duct height, s =streamwise coordinate), reproduced from Luo and Lakshminarayana (1996)	81
Figure 1.3.5.2.5 Prediction of concave boundary layer development, $\theta=30^\circ$ a), b), c) Reynolds stresses, d) skin friction coefficient (H =duct height, R =surface curve radius, s =streamwise coordinate), reproduced from Luo and Lakshminarayana (1996)	82
Figure 1.3.5.2.6 Reynolds stress prediction of 2-D flow in a 180° U-duct, a) $x/H=-2$, b) 0° in bend, reproduced from Rumsey et al. (1999)	84
Figure 1.3.5.2.7 Prediction of 2-D flow in a 180° U-duct, 90° into the bend a) streamwise velocity, b) Reynolds stress, c) turbulence kinetic energy, reproduced from Rumsey et al. (1999)	85
Figure 1.3.5.2.8 Prediction of 2-D flow in a 180° U-duct, a) inner (convex) C_f , b) outer (concave) C_f , reproduced from Rumsey et al. (1999)	86
Figure 1.3.5.2.9 Prediction of 2-D flow in a 180° U-duct, a) streamwise velocity at streamwise coordinate $s/H=0.5$ (separated region), b) inner (convex) surface pressure distribution, c) outer (concave) surface pressure distribution, reproduced from Rumsey et al. (1999)	87
Figure 1.3.5.3.1.1 Configuration of compound-angle VGJ , reproduced from Hassan et al. (1998)	89
Figure 1.3.5.3.1.2 Turbulence kinetic energy for compound-angle VGJ , R =jet-to-freestream velocity ratio, x/D =streamwise distance normalised by jet diameter— <i>SST</i> , -- <i>Baseline $k-\epsilon/k-\omega$</i> , ·· <i>standard $k-\epsilon$</i> , ● <i>experiment</i> , reproduced from Hassan et al. (1998)	89

Figure 1.3.5.3.1.3 Jet fluid concentration gradient, C , for compound-angle VGJ , $R=VR$, x/D =streamwise distance normalised by jet diameter — <i>SST</i> , -- <i>Baseline $k-\epsilon$</i> / <i>$k-\omega$</i> , standard <i>$k-\epsilon$</i> , ● experiment, reproduced from Hassan et al. (1998)	90
Figure 1.3.5.3.2.1 Jet entrance grid distribution, a) perspective view, b) top view, reproduced from Kim and Benson (1992)	90
Figure 2.2.1 Cell-vertex-centred control volume a) 3D space, b) 2D space	94
Figure 4.1.1.1 Experimental flat plate k data, reproduced from Patel et al. (1985)	123
Figure 4.1.1.2 Flat-plate prediction based on Rodi's $k-\epsilon$ model - sensitivity to ϵ_{min} a) C_f -distribution, b) velocity profile at $(x/L=0.6)$, c) k profile at $(x/L=0.6)$	126
Figure 4.1.1.3 Flat-plate prediction based on the $k-\omega$ model - sensitivity to ω_{min} a) C_f -distribution, b) velocity profile at $(x/L=0.6)$, c) k profile at $(x/L=0.6)$	128
Figure 4.1.1.4 Flat-plate prediction based on Menter's <i>SST</i> model - sensitivity to ω_{min} a) C_f -distribution, b) velocity profile at $(x/L=0.6)$, c) k profile at $(x/L=0.6)$	130
Figure 4.1.1.5 Flat-plate prediction based on the <i>SA</i> model - sensitivity to $\chi_{min}=\tilde{\nu}_{min}/\nu$ a) C_f -distribution, b) velocity profile at $(x/L=0.6)$, c) χ profile at $(x/L=0.6)$	131
Figure 4.1.3.1 Flat-plate prediction based on Rodi's $k-\epsilon$ model - sensitivity to grid resolution a) C_f -distribution, b) velocity profile at $(x/L=0.6)$, c) k profile at $(x/L=0.6)$	136
Figure 4.1.3.2 Flat-plate prediction based on the $k-\omega$ model - sensitivity to grid resolution a) C_f -distribution, b) velocity profile at $(x/L=0.6)$, c) k profile at $(x/L=0.6)$	137

Figure 4.1.3.3 Flat-plate prediction based on Menter's <i>SST</i> model - sensitivity to grid resolution	
a) C_f distribution,	
b) velocity profile at ($x/L=0.6$) c) k profile at ($x/L=0.6$)	138
Figure 4.1.3.4 Flat-plate prediction based on the <i>SA</i> model - sensitivity to grid resolution	
a) C_f distribution, b) velocity profile at ($x/L=0.6$),	
c) χ profile at ($x/L=0.6$)	139
Figure 4.1.3.5 Flat-plate prediction based on the $k-\omega$ model - sensitivity to κ_+ (97×45 nodes, $y_1^+ = 3$)	
a) C_f distribution,	
b) velocity profile at ($x/L=0.6$), c) k profile at ($x/L=0.6$)	140
Figure 4.1.3.6 Flat-plate prediction based on Menter's <i>SST</i> model - sensitivity to β_{wall} (273×129 nodes, $y_1^+ = 1.25$)	
a) C_f distribution,	
b) velocity profile at ($x/L=0.6$), c) k profile at ($x/L=0.6$)	141
Figure 4.1.4.1 Flat-plate prediction based on Rodi's $k-\epsilon$ model - sensitivity to inflow boundary location (dashed lines correspond to earlier simulations that include the leading-edge region)	
a) C_f distribution, b) velocity profile at ($x/L=0.6$),	
c) k profile at ($x/L=0.6$)	144
Figure 4.1.4.2 Flat-plate prediction based on Chien's $k-\epsilon$ model - sensitivity to inflow boundary location	
a) C_f distribution, b) velocity profile at ($x/L=0.6$),	
c) k profile at ($x/L=0.6$)	145
Figure 4.1.4.3 Flat-plate prediction based on the $k-\omega$ model - sensitivity to inflow boundary location (dashed lines correspond to earlier simulations that include the leading-edge region)	
a) C_f distribution, b) velocity profile at ($x/L=0.6$),	
c) k profile at ($x/L=0.6$)	146

Figure 4.1.4.4 Flat-plate prediction based on the Menter's <i>SST</i> model - sensitivity to inflow boundary location (dashed lines correspond to earlier simulations that include the leading-edge region) a) C_f distribution, b) velocity profile at ($x/L=0.6$), c) k profile at ($x/L=0.6$)	147
Figure 4.1.4.5 Flat-plate prediction based on the <i>SA</i> model - sensitivity to inflow boundary location (dashed lines correspond to earlier simulations that include the leading-edge region) a) C_f distribution, b) velocity profile at ($x/L=0.6$), c) χ profile at ($x/L=0.6$)	148
Figure 4.2.1 Computational domain used for Driver and Johnston's C.S0 test case	149
Figure 4.2.2 Comparison of predictions with Driver and Johnston's experimental data, a) C_p distribution, b) C_f distribution	155
Figure 4.2.3 Comparison of predicted velocity profiles with Driver and Johnston's experimental data, a) $x/R_0 = -0.181$, b) $x/R_0 = 0.726$	156
Figure 4.2.4 Comparison of predicted velocity profiles with Driver and Johnston's experimental data, a) $x/R_0 = 2.177$, b) $x/R_0 = 3.266$	156
Figure 4.2.5 Comparison of predicted velocity profiles with Driver and Johnston's experimental data ($x/R_0 = 4.354$)	157
Figure 4.2.6 Comparison of predicted k profiles with Driver and Johnston's experimental data, a) $x/R_0 = -0.181$, b) $x/R_0 = 0.726$	157
Figure 4.2.7 Comparison of predicted k profiles with Driver and Johnston's experimental data, a) $x/R_0 = 2.177$, b) $x/R_0 = 3.266$	158
Figure 4.2.8 Comparison of predicted k profiles with Driver and Johnston's experimental data ($x/R_0 = 4.354$)	158
Figure 5.1.1 Computational domain for the <i>VGJ</i>	161
Figure 5.1.1.1 Distribution of V_j and k at the jet exit plane, a) Contours, b) Curves, $\circ$ Findlay expt. (1998), — Surface fit	168

Figure 5.2.1 Features of the <i>VGJ</i> flowfield, based on Menter's <i>SST</i> model	
a) Cross-flow velocity vectors (V, W) and U contours,	
b) Q_2 contours (negative values indicated by dashed lines),	
c) k contours, d) C_f and streamlines at first node off the wall	171
Figure 5.3.1 Grids employed for the <i>VGJ</i> computational domain,	
a) Grid A ($N_i=47 \times N_j=37 \times N_k=29$), b) Grid B ($59 \times 37 \times 29$),	
c) Grid C ($65 \times 37 \times 43$), d) Grid D ($69 \times 61 \times 43$)	174
Figure 5.3.2.1 Predicted velocity profiles (plane A_1) based on Rodi's $k-\epsilon$ model,	
sensitivity to grid resolution ◦ Findlay expt. (1998), ---- Grid A,	
- - - - - Grid B, - · - - - - Grid C, ——— Grid D	179
Figure 5.3.2.2 Predicted velocity profiles (plane A_2) based on Rodi's $k-\epsilon$ model,	
sensitivity to grid resolution ◦ Findlay expt. (1998), ---- Grid A,	
- - - - - Grid B, - · - - - - Grid C, ——— Grid D	180
Figure 5.3.2.3 Predicted velocity profiles (plane A_3) based on Rodi's $k-\epsilon$ model,	
sensitivity to grid resolution ◦ Findlay expt. (1998), ---- Grid A,	
- - - - - Grid B, - · - - - - Grid C, ——— Grid D	181
Figure 5.3.2.4 Predicted k profiles based on Rodi's $k-\epsilon$ model,	
sensitivity to grid resolution (planes A_1, A_2, A_3), ◦ Findlay expt. (1998),	
---- Grid A, - - - - - Grid B, - · - - - - Grid C, ——— Grid D . .	182
Figure 5.3.2.5 Comparison of velocity and k profiles (based on Rodi's $k-\epsilon$ model)	
with experimental data, a) W and b) k across shear layer,	
c) W and d) k across streamwise vorticity field -	
sensitivity to grid resolution	183
Figure 5.3.2.6 Predicted maximum streamwise vorticity in the jet-induced vorticity field,	
a) sensitivity to grids, b) effect of turbulence models (based on Grid D),	
c) sensitivity to jet boundary conditions (based on Grid D)	184

Figure 5.3.2.7 Predicted location of max. streamwise vorticity in the jet-induced vorticity field, a) comparison of grids, b) comparison of models (based on Grid D), c) comparison of jet boundary conditions (based on Grid D)	185
Figure 5.4.1 Predicted velocity profiles (plane A ₁ , based on Grid D) ◦ Findlay expt. (1998), --- SA model, - - - - Rodi's k - ϵ model, - - - - - k - ω model, — Menter's <i>SST</i> model	185
Figure 5.4.2 Predicted velocity profiles (plane A ₂ , based on Grid D) ◦ Findlay expt. (1998), --- SA model, - - - - Rodi's k - ϵ model, - - - - - k - ω model, — Menter's <i>SST</i> model	189
Figure 5.4.3 Predicted velocity profiles (plane A ₃ , based on Grid D) ◦ Findlay expt. (1998), --- SA model, - - - - Rodi's k - ϵ model, - - - - - k - ω model, — Menter's <i>SST</i> model	190
Figure 5.4.4 Predicted k profiles (planes A ₁ , A ₂ , A ₃ , based on Grid D) ◦ Findlay expt. (1998), - - - - Rodi's k - ϵ model, - - - - - k - ω model, — Menter's <i>SST</i> model	191
Figure 5.5.1 Comparison of velocity profiles (plane A ₁) with experimental data, sensitivity to jet boundary condition (results are based on Grid D) ◦ Findlay expt. (1998), — Menter's <i>SST</i> model, ---- SA model, - - - - Rodi's k - ϵ model, - - - - - k - ω model, (red curves: uniform jet-exit flow; black curves: nonuniform jet-exit flow)	194
Figure 5.5.2 Comparison of velocity profiles (plane A ₂) with experimental data, sensitivity to jet boundary condition (results are based on Grid D) ◦ Findlay expt. (1998), — Menter's <i>SST</i> model, ---- SA model, - - - - Rodi's k - ϵ model, - - - - - k - ω model, (red curves: uniform jet-exit flow; black curves: nonuniform jet-exit flow)	195

Figure 5.5.3	Comparison of velocity profiles (plane A_3) with experimental data, sensitivity to jet boundary condition (results are based on Grid D) ◦ Findlay expt. (1998), — Menter's <i>SST</i> model, ---- SA model, - - - - - Rodi's <i>k-ε</i> model, - · - · - · - <i>k-ω</i> model, (red curves: uniform jet-exit flow; black curves: nonuniform jet-exit flow)	196
Figure 5.5.4	Comparison of <i>k</i> profiles (planes A_1 , A_2 , A_3) with experimental data, sensitivity to jet boundary condition (results are based on Grid D) ◦ Findlay expt. (1998), — Menter's <i>SST</i> model, - - - - - Rodi's <i>k-ε</i> model, - · - · - · - <i>k-ω</i> model, (red curves: uniform jet-exit flow; black curves: nonuniform jet-exit flow)	197
Figure 5.5.5	Comparison of velocity and <i>k</i> profiles (Grid D) with experimental data; a) <i>W</i> and b) <i>k</i> across shear layer, c) <i>W</i> and d) <i>k</i> across streamwise vorticity field - sensitivity to jet boundary condition ◦ Findlay expt. (1998), — Menter's <i>SST</i> model, ---- SA model, - - - - - Rodi's <i>k-ε</i> model, - · - · - · - <i>k-ω</i> model, (red curves: uniform jet-exit flow; black curves: nonuniform jet-exit flow)	198
Figure 6.1.1	Computational domain for the M2129 diffusing S-duct	200
Figure 6.2.1	M2129 diffusing S-duct flow predicted with the SA model; ($\bar{U}=13.4$ m/s) a) streamwise velocity, b) streamwise vorticity, c) total pressure	206
Figure 6.2.2	Vortex lift-off in the M2129 diffusing S-duct	208
Figure 6.2.3	Total pressure losses and static-pressure recovery performance of the M2129 diffusing S-duct	209
Figure 6.3.1	Grids tested for the M2129 diffusing S-duct a) Grid A ($N_i=57 \times N_j=25 \times N_k=53$), b) Grid B ($57 \times 31 \times 61$), c) Grid C ($97 \times 25 \times 53$), d) Grid D ($97 \times 31 \times 61$)	211
Figure 6.3.2	Highly-skewed region of the computational grid for the M2129 S-duct .	213
Figure 6.3.3	Grid sensitivity of simulation results in Plane A_1 ($s/\ell=0.0$) a) distribution along t_1 , b) distribution along t_2	215

Figure 6.3.4	Grid sensitivity of simulation results in Plane A_2 ($s/\ell=0.24$)	
	a) distribution along t_1 , b) distribution along t_2	215
Figure 6.3.5	Grid sensitivity of simulation results in Plane A_3 ($s/\ell=0.54$)	
	a) distribution along t_1 , b) distribution along t_2	216
Figure 6.3.6	Grid sensitivity of simulation results in Plane A_4 ($s/\ell=0.76$)	
	a) distribution along t_1 , b) distribution along t_2	216
Figure 6.3.7	Grid sensitivity of simulation results in Plane A_5 ($s/\ell=1.0$)	
	a) distribution along t_1 , b) distribution along t_2	217
Figure 6.4.1	Predictions of the velocity field in Plane A_1 ($s/\ell=0.0$) compared with the experimental data of Yu (1991)	
	a) distribution along t_1 , b) distribution along t_2	219
Figure 6.4.2	Predictions of the velocity field in Plane A_2 ($s/\ell=0.24$) compared with the experimental data of Yu (1991)	
	a) distribution along t_1 , b) distribution along t_2	219
Figure 6.4.3	Predictions of the velocity field in Plane A_3 ($s/\ell=0.54$) compared with the experimental data of Yu (1991)	
	a) distribution along t_1 , b) distribution along t_2	220
Figure 6.4.4	Predictions of the velocity field in Plane A_4 ($s/\ell=0.76$) compared with the experimental data of Yu (1991)	
	a) distribution along t_1 , b) distribution along t_2	220
Figure 6.4.5	Predictions of the velocity field in Plane A_5 ($s/\ell=1.0$) compared with the experimental data of Yu (1991)	
	a) distribution along t_1 , b) distribution along t_2	220
Figure 6.4.6	Predicted wall static pressure compared with the experimental data of Yu	
	a) streamwise distribution along $t_1 = 0, t_2 = D/2$	
	b) streamwise distribution along $t_1 = D/2, t_2 = 0$	
	c) streamwise distribution along $t_1 = 0, t_2 = -D/2$	221
Figure 8.1	Schematic of a conceptual multiblock S-duct+VGJ computational domain	228
Figure 8.2	Achievable level of grid orthogonality using the multiblock approach	229

Figure A.3.1a Leading edge section of flat-plate grid A	278
Figure A.3.1b Grid distribution of flat-plate grid A, cell stretch distribution in i) j direction, ii) i direction; iii) y_1^+ distribution	278
Figure A.3.1c Layout of flat-plate grid A	278
Figure A.3.2a Leading edge section of flat-plate grid B	279
Figure A.3.2b Grid distribution of flat-plate grid B, cell stretch distribution in i) j direction, ii) i direction; iii) y_1^+ distribution	279
Figure A.3.2c Layout of flat-plate grid B	279
Figure A.3.3a Leading edge section of flat-plate grid C	280
Figure A.3.3b Grid distribution of flat-plate grid C, cell stretch distribution in i) j direction, ii) i direction; iii) y_1^+ distribution	280
Figure A.3.3c Layout of flat-plate grid C	280
Figure A.3.4a Leading edge section of flat-plate grid D	281
Figure A.3.4b Grid distribution of flat-plate grid D, cell stretch distribution in i) j direction, ii) i direction; iii) y_1^+ distribution	281
Figure A.3.4c Layout of flat-plate grid D	281
Figure A.3.5a Leading edge section of flat-plate grid E	282
Figure A.3.5b Grid distribution of flat-plate grid E, cell stretch distribution in i) j direction, ii) i direction; iii) y_1^+ distribution	282
Figure A.3.5c Layout of flat-plate grid E	282
Figure A.3.6a A streamwise segment of the C.S0 grid	283
Figure A.3.6b Cell stretch distribution of C.S0 grid, i) j direction, ii) i direction; iii) y_1^+ distribution	283
Figure A.3.6c Layout of C.S0 grid	283
Figure A.3.7a <i>VGJ</i> Grid A	284
Figure A.3.7b Grid distribution of <i>VGJ</i> Grid A, i) k direction, ii) i direction (at $y/D=0$); iii) y_1^+ along $y/D=0$	284
Figure A.3.7c Layout of <i>VGJ</i> Grid A	284
Figure A.3.8a M2129 Grid A	285

Figure A.3.8b Details of M2129 Grid A, i) j -direction cell stretch distribution,	
ii) k -direction cell stretch distribution, iii) skew,	
iv) i -direction cell stretch distribution, v) y_1^* distribution	285
Figure A.3.8c Layout of M2129 Grid A	285

List of Tables

Table 1.3.3.2.1	AGARD Case 3.1 - Test and initial conditions (1991)	40
Table 1.3.3.2.2	AGARD Case 3.2 - Test and initial conditions (1991)	40
Table 1.3.3.2.3	Optimum <i>VG</i> configuration for Case 3.2	42
Table 1.3.4.3.1	<i>VGJ</i> parameters	51
Table 1.3.5.1.1.1	Percentage error in prediction of C_f reproduced from Wilcox (1993), (refer to caption of Figure 1.3.5.1.1.4 for detailed identification) . . .	70
Table 4.1.1	Values of turbulence parameters specified at the inflow boundary	121
Table 4.1.1.1	Simulation parameters for the results shown in Figure 4.1.1.2a-c	126
Table 4.1.1.2	Simulation parameters for the results shown in Figure 4.1.1.3a-c	128
Table 4.1.1.3	Simulation parameters for the results shown in Figure 4.1.1.4a-c	130
Table 4.1.1.4	Simulation parameters for the results shown in Figure 4.1.1.5a-c	131
Table 4.1.3.1	Grids tested in flat plate simulations	134
Table 4.1.3.2	Simulation parameters for the results shown in Figure 4.1.3.1a-c	136
Table 4.1.3.3	Simulation parameters for the results shown in Figure 4.1.3.2a-c	137
Table 4.1.3.4	Simulation parameters for the results shown in Figure 4.1.3.3a-c	138
Table 4.1.3.5	Simulation parameters for the results shown in Figure 4.1.3.4a-c	139
Table 4.1.3.6	Simulation parameters for the results shown in Figure 4.1.3.5a-c	140
Table 4.1.3.7	Simulation parameters for the results shown in Figure 4.1.3.6a-c	141
Table 4.1.4.1	Simulation parameters for the results shown with solid lines in Figures 4.1.4.1a-c	144
Table 4.1.4.2	Simulation parameters for the results shown Figures 4.1.4.2a-c	145
Table 4.1.4.3	Simulation parameters for the results shown with solid lines in Figure 4.1.4.3a-c	146
Table 4.1.4.4	Simulation parameters for the results shown with solid lines in Figure 4.1.4.4a-c	147
Table 4.1.4.5	Simulation parameters for the results shown with solid lines in Figure 4.1.4.5a-c	148

Table 4.2.1	Minimum turbulence model requirements based on simulations of the flat-plate and Driver & Johnston's C.S0 flows	150
Table 4.2.2	Inflow boundary freestream turbulence settings	151
Table 4.2.3	Simulation parameters employed in the C.S0 test case, (same settings used for all models)	154
Table 4.2.4	Turbulence-model initial and threshold setting employed in the C.S0 test case	154
Table 4.2.5	Predicted separation and reattachment points compared with Driver and Johnston's experimental data	155
Table 5.1.1	Summary of <i>VGJ</i> parameters for <i>BL</i> control studied by several researchers	162
Table 5.1.1.1	Inflow boundary freestream turbulence settings	163
Table 5.1.1.2	Treatments of jet-exit boundary in published literature	166
Table 5.1.1.3	Turbulence-model initial and threshold settings	169
Table 5.1.1.4	Simulation parameters employed for the <i>VGJ</i>	169
Table 5.3.1.1	Grid details for the <i>VGJ</i>	176
Table 5.3.1.2	Comparison of domain sizes and node counts for several <i>VGJ</i> studies	177
Table 5.3.2.1	Uncertainties due to spatial resolution and artificial diffusion	186
Table 6.1.1	Location and angle of planes used in data comparison and boundary settings	200
Table 6.1.2	Basic dimensions of the M2129 diffusing S-duct	200
Table 6.1.1.1	Freestream values of turbulence parameters specified at the inflow boundary	203
Table 6.1.1.2	Turbulence-model initial conditions and threshold settings	205
Table 6.1.1.3	Simulation parameters employed for the M2129 diffusing S-duct	205
Table 6.3.1	Grid details for the M2129 diffusing S-duct	212
Table 6.3.2	Comparison with grids of published M2129 simulation studies	212
Table 8.1	Node counts for <i>VGJ</i> , S-duct and S-duct+ <i>VGJ</i>	229

Table A.3.1 Node locations in j direction of flat-plate grid A	
i) distribution at $x/L=0.6$, ii) variation of BL resolution	278
Table A.3.2 Node locations in j direction of flat-plate grid B	
i) distribution at $x/L=0.6$, ii) variation of BL resolution	279
Table A.3.3 Node locations in j direction of flat-plate grid C	
i) distribution at $x/L=0.6$, ii) variation of BL resolution	280
Table A.3.4 Node locations in j direction of flat-plate grid D	
i) distribution at $x/L=0.6$, ii) variation of BL resolution	281
Table A.3.5 Node locations in j direction of flat-plate grid E	
i) distribution at $x/L=0.6$, ii) variation of BL resolution	282
Table A.3.6 Node locations in the C.S0 grid	
i) inflow boundary, ii) region of separation	283
Table A.3.7 Node locations in j direction of VGJ grid A	
at inflow boundary	284
Table A.3.8 Node locations in M2129 grid A	
at the inflow boundary ($z = -0.5D_t, y = 0$)	285

Nomenclature

Abbreviations

<i>ADI</i>	Alternating-Direction-Implicit iterative solution scheme
<i>AIP</i>	Aerodynamic Interface Plane
<i>AOA</i>	Angle Of Attack
<i>AOSS</i>	Angle Of Side Slip
<i>AR</i>	Diffuser area ratio ($=A_2/A_1$)
<i>ASM</i>	Algebraic Stress Model
<i>BAI</i>	Battlefield Air Interdiction
<i>BC</i>	Boundary Condition
<i>BL</i>	Boundary Layer
<i>BSL</i>	Menter's Baseline turbulence model
<i>BVR</i>	Beyond Visual Range
<i>CFD</i>	Computational Fluid Dynamics
<i>CFL</i>	Courant-Freidrichs-Lewy number, used in calculating the inviscid (convective) time-step limit
<i>DC(θ)</i>	Distortion Coefficient of total pressure over engine face sector angle θ
<i>DCA</i>	Defensive Counter Air
<i>DERA</i>	Defence Evaluation and Research Agency, Bedford UK
<i>EFM</i>	Enhanced Fighter Maneuverability
<i>FAS</i>	Full-Approximation-Storage multigrid
<i>FS</i>	Flow Solver (a full Navier-Stokes solver developed by Dr. M.I. Yaras)
<i>LDA</i>	Laser-Doppler Anemometer
<i>LO</i>	Low-Observable
<i>LOS</i>	Line-Of-Sight
<i>MDC</i>	McDonnell Douglas Corporation
<i>MEMS</i>	Micro Electrical Mechanical Systems
<i>MRF</i>	Multi-Role Fighter
<i>MS</i>	Multi-Scale turbulence model

<i>MSIP</i>	Modified Strongly Implicit Procedure
<i>Na</i>	Node of attachment
<i>NS</i>	Navier Stokes
<i>NASA</i>	National Aeronautics and Space Administration
<i>NBL</i>	Negative Bifurcation Line
<i>PBL</i>	Positive Bifurcation Line
<i>PDE</i>	Partial Differential Equation
<i>PST</i>	Post-Stall Technology
<i>RaNPAS</i>	Rapid Nose Point And Shoot
<i>RCS</i>	Radar Cross-Section
<i>RMS</i>	Root Mean Square
<i>RSM</i>	Reynolds Stress Model
<i>SA</i>	Spalart-Allmaras (1992) one-equation turbulence model
<i>SAM</i>	Surface-to-Air Missile
<i>SIP</i>	Strongly Implicit Procedure
<i>Ss</i>	Saddle of separation
<i>SST</i>	Shear-Stress-Transport turbulence closure of Menter
<i>VG</i>	Vortex Generator
<i>VGJ</i>	Vortex-Generator Jet
<i>VR</i>	Jet-in-crossflow velocity ratio (jet/crossflow)
<i>WG</i>	Wave Guide

English Symbols

a	constant used in conjunction with C_{AD} in the artificial compressibility scheme
a_s	a parameter used in the implementation of a less reflective outflow boundary condition (Eqn. 4.2.1)
A	cross sectional area (m^2)
$A_{c.v.}$	surface area of control volume
A_ε	damping-function constant employed in Chen & Patel's algebraic ε equation (=5.08)
A_μ	damping-function constant employed in determining ℓ_μ in Chen & Patel's (=70) and Rodi et al.'s (=50.5) algebraic ε approach
B_i	diffuser inflow (throat) blockage factor
C_{AD}	factor employed in the artificial compressibility scheme (m/s)
C_{b1}	SA turbulence closure coefficient (=0.1355)
C_{b2}	SA turbulence closure coefficient (=0.622)
C_f	skin friction coefficient ($=\tau_w/(1/2\rho_{ref}V_{ref}^2)$)
C_p	diffuser pressure recovery coefficient
$\bar{C}_{p0}$	mass-averaged total pressure coefficient
C_{pi}	ideal diffuser pressure recovery coefficient
C_{v1}	SA turbulence closure coefficient (=7.1)
C_{w1}	SA turbulence closure coefficient (Eqn. 3.4.10)
C_{w2}	SA turbulence closure coefficient (=0.3)
C_{w3}	SA turbulence closure coefficient (=2.0)
$C_{\omega1}$	turbulence closure coefficient ($k-\omega$: $C_{\omega1}=0.553$; SST : Eqn. 3.3.6)
$C_{\omega2}$	turbulence closure coefficient ($k-\omega$: $C_{\omega2}=0.075$; SST : Eqn. 3.3.6)
$C_{\varepsilon1}$	$k-\varepsilon$ turbulence closure coefficient ($k-\varepsilon$,=1.44; Chien's low- Re $k-\varepsilon$,=1.35)
$C_{\varepsilon2}$	$k-\varepsilon$ turbulence closure coefficient ($k-\varepsilon$,=1.92; Chien's low- Re $k-\varepsilon$,=1.80)

C_μ	turbulence closure coefficient (= 0.09)
CD_{kw}	cross-diffusion function used in Menter's <i>SST</i> model (Eqn. 3.3.5)
d	component of the artificial dissipation term
D	representative diameter; jet-exit streamwise dimension (m)
D_a	artificial dissipation term
D_h	hydraulic diameter
δ_k	rate of destruction of turbulence kinetic energy
E	flux vector for the x direction
E	dissipation correction function used in Chien's k - ϵ model
f_μ	eddy-viscosity damping function used in Chien's k - ϵ model (Eqn. 3.1.2.9)
f_2	eddy-viscosity damping function used in the Chien's k - ϵ model
f_{v1}	<i>SA</i> turbulence closure function (Eqn. 3.4.4a)
f_{v2}	<i>SA</i> turbulence closure function (Eqn. 3.4.4b)
f_w	<i>SA</i> turbulence closure function (Eqn. 3.4.6)
F	flux vector for the y direction
F_1	blending function used in Menter's <i>SST</i> model (Eqn. 3.3.4)
F_2	shear-stress-transport limitation switch function used in Menter's <i>SST</i> model (Eqn. 3.3.10)
g	<i>SA</i> turbulence closure function (Eqn. 3.4.7)
G	flux vector for the z direction
H	turbulence source term (scalar)
H	turbulence source term (vector)
k	turbulence kinetic energy (m^2/s^2)
k_R	wall-roughness height (for smooth walls it is evaluated from Eqn. 3.6.4)
K_d	factor used in determining the viscous-diffusion time-step limit
ℓ	length of M2129 diffusing S-duct along centreline (m)
ℓ_ϵ	length scale employed in algebraic ϵ functions (Eqns. 3.1.3.2 and 3.1.3.4)
ℓ_μ	length scale for μ_t calculation (Eqn. 3.1.3.6)
L	diffuser length (m)

m	number of Runge-Kutta time integration stages
M	Mach number
n_{vg}	number of VG 's
$\mathbf{n}$	surface-normal vector
N_i, N_j, N_k	number of nodes in the i, j , and k grid directions, respectively
i, j, k	node counters in the the i, j , and k grid directions, respectively
P	static pressure (Pa)
P_{BC}	static pressure fixed at an outflow boundary
$\bar{P}$	mass-averaged static pressure (Pa)
P_0	total-pressure (Pa)
$\bar{P}_0$	mass-averaged total pressure (Pa)
P_{of}	mean total pressure at the engine face (Pa)
$P_{\theta\theta}$	mean total pressure in the distorted sector (Pa)
ρ_k	rate of production of turbulence kinetic energy
Q	vector of dependent variables (solution vector)
r	SA turbulence closure function (Eqn. 3.4.8)
R	residual, radius, jet-to-crossflow velocity ratio
R_0	hub radius in Driver and Johnston's annular-flow test section (=70mm)
R_s	radius of the streamline forming the "outer" boundary of the computational domain in Driver and Johnston's C.S0 test case
Re	Reynolds number
Re_y	turbulence Reynolds number (Eqn. 3.1.3.3)
s	streamwise distance; distance along centreline
S	magnitude of mean-flow strain rate
S_R	function used in obtaining wall value of ω in the $k-\omega$ model (Eqn. 3.6.3)
t	pseudo time (s)
t_1, t_2	curvilinear coordinates associated with crossflow plane in a duct
Tu	turbulence intensity (%)
u	fluctuating velocity component (m/s)

U	velocity component in the x direction (m/s)
$\bar{U}$	area-averaged velocity component in the x direction (m/s)
U_0	inlet freestream velocity in Driver and Johnston's diffuser test section (m/s)
U_e	boundary layer edge velocity
U_τ	friction velocity ($= (\tau_w/\rho)^{1/2}$) (m/s)
V	velocity component in the y direction (m/s)
$V_{c.v.}$	volume of control volume (m^3)
V_j	jet velocity (m/s)
$\bar{V}_j$	area-averaged jet velocity at the jet-exit plane (m/s)
V_{local}	local velocity magnitude (m/s)
V_{ref}	reference velocity (m/s)
V_s	velocity component in the streamwise (s) direction (m/s)
V_{t_2}	velocity component in the tangential (t_2) direction (m/s)
W	velocity in the z direction (m/s)
W_t	throat width of a 2-D diffuser (m)
x, y, z	cartesian spatial co-ordinates (m)
y_1	perpendicular distance from wall to first grid node off the wall (m)
y_{wall}	perpendicular distance from wall to grid node off the wall (m)

Greek Symbols

α	<i>MSIP</i> coefficient
α_{KE}	kinetic energy factor
α_i	Runge-Kutta integration coefficient for stage i
α_{θ}	<i>VG</i> angular (circumferential) spacing
β, β_r, β_k	coefficients used in the implicit residual smoothing operator
β_{wall}	coefficient used for ω_{wall} calculation in Menter's <i>SST</i> turbulence model (Menter=10, Hellsten=1.25)

γ	orientation of crossflow planes in an S-duct; angular position on surface of a cylinder (deg.)
δ	boundary layer thickness (m)
δ^2	second order differencing operator
δ^*	boundary layer displacement thickness (m)
δ_{ij}	Kronecker's delta function
ε	dissipation rate of turbulence kinetic energy (m^2/s^3)
ε_0	wall boundary condition for turbulence dissipation rate, used in Chien's low- Re k - ε turbulence model (m^2/s^3)
$\varepsilon_2, \varepsilon_4$	scaling parameters used in the calculation of artificial dissipation
$\tilde{\varepsilon}$	modified turbulence dissipation rate used in Chien's low- Re k - ε model (m^2/s^3)
η_p	total-pressure ratio
θ	diffuser half angle; vectored-jet skew angle from crossflow direction; distorted engine-face sector (deg.)
θ_s	circumferential extent (deg.) of duct perimeter containing vortex generators
κ	von-Kármán's constant (=0.41)
κ_2	scaling constant for second-order artificial dissipation term
κ_4	scaling constant for fourth-order artificial dissipation term
λ	spectral radius of the inviscid flux Jacobian
μ	dynamic laminar viscosity (Ns/m^2)
μ_t	Eddy-viscosity (dynamic) (Ns/m^2)
ν	kinematic laminar viscosity (m^2/s); also pressure switch as defined in Eqn. (2.3.5)
$\tilde{\nu}$	modified eddy-viscosity used in the SA turbulence model (m^2/s)
ρ	density of fluid ($=\rho_{ref}$)
σ	SA turbulence closure coefficient (=0.667)
σ_ε	k - ε turbulence closure coefficient (=1.3)

σ_ω	turbulence closure coefficient ($k-\omega$: $\sigma_\omega=2.0$; SST : Eqn. 3.3.6)
σ_k	turbulence closure coefficient ($k-\epsilon$: $\sigma_k=1.0$; $k-\omega$: $\sigma_k=2.0$)
τ_{ij}	ij component of the Reynolds stress tensor
τ_w	wall shear stress
Φ	vectored-jet-in-crossflow pitch angle from wall (deg.)
χ	nondimensional eddy-viscosity variable ($=\tilde{\nu}/\nu$) employed in the SA turbulence model
ω	specific dissipation rate of turbulence kinetic energy (1/s)
Ω	vorticity magnitude (1/s)
$\tilde{\Omega}$	modified vorticity used in the SA turbulence model (1/s) (Eqn. 3.4.3)

Subscripts

c	convective
$c.v.$	control volume
CL	centreline
d	diffusive
i,j,k	index counts in the three grid directions; x , y and z components of a quantity; also used in the Einstein notation for summation
$initial$	value set at the beginning of a simulation as a first guess
J	jet
l	counter used in the Einstein notation for summation
max	maximum value - inner: inner (convex) wall of curved duct
min	minimum value - outer: outer (concave) wall of curved duct
ref	reference quantity used in normalisation of variables
s	streamwise component
t	turbulence quantity
$wall$	value evaluated at the wall

x, y, z	x, y and z components
∞	freestream
θ	based on momentum thickness as the length scale
1	denotes inlet plane for diffusers
2	denotes exit plane for diffusers

Superscripts

$*$	normalised in terms of reference variables
$+$	normalised by a combination of ρ , $U\tau$ and ν

1.0 Introduction

1.1 Flow Control in Diffusing S-Ducts with Vortex-Generator Jets

Due to a number of factors, aircraft engine intake diffusion ducts are often longitudinally curved. The most common reason for the use of curved intakes is to accommodate geometrical constraints while minimising weight (e.g., the centre engine duct on the Boeing 727 and the *MDC* F-18 side intakes). Minimising radar cross-section (*RCS*) of fighter aircraft has placed a more recent need for curved intakes. In order to completely obscure the radar beam line-of-sight (*LOS*) from the highly reflective engine face, S-shaped ducts have been chosen for this application.

It is well known that the presence of longitudinal curvature in a duct imposes a cross-stream pressure gradient on the flow which results in the birth of a pair of longitudinal vortices. The development of the flow in the first bend of an S-duct follows this classic pattern. An added characteristic of diffusing S-duct flows is the phenomenon of vortex lift-off in the second bend of the duct. Vortex lift-off refers to the development of a pair of vortices orthogonal to the duct wall under the combined influence of adverse streamwise pressure gradient and the pair of streamwise vortices generated in the first bend. Flow distortion produced at the engine face by these vortical flows can result in asymmetric fan blade loading, hence reduced engine performance and compressor stall. The degree of distortion increases as a function of incidence at the inlet of the intake duct which places a severe restriction on maximum sideslip angle and angle of attack.

A common flow control technique for S-ducts is the installation of Vortex Generators (*VG*'s). These devices are used to produce streamwise vorticity to locally mix high-momentum freestream fluid with low-momentum fluid near the duct walls in order to delay separation due to adverse streamwise pressure gradients. *VG*'s can also be used to manipulate cross-stream flow distortion that may already prevail in the duct due to separation near the duct inlet, or due to wakes of objects positioned upstream of the duct. The shortcoming of standard *VG*'s is that the generated vorticity magnitude scales on the streamwise momentum of approaching fluid and the *VG* geometry. A single design point must therefore be chosen which imposes a penalty on off-design operating conditions. It is therefore desirable to develop a *VG* system that can be actively controlled.

VG's based on Micro Electrical Mechanical System (*MEMS*) technology and Vortex-Generator Jets (*VGJ*'s) are two options that have been proposed for the active flow control in S-duct intakes. Vortex strength is controlled by changing the incidence of the *MEMS-VG* in the former case or by varying the air-jet mass flow in the latter. Both devices have drawbacks, such as possible maintenance problems of *MEMS-VG*'s and the requirement of an air supply for the *VGJ*'s.

A series of experimental and numerical studies on a generic diffusing S-duct known as the M2129 have been conducted by *DERA* (Bedford, *UK*) and *NASA* Lewis Research Center (now, *NASA* Glenn Research Center) over the last two decades. Experimental studies of the M2129, including tests with *VG*'s and *VGJ*'s, have been performed by Willmer et al.

(1981), Gibb and Anderson (1995) and Anderson and Gibb (1996). The measurements taken in these studies consisted of wall static pressures and total-pressure distribution at the engine / intake Aerodynamic Interface Plane (*AIP*). Yu's (1991) experiments with the M2129 geometry is distinguished from other data sets in that detailed measurements of both velocity and turbulence quantities were performed throughout the M2129 duct. Computational studies of the M2129 including configurations with *VG*'s and *MEMS-VG*'s have been conducted by Anderson and Gibb (1992, 1993, 1996, 1998), Anderson et al. (1993), Lee and Yu (1994), Marquis and Ong (1995), Anderson et al. (1999) and Bender et al. (1999). The earlier computational studies (1992, 1993) were based on zero-equation turbulence models whereas more recent studies have shifted toward one- and two-equation models. Only few comparisons have been made with the detailed experiments of Yu (1991). Computational studies by Anderson et al. with *VG*'s and *MEMS VG*'s employed vorticity source terms to avoid resolving the vortex roll up downstream of these devices, and the results were compared to standard *VG* computations wherein the complete development of the streamwise vortices were resolved. These comparisons indicated the vorticity-source-term approach to provide adequate results for capturing the first-order effects of *VG*'s.

In addition to the above-noted experiments involving *VG*'s and *VGJ*'s installed in diffusing bends, numerous studies have been performed where *VGJ*'s were tested in isolation. Such fundamental *VGJ* studies began with the original work of Wallis (1952) and more recent efforts include those of Johnston and Nishi (1990), Compton and Johnston (1992), and

Pearcey et al. (1993). These studies concentrated on *VGJ* configuration effects on wall pressure distribution and redistribution of mass flow. Later studies such as those of Zhang (1995, 1998), Zhang and Rona (1999) and Khan (1999) have provided a more complete description of the velocity and turbulence field. Numerical studies such as those of Henry and Pearcey (1994) focussed on capturing the correct vortex path, whereas Alvarez et al. (1993), Zhang (1993a,b) and Zhang et al. (1996) evaluated the prediction accuracy with respect to details of the velocity and vorticity fields.

1.2 Research Objectives

Computational-Fluid-Dynamics (*CFD*) analysis is emerging as a potential method for performing design-cycle analysis of S-duct intakes, including flow control systems required to provide high performance. However, this tool is only useful if the prediction accuracy is acceptable, any inherent numerical and modelling errors are quantifiable, and the required simulation time is sufficiently short to allow speedy investigation of multiple configurations during design optimisation. In state-of-the-art *CFD* algorithms, turbulence modelling is the primary factor influencing solution accuracy. The present study therefore concentrates on the capabilities and requirements of several turbulence models for S-duct and *VGJ* flows. Due to unavailability of detailed experimental data for the ultimate configuration consisting of an S duct furnished with *VGJ*'s, and the desire to isolate the prediction accuracy related to specific features of these flows, the S-duct and *VGJ* flows are investigated separately. Issues such as grid resolution, artificial dissipation,

preconditioning techniques and convergence acceleration schemes are also thoroughly addressed, since they influence both solution accuracy and computational time.

1.3 Literature Survey

A literature survey has been conducted to summarise the published work on S-duct flows with emphasis on engine intakes and its control with *VGJ*'s. The goal of the present survey is twofold: first to review published research on S-duct intake design, relevant aerodynamic phenomena and vortex flow control, and second to review modelling of turbulence in predicting these flows. The survey is divided into five topics:

- i) Modern intake requirements
- ii) Diffusing S-duct aerodynamics
- iii) Vortex flow control
- iv) Vortex-generator jets
- v) Turbulence modelling for high performance S-duct intakes

1.3.1 Modern Intake Requirements

The air has become the battlefield of the 21st century. It is believed by most air service chiefs today that the first priority in any war or regional contingency is attaining air superiority (AW&ST, 1998). In particular, "the USA is relying heavily on thrust vectoring and stealth" capability to provide this air superiority (Jane's, 1987). Vectored thrust affords the latest fighters with a vast improvement in rapid nose pointing and

shooting (*RaNPAS*) which has been identified as a required capability to win in future aerial combat (Gal-Or, 1990). The manoeuvring envelope of these fighters has been substantially widened by enabling them to operate in the post-stall regime. The introduction of Post-Stall Technology (*PST*) has rendered standard concepts such as ‘spin-danger’ and ‘performance-stall-limit’ obsolete (Gal-Or, 1990).

In addition to the substantial increases in manoeuvrability of state-of-the-art fighters, a stringent requirement for low observability (*LO*) has been identified. These aircraft must perform missions within range of threat warning and tracking radars and accompanying surface-to-air-missile (*SAM*) sites (Bingaman, 1992). It is therefore desirable to minimise the radar cross-section (*RCS*) of such aircraft.

1.3.1.1 Modern Fighter Design

The modern multi-role fighter (*MRF*) must rely on “supercruise” and “manoeuvrable stealth” to maintain air superiority (Gal-Or, 1990). The term supercruise refers to supersonic flight without afterburn. Such a capability minimises fuel burn when the aircraft must travel long distances at supersonic Mach numbers without refuelling. Notional air-to-air Defensive Counter Air (*DCA*) and air-to-ground Battlefield Air Interdiction (*BAI*) *MRF* missions are depicted in Figure 1.3.1.1.1. Propulsion system sizing requirements for such missions are a sustained turn with a load factor of 4.7 at Mach 0.9, and Mach 0.9 to 1.6 acceleration in 60 seconds at 30,000 feet altitude.

Transonic and supersonic manoeuvring requirements defined by air combat segments of the *DCA* mission can be met by existing transonic fighters such as F-16 and F-18 (Bingaman, 1992). However, modern threat aircraft capabilities have extended the envelope of air-to-air combat. Examples include the Sukhoi Su-27 and Su-37, Mikoyan MiG-29 variants MiG-29SMT and MiG-35. These aircraft incorporate advances such as phased array multi-mode radar, helmet-mounted cueing systems, missiles with high off-boresight targeting capability and powerful multi-axis thrust vectoring engines (AIR International, July 1998).

Figure 1.3.1.1.1 Defensive Counter Air (*DCA*) and Battlefield Air Interdiction (*BAI*) missions, reproduced from Bingaman (1992)

Previously, fighters such as the F-15 relied on Beyond Visual Range (*BVR*) radar guided missiles to target threat aircraft before being detected by the enemy. Using *BVR* becomes much less effective when the opponent has equal or greater capability. In addition, current radar jamming techniques and the problem of positive target identification in populated theatres detract from the *BVR* advantage (Tamrat, 1988). Advanced threat aircraft capabilities clearly dictate a stealth requirement and a greater probability of close-in engagement. The latest fighter aircraft must therefore use Post-Stall Technology (*PST*) in low speed engagements to maintain air superiority. Figure 1.3.1.1.2 displays the drastic change in manoeuvring envelopes incurred by the advent of vectored thrust. The vast increase in angle of attack (*AOA*) and sideslip (*AOSS*) attainable at or below stall speed is referred to as “supermanoeuvrability”. This concept was developed in the late 70's in response to the advent of all-aspect targeting short-range missiles (Herbst, 1990). Before this development a threat aircraft could only be targeted from behind. With all-aspect targeting, two aircraft facing one another could launch missiles at each other simultaneously, thereby introducing the possibility of mutual destruction. Victory was now dependent on who launched the first missile and who was able to get out of firing range first. Both rate and radius-of-turn had to be optimised to provide this new manoeuvring requirement which led to the advent of *PST*.

Figure 1.3.1.1.3 displays the value of *PST* in defensive manoeuvring. The ability to vary attitude substantially without changing direction at high rates is seen to give the *PST* fighter a major targeting advantage. The *PST* defender begins within the firing envelope

of a conventional fighter. Rapidly pitching upward maximises aircraft deceleration thereby forcing the conventional fighter to overshoot into the defender's firing envelope.

Figure 1.3.1.1.2 Future *MRF PST* vs. current F-16 transonic manoeuvring envelope, reproduced from Bingaman (1992)

Figure 1.3.1.1.3 Defensive *PST* manoeuvre, reproduced from Tamrat (1988)

A joint flight testing program to explore super-maneuvrability began in 1990 at the *NASA* Dryden Flight Research Center (Herbst, 1990) on a new highly manoeuvrable aircraft.

The X-31 Enhanced Fighter Maneuverability aircraft (*EFM*) was subsequently developed by MBB Germany and Rockwell USA. This aircraft employed three thrust vectoring paddles aft of the single engine nozzle to produce multi-axis thrust vectoring.

The requirements of fighter engine intakes have changed as a result of *PST*. Herbst (1990) states that a *PST* intake configuration must allow full power engine operation at angles of attack up to 70° . The X-31 design facilitated this requirement by positioning the intake on the underside of the aircraft and employing a movable lower lip to allow air to enter the intake at post-stall condition with minimum disturbance. Incorporating stealth in addition to *PST* into an aircraft design makes optimization of the intake aerodynamics even more challenging. Low *RCS* is attained by blocking the engine face radar line-of-sight. One way to provide this is to offset the intake duct and engine face axes using an S-duct such that a radar beam from any viewing angle cannot directly contact the fan. Another method is to install a Wave Guide (*WG*) or Line-of-Sight (*LOS*) type of engine front frame (Bingaman, 1992). Engine front frames are engine-mounted or engine-integrated radar absorbing devices that hide the engine face from direct illumination by radar. These devices add extra weight and contribute to significant total-pressure losses observed at the engine face. Since the substantial streamline curvature in S-duct intakes can also result in significant losses, trade-off studies must be conducted for each specific aircraft to select the best intake system.

Figure 1.3.1.1.4 Serpentine S-duct intake,
 a) Three view schematic,
 b) Installation in a modern concept *MRF*,
 reproduced from Bingaman (1992)

Optimisation studies conducted by McDonnell Aircraft Co. and NASA Langley Research Center (Bingaman, 1992) identified a highly curved serpentine S-duct diffuser as a desirable compromise between *RCS*, weight and performance requirements. This intake

system, shown in Figure 1.3.1.1.4, provides 100% radar *LOS* blockage. It has been found that engine front frames are not required with this configuration. In addition to providing low *RCS*, the compactness of this design affords space under the curved section for landing gear and internal weapons storage.

1.3.1.2 S-Duct Intake / Engine Compatibility

Severe flow distortion at the engine face can be induced due to high inlet-flow incidence and may even occur in the cruise condition due to intake duct offset. Intake distortion is described by Greitzer (1981) as “a situation in which substantial total pressure, velocity, and/or flow angle (spatial) variation exists at the compressor inlet face.” The effect that intake distortion can have on engine performance is illustrated in Figure 1.3.1.2.1. The characteristic map of an axial compressor is given with undistorted flow conditions and a circumferential distortion condition superimposed. Compressor pressure ratio is plotted against corrected mass flow for varying rotor speeds. The maximum pressure ratio attained at a given rotor speed is dictated by the stall line. As the compressor is driven past this boundary, the flow will first exhibit rotating stall and ultimately full stall. The imposed circumferential distortion shifts the stall line towards lower pressure ratios, thereby substantially reducing the compressor stability margin. A full discussion of the effects of total-pressure distortion on compressor stability is beyond the scope of this study. The topic has been well documented by such authors as Emmons et al. (1955), Greitzer (1980, 1981), Bowditch et al. (1983), Cumpsty (1989), Williams and Surber (1993), and Hall et al. (1996).

Figure 1.3.1.2.1 Effect of circumferential distortion on axial compressor stability, (nine stage, single spool), reproduced from Greitzer (1981)

Circumferential distortion is measured in terms of the distortion coefficient $DC(\theta)$ as defined by Seddon and Goldsmith (1989):

$$DC(\theta) = \frac{P_{0f} - P_{0\theta}}{\frac{1}{2} \rho \bar{U}^2} \quad (1.3.1.2.1)$$

The distortion coefficient measures the difference between mean total-pressure, P_{0f} at the engine face and mean total-pressure in the distorted sector, $P_{0\theta}$ nondimensionalised by the dynamic pressure based on mean velocity. Hall et al. (1996) explained that, in addition to sector angle, the number of sectors also has a dramatic effect on compressor performance. Circumferential distortion is seen as a transient drop in total-pressure by the compressor blades. The authors suggested that distortion with lower harmonic content (fewer and larger spoiled sectors) is the most harmful type, as shown in Figure 1.3.1.2.2 (a). Low harmonic distortion tends to be the type that prevails in S-duct intakes, and is caused by

boundary layer separation and vortex lift-off. This suggests that flow control in such intakes should concentrate on reducing the impact of these particular flow phenomena on the overall duct flow.

Figure 1.3.1.2.2 Effect of circumferential distortion on compressor exit static pressure, a) number of sectors, b) sector angle, reproduced from Hall et al. (1996)

1.3.2 Diffusing S-Duct Aerodynamics

The extent of diffuser pressure recovery is dependent on several possible mechanisms and is expressed in nondimensional form as:

$$C_p = \frac{P_2 - P_1}{\frac{1}{2}\rho U_1^2} \quad (1.3.2.1)$$

The one-dimensional incompressible form of mass conservation dictates an inversely proportional relationship between streamwise velocity and the exit-to-inflow area ratio,

$AR=A_2/A_1$. Ideal pressure recovery, C_{pi} is derived by assuming no losses and considering such a velocity distribution, yielding:

$$C_{pi} = 1 - \frac{1}{AR^2} \quad (1.3.2.2)$$

Frictional losses and deviation from one-dimensionality of the flow (i.e. flow distortion) result in C_p/C_{pi} ratios less than one. This so-called diffuser effectiveness ratio was used by Sovran and Klomp (1967) to quantify diffuser efficiency. In engine intake design the total-pressure ratio (η_p) is the more common parameter used to quantify efficiency (Seddon and Goldsmith, 1989):

$$\eta_p = \frac{P_{01} - P_{02}}{P_{01}} \quad (1.3.2.3)$$

Diffuser efficiency can vary substantially depending on the flow condition within the diffuser. Kline et al. (1959) identified four regimes that can exist in two-dimensional diffusers. Figure 1.3.2.1 identifies these regimes as a function of diffuser length-to-inlet width ratio, L/W , and included angle, 2θ . Flow quality deteriorates as the flow shifts regimes from the bottom to the top of the chart. In the “no appreciable stall” regime the flow is uniform (except, of course, for the relatively thin boundary layers) and steady, whereas the region of transitory stall is unsteady and nonuniform. Fully-developed stall and jet-flow regimes are approximately steady but highly nonuniform. It is noted that the transitory stall regime contains two different lower bounds. The line of first stall denotes a diffuser geometry that yields the highest static pressure recovery. The diffuser in this case

is just at the verge of stalling which is characterised by small transient flow reversals.

Increase of either the diffuser length or the divergence angle will produce larger transient-separation regions which will result in a large reduction in the effective area ratio and hence pressure recovery. Although the figure addresses two-dimensional diffusers, the same behaviour exists in axisymmetric geometries.

Figure 1.3.2.1 Flow-regime map for a 2-D plane-wall diffuser, reproduced from Sovran and Klomp (1967)

Before an intake designer can correct for more complicated loss mechanisms such as vortex lift-off in S-ducts, the basic geometrical parameters must be chosen. A study conducted by Reneau et al. (1964) quantified the dependence of diffuser performance on geometry by plotting measured pressure recovery as a function of area ratio, length-to-inlet width ratio and diffuser half angle (Figure 1.3.2.2). Two lines of optimum pressure

recovery are defined. The C_p^* line identifies the minimum length required to obtain a desired pressure recovery whereas the C_p^{**} line denotes minimum area ratios.

Differentiating between the two values is useful when size restrictions must be considered in design. For instance, modern fighter aircraft configurations commonly require the engine intake ducts to be compact in length. In this case the C_p^* line would be used.

Figure 1.3.2.2 Performance map for a 2-D plane-wall diffuser, reproduced from Sovran and Klomp (1967)

In addition to geometry, the inlet displacement thickness, δ^* , is given. The condition of flow at the diffuser entrance has a strong influence on diffuser performance, and thus must be represented on such plots. One of the most complete sets of performance maps is the set of Dolan and Runstadler (1973) who studied a wide range of conical diffusers with varying geometry and inlet conditions. Specifically, these authors correlated diffuser

performance against divergence angle, length, inlet Mach number, Reynolds number and blockage factor. Amongst the inlet flow parameters considered, blockage factor, which relates to deviation of the inflow velocity distribution from a plug profile, was found to have a substantial effect on diffuser performance.

The following sections identify and describe the various factors that influence the performance of diffusers as they relate to flow in engine intakes. These discussions have been organised into two main topics relating to inflow conditions and diffuser geometry.

1.3.2.1 Inflow Conditions

When designing an aircraft engine-intake system there are several types of inflow conditions that must be considered. These include separation at the cowl lip during manoeuvring flight, separation of the duct boundary layer due to shock-wave / boundary layer interaction in supersonic flight, ingestion of fuselage boundary layer, and ingestion of vortices and wakes due to upstream disturbances (Anderson and Gibb, 1996). Different types of adverse effects can result from such disturbances. In manoeuvring flight, cowl lip separation can result in severe blockage if left untreated. Diffusion within the intake duct will amplify the separation resulting in a substantial reduction in pressure recovery and mass flow, ultimately leading to engine stall. The same effect of blockage will result from fuselage boundary layer ingestion. Upstream disturbances such as other aircraft or landing gear deployment can result in the ingestion of streamwise vortices. These can lead to total-pressure distortion at the engine face that can induce compressor stall. Similar

effects can result from the launch of missiles where the added adverse effect of temperature distortions would be present. The presence of adverse streamwise pressure gradient and streamline curvature in the intake duct tends to amplify the harmful effects of these types of distortions. The intake must therefore be designed to be compatible with the engine for such inflow conditions. Requirement of engine / intake compatibility guides the selection of duct geometry as well as possible flow-control devices. In order to make such decisions, the effects of each type of inflow condition must be well understood.

1.3.2.2 Inflow Turbulence Level

A study by Moore and Kline (1955) of two dimensional wide-angle diffusers showed the effect of two different levels of inflow turbulence on pressure recovery performance.

When a single rod was placed at the lip of a diffuser of 45° included angle, a previously existing separation region disappeared. As a result, the pressure recovery increased by approximately 92 percent. They observed that an increase in freestream turbulence did not change the divergence angle at the inception of transient stall. However, it was shown that the divergence angle could be increased past the original point of full stall with increased inlet turbulence intensity (Tu). Fox and Kline (1962) refined these results and found that increasing inflow turbulence raised the margin of first appreciable stall as well as the full-stall margin. It was found, however, that the line of first appreciable stall was only affected for low values of length-to-inlet width ratio.

Hoffmann (1981) found that an increase in inflow turbulence intensity on a non-separating 2-D diffuser produced a significant increase in pressure recovery. The increase in pressure recovery between the lowest and highest inlet freestream turbulence intensity was greater than 20 percent, as shown in Figure 1.3.2.2.1. In addition to the higher turbulence intensity, Hoffmann showed that larger turbulence length scale also increases pressure recovery. Cutler and Johnston (1981) suggested that a larger length scale would decrease the rate of decay of freestream turbulence with downstream distance. This would result in higher turbulence intensity downstream, and hence higher performance.

Figure 1.3.2.2.1 Increase in diffuser pressure recovery with inflow turbulence intensity, reproduced from Hoffmann (1981)

The conclusion of Hoffmann's (1981) study was that increased inlet freestream turbulence intensity significantly enhances mixing in the near wall region. This was also noted by Stevens and Williams (1980), who observed reduction in the rise of boundary layer shape factor with downstream distance with increasing freestream turbulence levels. Although the enhanced mixing would promote boundary layer growth, the profile would be fuller,

the net effect being reduced axial-flow distortion. It is noted in Figure 1.3.2.2.1 that pressure recovery increases rapidly between turbulence intensities of 2 and 4 percent. This trend does not continue at higher turbulence levels, however. The boundary layer has likely reached the maximum benefit in terms of reduced shape factor at a freestream turbulence intensity of 4 percent, yielding the maximum effective area ratio at this condition.

Similar performance improvements with increasing freestream turbulence have been observed in transitional diffuser flows (e.g., Klein, 1981). Cutler and Johnston (1981) explained that the increased pressure recovery in this case is due to earlier transition to turbulence rather than enhanced mixing. It should also be noted that the influence of inflow turbulence intensity on boundary layer velocity profiles may not be as evident in longitudinally curved diffusers. This was suggested by Yaras (1996) who stated that “secondary flows caused by the flow turning tend to have a dominating influence on the boundary layer.” It is clear, however, that knowledge of inflow turbulence level is important for the analysis of diffusers, and that incorrect specification of inflow turbulence in *CFD* simulations should be expected to adversely affect prediction accuracy.

1.3.2.3 Inflow Velocity Profile

The type of velocity profile existing at the diffuser entrance can have a strong influence on performance. Early studies by Horlock and Lewis (1961), Livesey and Turner (1964) and

Wolf and Johnston (1969) showed that the effect of inflow axial-flow distortion on diffuser performance cannot be reliably predicted by inviscid analysis. Masuda et al. (1971, 1975) demonstrated turbulent mixing to be a dominant mechanism in diffusers with such inflow conditions which is not captured by an inviscid treatment. Many researchers have therefore sought empirical correlations to represent the effect of inflow “quality” on diffuser performance.

Common types of velocity distortion are shown in Figure 1.3.2.3.1. The experiments of Wolf and Johnston (1969) included inlet velocity profiles with step shear. Diffuser performance under these conditions was observed to be lower than that of essentially uniform profiles. Regions of momentum deficit in the profiles were amplified by the diffusion process and in some cases, with low momentum regions located in the vicinity of the diffuser wall, separation resulted. The result of either effect was a lower effective area ratio, and hence lower pressure recovery.

Figure 1.3.2.3.1 Types of inflow velocity profiles

Kaiser and McDonald (1980) introduced wakes at the inlet of a planar diffuser and measured the effect on first appreciable stall. They found that the area ratio yielding first appreciable stall increases for wakes with low velocity deficit. Further increase of velocity deficit, however, resulted in a decrease of diffuser area ratio for first appreciable stall. For small velocity deficit the wake profile tended to retard boundary layer separation due to the high boundary layer edge velocity relative to mean velocity. As the velocity deficit was increased, the wake shear region eventually extended to the boundary layer thus contributing to its growth and ultimate separation. In addition, inviscid amplification of large wakes resulted in a pocket of stalled fluid at the diffuser centreline, which was also observed by Wolf and Johnston.

The throat area blockage factor, B_t , is commonly used to correlate inflow distortion effects on diffuser performance:

$$B_t = 1 - \frac{1}{A} \int \frac{U}{U_{max}} dA \quad (1.3.2.3.1)$$

Compressibility can be accounted for by including density in the integral, but the incompressible form is favoured because it can be measured or inferred more readily.

Dolan and Runstadler (1973) published a comprehensive set of conical diffuser performance maps that presents pressure recovery as a function of geometrical (AR , 2θ , L/D_1) and mean-flow parameters (Re_D , M), as well as the inlet blockage factor (B).

Dolan and Runstadler tested inflow conditions involving typical boundary-layer related axial flow distortion ($B_i = 0.03$ to 0.12) for which diffuser performance appeared to be well correlated with B_i . Klein (1981) revealed a problem with such correlations when he superimposed the results of several different researchers as shown in Figure 1.3.2.3.2.

Figure 1.3.2.3.2 C_p Variation with B_i for $AR=4$, reproduced from Klein (1981)

The diffuser of Winternitz and Ramsay (1957) suffers a substantial decrease in pressure recovery at blockages greater than 0.12 (with $2\theta = 10^\circ$, $L/D_i = 5.7$) whereas the pressure recovery of the diffuser tested by Cockrell et al. (1965) actually begins to increase at this point. This difference exists because two different methods were used to produce blockage, which was not distinguished by B_i since the parameter B_i only quantifies the amount of momentum deficit and not the location or extent of the affected region. Winternitz and Ramsay used annular screens whereas Cockrell et al. developed a thick boundary layer in a straight duct. The shape factor in the first case would therefore be

much higher (Figure 1.3.2.3.3) and earlier separation might be expected, thus explaining the trend in C_p . It is thus clear that a designer should not rely solely on performance maps based on inlet blockage factor to optimise the diffuser geometry.

Figure 1.3.2.3.3 Two different inlet profiles with similar values of B_p ,
 a) step profile created by obstacle,
 b) thick turbulent boundary layer created by long inlet section

In the review by Sovran and Klomp (1967) another parameter was introduced that accounted for distortion of the inlet velocity profile. This is the kinetic energy factor (α_{KE}) which is so-named because it represents the ratio of actual kinetic energy flux at a given cross-section to the minimum value that could exist at the given flow rate:

$$\alpha_{KE} = \frac{\int U^3 dA}{\bar{U}^3 A} \quad (1.3.2.3.2)$$

Increased nonuniformity of velocity results in an increase of excess kinetic energy flux and therefore increases α_{KE} . It is likely that α_{KE} would better describe the effect of inlet velocity profile distortion on pressure recovery, since this parameter readily appears in the definition of C_p based on average pressure and velocities (Klein, 1981). The incompressible form of this definition is expressed as:

$$C_p = \frac{\Delta \bar{P}}{\alpha_{KE1} \frac{1}{2} \rho U_1^2} = 1 - \frac{\alpha_{KE2} \overline{U_2^2}}{\alpha_{KE1} U_1^2} - \frac{\Delta \bar{P}_0}{\alpha_{KE1} \frac{1}{2} \rho U_1^2} \quad (1.3.2.3.3)$$

The value of α_{KE} is seldom quoted in the literature however, because calculating this parameter requires detailed measurement of the velocity profile, whereas blockage factor can be determined based on flow rate and the maximum velocity. An experimentalist must therefore have a good reason to justify the extra effort needed to quantify α_{KE} . With the numerous sources of inflow distortion and their possible ramifications on compressor performance, aircraft-intake design considerations (especially those of fighters) may provide this justification. Development of advanced intake performance maps based on α_{KE} would likely provide more consistent pressure recovery data for most inflow conditions.

1.3.2.4 Inflow Swirl

Researchers have observed adverse effects on performance due to inflow swirl (e.g., So, 1967, Anderson, 1991). The variety of results observed suggests that the flow field due to inlet swirl may pass through different states. Early studies of conical diffuser flow with inlet swirl performed by So (1967) suggested that this was indeed the case. Smoke flow-visualisation displayed five distinct flow regimes depending on the extent of swirl. These results are reproduced schematically in Figure 1.3.2.4.1. Swirl was produced by variable-angle guide vanes placed at the inlet. Vortex strength was adjusted by changing the vane incidence and diffuser mass flow rate.

Flow in regime 1 was laminar and a one-celled streamwise vortex existed at the inlet.

Regime 2 was characterised by transition to turbulence. Here a stagnant bubble of irregularly moving fluid existed between the laminar and fully turbulent regions. Further increase in vorticity resulted in fully turbulent flow in regime 3. In this third regime the transfer of momentum to the boundary layer by the spiralling core may yield an improvement in pressure recovery. This is likely the regime that McDonald et al. (1971) were operating in when they observed significant performance improvements of diffusers operating with inlet swirl (Figure 1.3.2.4.2). Further increase in swirl in So's experiment resulted in a two-celled vortex system forming in regime 4. In addition to streamwise vorticity, recirculation at the core in the axial direction was now present which extended down the entire length of the diffuser in regime 5.

Figure 1.3.2.4.1 Flow regimes in a conical diffuser with increasing inlet swirl vorticity, (vorticity increases from regime 1 to 5) reproduced from So (1967)

Figure 1.3.2.4.2 Observed pressure recovery performance enhancement with inlet swirl, a) no swirl b) with inlet swirl, reproduced from McDonald et al. (1971)

The cause of this streamwise recirculation in the core flow is explained by Armfield et al. (1989, 1990) among others. A positive radial pressure gradient is set up to balance centrifugal acceleration imposed by the streamwise vorticity. The radial pressure gradient decays with downstream distance due to diffusion of streamwise vorticity which results in a higher positive axial pressure gradient at the core than would exist with zero swirl. As the magnitude of streamwise vorticity is increased, the core fluid reaches a stage where there is no longer sufficient momentum to sustain forward flow. Anderson (1991) studied the effect of vortex ingestion in a F/A-18 inlet duct using *CFD*. He showed that strong inflow vorticity is notably preserved throughout the duct and results in substantial distortion at the engine face. Stream traces display the propagation of an ingested vortex from this study in Figure 1.3.2.4.3. In the same context, Wendt and Reichert (1996) observed vortex ingestion into a diffusing S-duct to aggravate vortex lift-off and result in substantially higher exit distortion.

Figure 1.3.2.4.3 Ingested vortex trajectory in a F/A-18 inlet duct, reproduced from Anderson (1991)

1.3.2.5 Diffuser Geometry

Diffuser geometries such as aircraft engine intakes can vary in centreline geometry as well as cross-sectional shape over the diffuser length. The most recent designs of fighter aircraft incorporate intakes with severe streamline curvature such as the type shown in Figure 1.3.2.5.1 (a). In this case the duct has been designed to yield total line-of-sight (*LOS*) blockage of the engine fan to radar as well as to minimise weight and maximise space for weapons storage etc. (Mayer et al., 1998). The effect of such severe geometry on the intake flow is shown in Figure 1.3.2.5.1 (b), where both vortex lift-off, and boundary layer separation due to adverse pressure gradient result.

Figure 1.3.2.5.1 Advanced fighter / attack aircraft serpentine S-duct intake, a) geometry, b) internal flow, reproduced from Mayer et al. (1998)

The incorporation of streamline curvature in engine intakes results in the fluid motion displayed in Figure 1.3.2.5.2. This cross-stream flow pattern is due to the well known imbalance between centrifugal forces and radial pressure gradients acting on the relatively low momentum fluid along the walls of the duct (e.g., Yaras, 1996). As seen in the figure, in the first bend fluid particles move from the side walls and along the inner (bottom) wall towards the geometric symmetry plane. The flow separates from the wall as it approaches the symmetry plane yielding a pair of counter-rotating streamwise vortices. In addition, the region of low momentum fluid centred along the inner wall is affected by streamwise adverse pressure gradient as well as diminished turbulence intensity due to the damping effect of convex longitudinal curvature. Both the secondary flows associated with the cross-stream vortices, and the axial nonuniformity of the flow are sources of significant flow distortion at the duct exit, hence the engine face.

Figure 1.3.2.5.2 Secondary flow due to streamline curvature, reproduced from Tindell (1987)

Early work by Fox and Kline (1962) defined flow regimes in curved two-dimensional diffusers for several divergence and turning angles. Figure 1.3.2.5.3 is a performance map generated from this study. They observed the same previously described three regimes that are known to exist in straight diffusers. As area ratio was increased, the flow shifted from a well-behaved (unseparated) state to transitory stall, and ultimately to full stall. In this case, however, approach towards first appreciable stall (line a-a) was influenced by turning angle, and consequently radius of curvature, as well as the conventional geometric parameters, i.e., AR (or 2θ) and L/W_1 . The effect of curvature was observed to be negligible for turning angles up to 30° .

Figure 1.3.2.5.3 Location of first appreciable stall in 2-D curved diffusers, reproduced from Fox and Kline (1962)

More recent studies of curved diffuser physics have shown varying results depending on the specific geometry tested. These studies have shown a strong sensitivity of performance to the secondary flow vortices and to the extent of streamwise curvature. Recent experiments by Yaras (1996, 1999) on a 90° curved *fishtail* diffuser of strong curvature showed that the cross-stream vortices actually helped to improve the diffuser performance as compared to that of an equivalent straight conical diffuser. It was explained that in this geometry the secondary flows had an energising effect in the stalled region along the inner wall of the bend, as well as a thinning effect on the boundary layers on the remaining walls of the bend. As a result, flow distortion at the outlet of the bend was reduced, thus increasing the effective area ratio. An early study of diffusing S-ducts by Bansod and Bradshaw (1972) suggested the same phenomena which they referred to as a “built-in vortex generator” effect. However, the presence of secondary flows in S-duct aircraft engine intakes have not been observed to enhance performance. In fact, the opposite has been the case. In the study by Bansod and Bradshaw, the authors identified a region of low total-pressure at the diffuser exit that had been observed previously in unpublished research by aircraft manufacturers.

A series of experimental and numerical studies conducted on the M2129 diffusing S-duct were mentioned in Section 1.1. These studies showed a decrease in pressure recovery performance in the region between the first and second bend and severe total-pressure distortion was measured at the exit. Experiments carried out at the University of Tennessee (described by Anderson, 1991) on a diffusing S-duct showed that cross-stream

vortices formed as expected in the first turn but did not continue in the same fashion in the second turn. Oil flow visualisation revealed two spiral nodes on the outer radius of the second bend (one side shown in Figure 1.3.2.5.4). This phenomenon is commonly referred to as “vortex lift-off” and seems to enhance the longitudinal vortices of the first bend resulting in a region of substantial flow distortion at the engine face as depicted in Figure 1.3.2.5.1. The internal velocity and turbulence field of the M2129 duct was measured by Yu (1991) using a Laser-Doppler Anemometer (*LDA*) which showed the evolution of the vortex lift-off phenomenon. Although this data set has not been used extensively for *CFD* prediction evaluation to date, it appears to be the best test case available to determine the capability of algorithms and turbulence models to capture such features of the duct flow field.

Figure 1.3.2.5.4 Spiral node indicating vortex lift-off, U. Tennessee oil flow-visualisation, reproduced from Anderson (1991)

Researchers have made several attempts to design S-duct intake cross-sectional geometries so as to minimize secondary flow (e.g., Lee and Boedicker, 1985). The most successful method to date is referred to as “Gerlach shaping,” after Gerlach and Schroeder (1969). Cross-section shape is varied using this technique by widening the duct wall on the inner radius of the bend and narrowing the wall on the outer radius (e.g., Figure 1.3.2.5.1). This results in a relative increase in velocity on the outer radius. Hence, static pressure is decreased at the outer radius, thereby opposing the original cross-stream pressure gradient. Although this method is successful in reducing secondary flow, vortex lift-off has still been observed in such advanced intake geometries (Mayer et al., 1998). In order to provide acceptable flow quality to the engine, flow control must therefore be used.

1.3.3 Vortex Flow Control

Classical Vortex Generators (*VG*'s) are small vanes set at an angle of attack to the freestream, as shown in Figure 1.3.3.1. The tip vortex of the vane enhances the intermixing of freestream and boundary layer fluids, thereby delaying potential separation of the boundary layer. Common aircraft applications of *VG*'s include separation control on lifting surfaces to increase maximum angle of attack, on aft-fuselage surfaces to reduce form drag, and in the engine intake to control flow distortion at the engine face.

Vortex-Generator Jet's (*VGJ*'s) are alternatives to solid vane-type *VG*'s. These devices produce a vortex through the mixing of a jet of fluid issuing from an opening on a solid

surface with the freestream. The advantage of *VGJ*'s is that generated vorticity may be varied from zero to large magnitudes by simply adjusting the jet mass flow. Since the fundamental purpose of either device (i.e., *VG* vs. *VGJ*) is the same, relevant design considerations will be discussed in the context of more thoroughly investigated solid *VG*'s. A description of *VGJ* physics will be provided at the end of this section.

Figure 1.3.3.1 Schematic of a vane *VG*, reproduced from Kunik (1986)

1.3.3.1 Guidelines for Vortex Control in S-Duct Intakes

This survey concentrates on Vortex Generator (*VG*) installations in S-duct intakes. For complete reviews of *VG* aerodynamics covering a broader range of applications the reader is referred to the works of Schubauer and Spangenberg (1960) and Bushnell (1992).

Many different types of *VG*'s and *VG* configurations have been tested for the purpose of diffuser flow control (e.g., Brown et al., 1968; Senoo and Nishi, 1974) resulting in the popularity of vane-type *VG*'s (Figure 1.3.3.1). Pearcey (1961) suggested that the first device of this kind was introduced by Bruynes and Taylor at United Aircraft Corporation in 1947. Taylor (1948) recommended placing the *VG*'s 10 to 30 boundary layer

thicknesses upstream of the separation point for optimum performance. The definition of a vortex generator given by Rae and Pope (1984) is *a low-aspect-ratio wing set at an angle of attack near 15°* . They noted that the span of the *VG* must be of the same order of magnitude as the local boundary layer thickness. This definition reflects the classical *VG* application to separation control where the vortices increase the streamwise momentum in the boundary layer. However, if thick boundary layers (e.g., Anabtawi et al. 1999a,b) or other large disturbances prevail near the inflow plane of the intake, or if large scale secondary motion has already established itself upstream of the *VG*'s, vortices that penetrate the boundary layer by a significant margin appear to be required (e.g., Zhang, 1995) to manipulate such pre-existing flow distortions.

The difference in vortex strength and location required for the two scenarios is due to two different effects expected from the *VG*'s. In the case where boundary layers are relatively thin and the core flow is mostly free of secondary motion or any other form of distortion, the *VG*'s prevent distortion from developing by transferring higher momentum fluid towards the wall to energise the boundary layer, and at the same time oppose any cross-stream motion that may be developing in the boundary layer. This application appears to be best satisfied by *VG*'s that are spaced uniformly and relatively closely about the duct circumference, sized to produce vortices that are of the order of the local boundary layer thickness. In the second case, where large distortion in the form of crossflow or axial velocity nonuniformity is present at the duct entrance spanning a notable portion of the duct cross-section, smaller vortices become ineffective. Rather than opposing the

development of distortion, as in the first case, the role of the *VG* in this case is to *remove* (or *mix-out*) pre-existing distortion. Relatively strong vortices that penetrate well beyond the boundary layer are required to achieve this goal.

1.3.3.2 Impeding Distortion

An example of the first case is the *NASA Lewis / DERA M2129* diffusing S-duct flow control program (Anderson and Gibb 1993, 1996; Gibb and Anderson, 1995). Figure 1.3.3.2.1 displays the M2129 and one of the *VG* configurations employed (denoted *VG170*). Parametric studies were undertaken to determine the optimum configuration of *VG*'s for a given set of inflow boundary conditions. These conditions are defined in the AGARD Working Group 13 report AR-270 (1991) and reproduced in Tables 1.3.3.2.1 and 1.3.3.2.2. The parametric studies were carried out using a parabolised *CFD* code employing a zero-equation turbulence model. *VG*'s were modelled using a vorticity-source boundary condition. Assuming parabolic flow and application of a zero-equation model can be expected to diminish prediction accuracy in the presence of strong adverse pressure gradient and streamline curvature. However, the parametric studies produced correct trends and reasonably accurate magnitudes in most cases. The effect of several parameters such as *VG* vane height, streamwise location within the duct, angle of sector containing *VG*'s and circumferential spacing were investigated for both Case 1 and 2, as well as for flight conditions. Sector angle (θ_s) and angular spacing (α_{VG}) in the circumferential direction are defined schematically in Figure 1.3.3.2.1 (b). The streamwise

positions investigated are shown in Figure 1.3.3.2.2. Comparison of engine face distortion ($DC(60)$) levels for varying blade heights and sector locations is given in Figure 1.3.3.2.3.

Figure 1.3.3.2.1 M2129 VG170 vortex generator configuration, a) co-rotating VG geometry, b) VG spacing, c) S-duct, reproduced from Anderson and Gibb (1996)

Total pressure	$P_{t_0} = 29.889 \text{ in./Hg}$
Total temperature	$T_{t_0} = 293 \text{ K}$
Throat Mach number	$M_i = 0.794$
Throat diameter	$D_i = 5.071 \text{ in.}$
Throat area	$A_i = 25.254 \text{ in.}^2$
Reynolds number (based on D_i)	$Re_y = 1.848 \times 10^6$

Table 1.3.3.2.1 AGARD Case 3.1 - Test and initial conditions (1991)

Total pressure	$P_{t_n} = 29.865 \text{ in./Hg}$
Total temperature	$T_{t_n} = 293 \text{ K}$
Throat Mach number	$M_i = 0.412$
Throat diameter	$D_i = 5.071 \text{ in.}$
Throat area	$A_i = 25.254 \text{ in.}^2$
Reynolds number (based on D_i)	$Re_y = 1.158 \times 10^6$

Table 1.3.3.2.2 AGARD Case 3.2 - Test and initial conditions (1991)

Figure 1.3.3.2.2 Axial location of vortex generator sectors tested on the M2129, reproduced from Anderson and Gibb (1993)

Figure 1.3.3.2.3 Comparison of numerical and experimental M2129 performance results, a) total-pressure, b) engine face flow distortion, reproduced from Anderson and Gibb (1993)

Figure 1.3.3.2.4 shows the optimum vane height to be sensitive to the inflow boundary conditions. This is to be expected since the optimum size of the VG 's correlates with the thickness of the BL being manipulated. It is noted that the optimum axial location is significantly downstream of the first bend (Figure 1.3.3.2.2). Due to diffusion and dissipation of the generated vorticity, the VG 's must be placed close enough to the location of separation / vortex lift-off to be effective.

Figure 1.3.3.2.4 Effect of VG vane height and axial location on engine face distortion, a) VG vane height, b) VG sector location, reproduced from Anderson and Gibb (1993)

The optimum parameters for Case 3.2 derived from these tests were incorporated into the configuration *VG130*, defined in Table 1.3.3.2.3. Using *VG130* as the baseline configuration, the effect of *VG* sector angle and lateral spacing were tested. A linear trend between flow distortion level and *VG* sector angle is evident in Figure 1.3.3.2.5 (a).

Figure 1.3.3.5 (b) suggests that approximately 11 *VG*'s evenly spaced over each symmetric half in the circumferential direction yields the lowest distortion levels in this case. It is noted that increasing the number of *VG*'s further is undesirable. This is due to the requirement of avoiding interaction of the vortices such that the low momentum fluid displaced from the surface by one vortex is not swept back by another.

Number of corotating generator pairs	$n_{vg} = 11$
Vortex generator sector location	$X_{vg}/R_i = 3.0$
Generator blade height	$h/R_i = 0.075$ ($h/\delta_i = 0.625$)
Generator chord length	$c/R_i = 0.2896$
Generator spacing angle	$\alpha_{vg} = 15.0$ deg
Generator vane angle of attack	$\beta_{vg} = 16.0$ deg
Generator sector angle	$\theta_i = 157.5$ deg

Table 1.3.3.2.3 Optimum *VG* configuration for Case 3.2

Figure 1.3.3.2.5 Effect of *VG* sector and spacing angles on total pressure, a) *VG* sector angle, b) *VG* spacing angle, reproduced from Anderson and Gibb (1993)

It is noted that co-rotating *VG*'s were used in these tests. Rae and Pope (1984) suggested that pairs of vortex generators with angles of attack to produce counter-rotating vortices are most effective in delaying (or ultimately preventing) separation. However, in counter-rotating installations the superposition of velocity vectors from closely spaced adjacent vortices tends to cause each vortex pair to leave the surface prematurely with downstream distance. Thus, the specific *VG* configuration that is optimum in a given case is strongly dependent on the individual application.

1.3.3.3 Counteracting Distortion

Anabtawi et al. (1999a,b) have studied *VG* distortion control of an S-duct intake ingesting a thick boundary. The optimum configuration, based on minimum DC(60), is compared to the uncontrolled intake in Figure 1.3.3.3.1. In this case, *VG*'s of height $h/\delta = 1/4$ and 1 were installed on the flat and curved surfaces of the duct, respectively. Vortices generated by the larger *VG*'s, that clearly penetrated the boundary layer, were required to redistribute the low momentum fluid around the engine face. As seen from the figure, the choice of *VG* configuration must be based on a trade-off between distortion and total-pressure loss, since achieving the lowest DC(60) resulted in the highest ΔC_{p_0} . The *VG* sizes used to prevent development of flow distortion versus those for reducing prevailing distortion indicates the need for a wide range of vortex strengths. This underlines the need for active vortex control in aircraft engine intakes.

Figure 1.3.3.3.1 Counteraction of inflow distortion using VG 's
a) uncontrolled, b) non-optimum control, c) optimum
control, reproduced from Anabtawi et al. (1999),
($DC(60)$ normalised by inlet dynamic)

1.3.4 Vortex-Generator Jets

In Section 1.3.3.3 it was shown that intake flow control can demand a substantial variation of vortex strength. Gibb and Anderson (1995) pointed out that this demand is particularly evident for manoeuvring combat aircraft. In this case, the required vortex strength to provide acceptable distortion levels at the engine face is dependent on the duct geometry, aircraft speed, attitude and proximity to upstream disturbances. As discussed in Section 1.3.3.3, boundary layer control is most effectively provided by vortices that

span the boundary layer whereas the attenuation of existing flow distortion covering large portions of the duct flow path requires stronger vortices that extend beyond the edge of the local boundary layer.

While several researchers have argued that solid *VG*'s can not be actively controlled (e.g. Johnston and Nishi, 1990; Zhang, 1995), Barrett and Farokhi (1993) showed that solid *VG*'s can be modified to provide active control of vortex strength. The authors devised a mechanism that used tiny wires to push up ramp type *VG*'s that were otherwise flush with the surface. More recently, Anderson et al. (1999) studied the application of *VG*'s based on Micro Electrical Mechanical Systems (*MEMS*) technology for intake flow control. In practice such mechanical methods could be problematic, however, due to inherent maintenance difficulties. Wallis (1952) demonstrated that streamwise vorticity may also be produced by introducing a jet of fluid pitched and yawed (also referred to as skew) relative to the local freestream. This so-called Vortex-Generator Jet (*VGJ*) is shown in Figure 1.3.4.1. Using this technique, vortex strength may be varied substantially by merely adjusting the jet mass flow.

It was argued by Barret and Farokhi (1996) that the *VGJ* system suffers from increased complexity, weight and power consumption. However, these disadvantages must be weighed against the possibility of solid *VG* vanes detaching during operation and damaging the engine in intake applications (Gibb and Anderson, 1995). Furthermore, as

pointed out by Pearcey et al. (1993), solid VG 's become "ineffective - when submerged in local regions of separation or extra thick shear layers."

Figure 1.3.4.1 The vortex-generator jet, reproduced from Pearcey, (1961)

Choice of solid VG 's versus VGJ 's must be based on application, such as control of BL on the aft-fuselage of aircraft, control of BL flows on lifting surfaces and flow manipulation in engine intakes. Ball (1984), Gibb and Anderson (1995) and Senseney et al. (1995) have demonstrated the ability of VGJ 's to control boundary layer separation as well as manipulate distorted flows in a diffusing S-duct. However, Senseney et al. (1995) noted that the mechanism by which the VGJ 's provided these controls was not well understood.

A jet exhausting into a crossflow is deflected by the freestream fluid such that it follows a curved path downstream. In addition, the interaction of the jet with the crossflow results in a change in its cross-section. Complex vortical structures result from a normal jet / crossflow interaction which are characterised by three mechanisms: (1) a rolled up

longitudinal vortex pair, (2) a highly turbulent wake immediately downstream of the jet exit and (3) a horseshoe vortex around the jet exit. The longitudinal vortices dominate over the other mechanisms, as shown in Figure 1.4.2.2. As will be seen later, pitching and skewing the jet produces a single dominant longitudinal vortex.

Figure 1.3.4.2 Structure of a jet / crossflow interaction, reproduced from Chiu et al. (1993)

1.3.4.1 Longitudinal Vortex Structure

One can begin to explain the interaction between a jet and a crossflow by considering a round jet and modelling it as a rigid cylinder. The well known potential flow solution for a cylinder in a crossflow yields the relationship between pressure coefficient and angular surface position as:

$$C_p = \frac{P - P_\infty}{\frac{1}{2}\rho U^2} = 1 - 4 \sin^2 \gamma \quad (1.3.4.1.1)$$

Figure 1.3.4.1.1 Pressure distribution on a cylinder in crossflow, reproduced from Panton (1996)

This relationship is shown graphically in Figure 1.3.4.1.1. At the upstream and downstream positions ($\gamma = 0^\circ$ and 180° , respectively) stagnation points exist which correspond to a pressure coefficient of unity. At the point of maximum distance from the centreline (90° or 270°) the pressure is minimum. When this pressure distribution is applied to a non-rigid cylinder, such as a round fluid jet, the cylinder would deform into an elliptical cross-section. Additionally, the crossflow applies a shear stress along the lateral edges of the jet causing it to deform into a kidney shape. As the jet progresses downstream, the crossflow shear causes the kidney shape to fold over itself and produce a vortex pair. The jet path can be described by two different loci. The presence of the vortex pair results in loci of maximum vorticity, which is identified as the vortex path. The jet centreline path, on the other hand, is identified as the locus of maximum velocity. These two loci are shown in Figure 1.3.4.2 of the previous section.

The formation of this vortex pair was studied by Rosenhead (1932) and Westwater (1935), and later by Chang (1942) who used the earlier results to treat jet deflection in a crossflow analytically. The success of such an analytical treatment would be of interest to intake designers since analytical modelling of *VGJ* effects in a *CFD* framework, such as the vortex-source approach used for *VG*'s (Section 1.3.3.2), would reduce the simulation time for the flow in an engine intake significantly. Chang analysed the deformation of a round jet using potential flow theory in an attempt to describe the formation of vortex pairs. This method was extended by Margason (1969) using an empirically determined jet path. Margason (1993) concluded that this treatment was deficient however, due to a lack of entrainment into the jet wake, and incorrect levels of induced pressures in the jet exit plane. To date, no analytical functions exist that provide sufficient representation of jet-in-crossflow physics.

1.3.4.2 Wake and Horseshoe Vortex Structures

The assumption of inviscid, irrotational flow over a cylinder in crossflow dictates a maximum pressure coefficient at the downstream stagnation position equal to the value at the upstream stagnation position. Figure 1.3.4.1.1 shows experimental results as well as this ideal distribution of pressure. In the viscous case, boundary layer separation occurs due to the adverse streamwise pressure gradient, except for the case of extremely low Reynolds numbers, thereby forming a wake behind the jet exit. In addition to the vortex pair described in the previous section, a third vortical structure evolves from the interaction between the jet issuing from the wall and the oncoming wall boundary layer.

As the oncoming boundary layer encounters the jet, the vorticity lines associated with the boundary layer fluid wrap around the jet and form a U-shaped or horseshoe vortex.

1.3.4.3 VGJ Streamwise Vorticity Field

The inviscid and viscous mechanisms of a normal jet-in-crossflow responsible for the formation of streamwise vortices have been explained. It was determined by Wallis (1960) that vortices formed by a normal jet-in-crossflow did not persist far enough downstream to provide sufficient flow control. This is due to the mutual interaction and induction between the two vortices of equal strength and opposite sense of rotation.

Wallis performed experiments on an aerofoil with circular jets issuing from the surface at a pitch angle of 45° and skew angle of 90° (i.e., in the spanwise direction). This Vortex-Generator Jet (*VGJ*) configuration resulted in one dominant vortex which persisted farther downstream than the normal jet and remained closer to the surface. The terminology for a *VGJ* installation is summarised in Figure 1.3.4.3.1. This terminology will be adopted for *VGJ* discussion in the remainder of the survey unless stated otherwise.

A pitched and skewed jet produces streamwise vortices through the same physical mechanisms as a normal jet. The difference is that typically one of the vortices is dominant. Pearcey (1961) suggested that enhanced entrainment on the side of the pitched and skewed jet that is angled away from the wall strengthens the vortex on this side, which then sweeps the weaker one around and into itself. This argument explains the vortex

sense of rotation observed by Compton and Johnston (1992) and Johnston and Nishi (1990) displayed in Figure 1.3.4.3.2.

Figure 1.3.4.3.1 Schematic of a vortex-generator jet, reproduced from Barberopoulos and Garry (1998)

Figure 1.3.4.3.2 Observed *VGJ* vortex sense of rotation (looking downstream) by: a) Compton and Johnston (1992), b) Johnston and Nishi (1990)

Reference	VR	ϕ	θ
Compton and Johnston (1992)	1	45°	90°
Johnston and Nishi (1990)	0.8 - 1.0	45°	-90°

Table 1.3.4.3.1 *VGJ* parameters

Barberopoulos and Garry (1998) explained that skewing the jet perturbs the symmetrical pressure distribution which strengthens one of the vortex focal points. Previous research has shown dependence of *VGJ* performance on the following parameters:

- jet-to-freestream velocity ratio (VR)
- skew and pitch angles (θ, ϕ)
- ratio of a representative thickness of crossflow boundary layer to jet diameter
- jet Reynolds number (Re_D)
- jet cross-section geometry

Skew and pitch angles determine location and strength of the vortex for a given velocity ratio. The ratio of boundary layer thickness-to-jet diameter for a given velocity ratio determines the extent of jet penetration into the crossflow. The jet cross-sectional geometry affects fluid mixing since, for instance, a slender rectangular jet oriented with its longer side perpendicular to the crossflow will tend to fold over and dissipate more readily than a circular jet.

Barberopoulos and Garry (1998) described the aerodynamics of a *VGJ* based on vector field topology. This analysis uses the theory of critical points introduced by Legendre (1956). Earlier discussions of this theory are given by Hunt et al. (1978), Tobak and Peake (1981) and Perry and Chong (1987). At certain critical points in the flow field, the streamline slope is indeterminate. Perry and Chong state that asymptotically exact

solutions of the Navier-Stokes and continuity equations can be derived close to these points, which yield local flow patterns.

Figure 1.3.4.3.3 shows a composite streamline pattern of a normal jet / crossflow interaction in three dimensions, given by Kelso et al. (1996). In the x - z plane, upstream of the jet (where the horseshoe vortex system is formed) there exists a series of 3-D attachment lines or positive bifurcation lines (*PBL*) and 3-D lines of separation or negative bifurcation lines (*NBL*). These bifurcation lines converge towards the rear of the jet and spiral into a focus (critical point A). An additional *NBL* is shown to leave the jet pipe and spiral into a second focus upstream of the first focus (critical point B). Fluid is lifted off the wall in tornado-like vortices at these two foci.

Figure 1.3.4.3.3 Composite 3-D jet / crossflow interaction streamline pattern, reproduced from Kelso et al. (1996)

The three-dimensional flow field topology of a jet pitched at 45° without skew, deduced by Barberopoulos and Garry (1998), is shown in Figure 1.3.4.3.4. Nodes of attachment

(Na) and saddles of separation (Ss) as well as a focus node (F) are identified. Unlike the normal jet, only one focus exists on each side of the jet. No comment of this difference is made by the authors. It appears that the two foci tend to converge as the jet is pitched.

Figure 1.3.4.3.4 3-D flow field of a jet pitched at 45° and zero skew angle, reproduced from Barberopoulos and Garry (1998)

Barberopoulos and Garry state that the position and size of the focal points are crucial in understanding vorticity production. They explain that these points characterise the separation around the jet where the streamwise vortices originate. Figures 1.3.4.3.5 (a) and (b) display the surface flow topology of a jet pitched at 45° , with skew angles of 0° and 60° respectively. Although the values of VR are different for these two cases, skew angle was found to be the dominating parameter. In both diagrams the two attachment nodes ($Na1$ and $Na2$) are necessary to isolate saddle point $Ss2$ from $Ss1$ and $Ss3$.

Figure 1.3.4.3.5 Surface flow topology of a jet pitched at 45°
a) 0° Skew, $VR=1.2$, b) 60° Skew, $VR=1.4$,
reproduced from Barberopoulos and Garry (1998)

1.3.4.4 Effect of Jet / Cross-stream Velocity Ratio on Performance

It was found by Barberopoulos and Garry (1998) that the streamwise extent of the reverse flow region on the leeward side of the jet was dependent on the jet / freestream velocity ratio (VR). Referring back to Figure 1.3.4.3.5, this reverse flow region is located between separation saddle point $Ss6$ and attachment node $Na4$. They found that the lateral extent of the separated region increased with an increase in VR . This, in turn, displaced the vortex foci $F1$ and $F2$ which altered the dominant vortex path as shown in Figure 1.3.4.4.1.

A numerical study conducted by Zhang (1995) showed the dependence of streamwise vortex circulation on VR . Figure 1.3.4.4.2 displays these results at several streamwise locations. As expected, an increase in circulation with increasing VR is observed. It is

also noted that the rate of circulation increase with VR diminishes as VR is increased. The circulation level is, however, maintained over a somewhat longer streamwise distance at higher VR .

Figure 1.3.4.4.1 Variation of dominant VGJ vortex path with VR for 45° pitch and 45° skew, reproduced from Barberopoulos and Garry (1998)

Figure 1.3.4.4.2 Variation in VGJ vortex normalised circulation with VR , 45° pitch and 45° skew, reproduced from Zhang (1995)

The effect of VR on skin friction distribution was explored by Zhang and Collins (1993).

The results for three different values of VR are given in Figure 1.3.4.4.3. A region of high skin friction due to the streamwise vortex exists on one side of the jet. At higher jet velocities ($VR=2.0, 3.0$) a second region of high skin friction exists due to a weaker secondary vortex.

Figure 1.3.4.4.3 Computed variation in VGJ skin friction coefficient distribution with VR for 45° Pitch and 45° Skew, a) $VR=0.7$, b) $VR=2.0$, c) $VR=3.0$, reproduced from Zhang and Collins (1993)

Zhang and Collins (1993) stated that this is due to initial vorticity production at the jet exit. An area of low skin friction exists between these two regions due to boundary layer fluid not swept away by the two vortices. At the lower VR , the vorticity field is weaker and only one vortex is apparent. The same researchers observed the vertical migration of

jet centrelines as a function of VR , as displayed in Figure 1.3.4.4.4. Evidently, while an increase in VR decreases spanwise movement, vertical penetration is increased. Similar vortex migration patterns were observed by Johnston and Nishi (1990) for multiple jets. In this case the percentage of jet mass flow near the wall with respect to total jet mass flow just downstream of the jet entrance was plotted. These results can be compared to the previous figures displaying vortex trajectories since a low percentage of jet fluid near the wall indicates jet-fluid migration away from the wall and hence an ineffective streamwise vortex in terms of energising the local boundary layer. Several jet orientations were explored as shown in Figure 1.3.4.4.5. The results are shown for six different velocity ratios in Figure 1.3.4.4.6.

Figure 1.3.4.4.4 Vertical migration of vortex centreline with VR for 45° pitch and 45° skew, a) $VR=0.7$, b) $VR=2.0$, c) $VR=3.0$, reproduced from Zhang and Collins (1993)

Figure 1.3.4.4.5 Five VGJ configurations, reproduced from Johnston and Nishi (1990)

Figure 1.3.4.4.6 Percent jet fluid mass flow variation for five *VGJ* configurations, reproduced from Johnston and Nishi (1990)

1.3.4.5 Effect of Jet Skew Angle Performance

The change in skew angle was found by Zhang and Collins (1993) to alter the jet path by introducing a lateral velocity component. In addition, skew angle was observed to affect the resultant vorticity appearing in the dominant streamwise vortex. The primary effect of skew is the asymmetric bias it introduces to the local pressure distribution, which results in the development of a single dominant vortex. Plots of skin friction coefficient (C_f) distribution observed in this study are shown in Figure 1.3.4.5.1. An increase in jet skew angle results in an increase in the area of high stress at the wall. A rapid increase in maximum C_f occurs between 0° and 45° skew. Subsequent increase in skew angle causes only slight changes. The conclusion drawn by Compton and Johnston (1992) was that, for

boundary layer separation control, the optimum value of skew angle (resulting in minimum migration of vorticity from the wall) lay between 45° and 90° . Selby et al. (1992) concluded that the dominant streamwise vortex was strongest at skew angles between 60° to 90° . In this case they studied the pressure recovery on a rearward facing ramp with varying skew angle at a given pitch angle. They found that pressure recovery was maximised at 60° skew and 15° pitch as well as at 90° skew and 45° pitch. The optimum skew angle was therefore observed to be dependent on the pitch angle. The experimental findings of Pearcey et al. (1993) and computations of Zhang and Collins (1993) showed an optimum skew angle of approximately 60° at 15° pitch. Henry and Pearcey (1994) also found this angle combination to be optimum based on observations of the cross-stream variation of skin friction with skew angle. Maximum separation-control performance was observed by Barberopoulos and Garry (1998) at 60° skew angle and 45° pitch.

It was noted, however, that this angle would change as a function of the other jet parameters, and that a detailed study would be required to determine such variations. This statement reflects the findings of Selby et al. (1992) since in this case the optimum skew angle was found to be dependent on pitch angle.

1.3.4.6 Effect of Jet Pitch Angle on Performance

Most studies of *VGJ* performance have considered a fixed value for the jet pitch. As a result, less data is available on the effects of this parameter. Maximum pressure recovery on a rearward facing ramp was observed by Selby et al. (1992) at pitch angles between 15° and 25° for a skew angle of 90° . From the discussion in the previous section it is

Figure 1.3.4.6.1 Cross-stream skin friction variation with pitch angle for 60° skew and $VR=1.0$, reproduced from Henry and Pearcey (1994)

clear, however, that this optimum range is dependent on the chosen skew angle. Henry and Pearcey (1994) showed that a decrease in pitch angle resulted in an increase in the streamwise extent of the *VGJ* effectiveness. Figure 1.3.4.6.1 shows an increase in skin friction coefficient over a wider region as pitch angle is decreased. Pearcey et al. (1993) observed the same trend, noting a higher performance for 30° pitch than for 45°.

1.3.5 Turbulence Modelling for High Performance S-Duct Intakes

The physics of the flow in aircraft engine intakes utilising Vortex-Generator Jets (*VGJ*'s) is influenced by three main factors: adverse pressure gradient, streamline curvature, and jet/ crossflow mixing. Each of these contribute to highly three dimensional phenomena that present substantial challenges to current turbulence modelling techniques. This survey is divided into three sections in attempt to isolate problems associated with each phenomenon. Turbulence modelling of: 1) straight diffusers, 2) curved ducts, 3) *VGJ*'s, all of which are discussed with respect to physical issues and demonstrated model performance in the literature. A limited number of results from combinations of these categories are also shown. Zero-, one- and two- equation model results are presented and discussed. In addition, some results from other closure methods are given. The review is not intended to recount the history of turbulence modelling in general or present derivations of the model equations, since numerous texts, such as those of White (1991) and Wilcox (1994, 1998), contain such information. However, for the sake of completeness, brief outlines of each model are given to support arguments on

performance. Formulations of the models chosen for the present study are documented in Chapter 3.

1.3.5.1 Turbulence Modelling in Straight Diffusers

Although conical or planar diffusers are generally regarded as simple devices, the complete description of the turbulent flow in these geometries has posed considerable challenge.

One conclusion from the 1980-81 AFOSR-HTTM Stanford Conference on Complex Turbulent Flows (Kline et al., 1982) was that *eddy-viscosity* type turbulence closures predicted the influence of pressure gradient on shear layers rather poorly. These type of models follow the approximation of Boussinesq (1877) that the Reynolds stresses are related to the rate of mean strain through a linear constitutive relation. This treatment of Reynolds stress is analogous to the dependence of viscous stresses on the mean rate of strain. Therefore, the constant of proportionality in the Reynolds stress / rate of strain relation is commonly referred to as the eddy- viscosity (μ_t):

$$\tau_{ij} \equiv -\overline{\rho u_i u_j} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \left(\mu_t \frac{\partial U_l}{\partial x_l} + \rho k \right) \quad (1.3.5.1.1)$$

In this relation U and u denote mean and fluctuating velocities respectively, and k is the turbulent kinetic energy defined in as:

$$k = \frac{\overline{u_i u_i}}{2} \quad (1.3.5.1.2)$$

Closures that follow this approximation solve algebraic or partial differential equations, or a combination of both, to determine length and time scales of the turbulent eddies which are in turn used to calculate the eddy-viscosity, μ_t .

1.3.5.1.1 Two-Equation Turbulence Models

Two-equation models employ partial differential equations to determine the length and time scales of turbulence. The two most popular such models currently used in most commercial *CFD* codes are the k - ϵ and k - ω closures. Rodi and Scheuerer (1986) state that the problem of accounting for pressure gradient recognised in the 1981 Stanford Conference is apparent in any model which solves a partial differential equation for the length-scale of turbulence. This problem is explained after a brief description of the models that follows.

The k - ϵ model solves partial differential equations for turbulence kinetic energy, k , and rate of dissipation of turbulent kinetic energy, ϵ . The standard k - ϵ model (*SKE*) was introduced by Jones and Launder (1972). There are five closure coefficients present in this formulation that are introduced to close the system which would otherwise contain higher order correlations that require additional equations to be solved. A fundamental problem with this model is the ϵ equation, which has been derived based on dimensional

analysis and physical reasoning. Taking the appropriate moments of the Navier-Stokes equations to derive a relation for ε would produce higher order correlations that are extremely difficult to measure experimentally. The constants in the k and ε equations are chosen based on asymptotic analysis to match the data for equilibrium turbulent boundary layers. Inherent problems occurring in diffuser flow computations using such turbulence models are due in part to these closure coefficients.

A problem with the standard k - ε model, as highlighted by Rodi and Scheuerer (1986), is that it tends to over-predict skin friction in adverse-pressure-gradient flows. They state that the length-scale determined via the partial differential equation for ε rises steeper near the wall than in the case of zero-pressure gradient, whereas experiments suggest that the length-scale gradient is independent of pressure gradient over a wide range. Figure 1.3.5.1.1.1 displays the results of standard k - ε closure in an adverse-pressure-gradient flow and compares the results with a low-Re k - ε model (Rodi et al., 1982), a one-equation model (Norris and Reynolds, 1975) and the experimental results of Andersen et al. (1972). Here the pressure gradient is dictated by the relation of boundary-layer edge velocity (U_e) to streamwise location (x) given by $U_e \propto x^m$ where m is a constant given in the figure. The definitions of the other turbulence models are given in later sections but it is noted here that the one-equation model replaces the partial differential equation for ε with an algebraic one. From the figure it is observed that skin friction in the adverse-pressure-gradient flow is over-predicted by both versions of the k - ε model.

A problem that must be addressed for any turbulence model is the specifications of boundary conditions at the wall. Some models impose boundary conditions on turbulence properties at the wall surface and require integration all the way to the wall which results in a significant increase in the number of grid nodes, and hence substantial computational cost. The standard k - ϵ model reduces grid resolution requirements by using *wall functions* to prescribe the near-wall turbulent profile. Although significant computational savings

Figure 1.3.5.1.1.1 Comparison of skin friction predictions in zero ($m=0.0$) and adverse ($m=-0.20$) pressure gradient flow,
 $\circ$ experiment for $m=0.0$, $\bullet$ experiment for $m=-0.20$,
 $\times$ standard k - ϵ ($m=-0.20$ only), — low- Re k - ϵ ,
 -- One-equation, reproduced from Rodi and Scheuerer (1986)

can be realised with the use of such boundary conditions, there are inherent problems with this technique. The prediction of the near wall flow tends to be sensitive to the location of the first node off the wall where the conditions on k and ϵ are imposed, which requires additional effort for grid generation particularly in three-dimensional computational domains. As shown in Figure 1.3.5.1.1.2 the range of applicability of the wall function approach is dictated by the strength of the adverse pressure gradient. Furthermore, the wall function approach does not apply at all in separated flows.

Researchers have made attempts to remove the constraint of using wall functions by integrating the k and ε equations to the wall. Models of this type are commonly referred to as low-Re k - ε models. The standard formula for ε does not account for viscous damping in the sublayer and must be modified for this. The earliest proposal for viscous damping correction was made by van Driest (1956), and since then there have been many others. A popular set of functions are those of Chien (1982).

Figure 1.3.5.1.1.2 Turbulent velocity profile in adverse pressure gradient, reproduced from White (1991)

The value of ε at the wall is dependent on the second derivative of turbulence kinetic energy in this region. Calculation of the second derivative of k in a discretised framework is a noisy process and would require unreasonably high grid resolution. To circumvent

this problem, in Chien's model the equation for ε is reformulated in terms of an alternative dissipation rate, $\tilde{\varepsilon}$, which approaches zero at the wall. The drawback of such low- Re models is the number of empirical damping functions that are required which cannot be assigned "universal" values and are responsible for numerical stiffness. Furthermore, the order of magnitudes of turbulent convection and diffusion in the inner region of a turbulent boundary layer are much lower than those of production and dissipation, which results in added numerical stiffness in the solution of these models.

Problems apparent in the k - ε closure, such as poor prediction capability in adverse-pressure-gradient flows and the requirement of complicated damping functions for integration through the sublayer, have motivated researchers to investigate alternative models. Wilcox (1994) proposed one version of a k - ω closure as a model that does not suffer from these shortcomings. Nonetheless, low Re versions of the k - ω model have been proposed to improve prediction accuracy of turbulence kinetic energy close to the surface. The k - ω closure solves an equation for turbulence kinetic energy, and replaces the partial differential equation for dissipation rate of turbulence kinetic energy, ε , with one for "specific dissipation rate" $\omega = \varepsilon/k$. Many researchers have proposed models based on k and ω , the first of which was Kolmogorov (1942).

Coles (see Kline et al., 1982) described the turbulent boundary layer as a "wake-like structure constrained by a wall." Using perturbation methods, Wilcox (1988) showed that, in the case of equilibrium turbulent boundary layers, the k - ω model is consistent with

the defect layer and that the $k-\epsilon$ model is not. He also stated that the $k-\omega$ model was within “engineering accuracy” for sixteen non-equilibrium flows tested. These consisted of favourable, as well as varying degrees of adverse pressure gradient conditions. The results for the strong adverse pressure gradient flows are reproduced in Figure 1.3.5.1.1.3 and Table 1.3.5.1.1.1. These are, flow 0141 from the 1980-81 Stanford conference (Kline et al., 1982) and flows 1200, 4400 and 5300 from the 1968 Stanford conference (Coles and Hirst, 1969). The models employed in this study were all of the low- Re type except for the Wilcox high- Re $k-\omega$ WH closure (Wilcox, 1988).

Figure 1.3.5.1.1.3 Computed and measured C_f in strong adverse pressure gradient, o Expt., JL Jones & Launder low Re $k-\epsilon$, LS Launder & Sharma low Re $k-\epsilon$, LB Lam & Bremhorst low Re $k-\epsilon$, CH Chien low Re $k-\epsilon$, YS Yang & Shih low Re $k-\epsilon$, FL Fan, Lakshminarayana and Barnett low Re $k-\epsilon$, WH Wilcox $k-\omega$, WL Wilcox low Re $k-\omega$, reproduced from Wilcox (1994)

The errors presented in the table are averaged absolute values of the difference between experimental and computed skin friction. It appears that at least for these test cases the k - ω models produce substantially better results than the others.

Model	Favorable	Mild Adverse	Moderate Adverse	Strong Adverse	Overall Average
WL	2	4	4	6	4
WH	1	4	6	12	6
LS	7	27	40	71	36
CH	7	32	50	55	36
YS	3	34	46	75	40
FL	6	29	44	84	41
LB	3	38	50	85	44
JL	9	51	66	105	58

Table 1.3.5.1.1.1 Percentage error in prediction of C_f , reproduced from Wilcox (1994), (refer to caption of Figure 1.3.5.1.1.3 for detailed identification)

More recent studies by Dudek (1996) and Dudek et al. (1996) have compared the Chien low- Re k - ϵ , the Wilcox low- Re k - ω , as well as the Baldwin-Lomax (Baldwin and Lomax, 1978) zero- equation model and the Baldwin-Barth (Baldwin and Barth, 1991) one- equation model which will be described in the next sections, to the conical diffuser data of Fraser (Fraser Flow A, Coles and Hirst, 1969). The results of this study are displayed in Figures 1.3.5.1.1.4 and 1.3.5.1.1.5.

Figure 1.3.5.1.1.4 Conical diffuser velocity profile predictions, reproduced from Dudek (1996)

Figure 1.3.5.1.1.5 Conical diffuser predictions

a) displacement thickness, b) momentum thickness, c) shape factor, d) skin friction coefficient, e) static pressure coefficient, reproduced from Dudek (1996)

A notable discrepancy between the $k-\epsilon$ and $k-\omega$ predictions is evident in the plot of skin friction coefficient (Figure 1.3.5.1.1.5 d). The $k-\epsilon$ model over predicts skin friction substantially whereas the $k-\omega$ model matches the skin friction data quite well, thereby supporting the claims of Wilcox (1994). The sensitivity of the velocity profile predictions

to the skin friction predictions is evident in Figure 1.3.5.1.1.4. Even though the $k-\omega$ model only slightly over predicts the skin friction coefficient near the end of the diffuser (station 11) this is accompanied by significant underestimation of the near-wall momentum deficit. Surprisingly, the $k-\epsilon$ profile prediction at station 11 is comparable to the $k-\omega$ model. Another interesting observation is that the prediction of the Baldwin Lomax zero-equation model is not much worse than that of $k-\omega$.

Menter (1992a, b) identified important issues regarding the implementation of the $k-\omega$ model in simulating such flows as the adverse-pressure-gradient experiment of Driver and Johnston (1990). The boundary layer in this case develops along the surface of a circular cylinder under the influence of an adverse pressure gradient imposed by diverging wind-tunnel walls with suction. Results of this test case are displayed in Figure 1.3.5.1.1.6. Menter identified notable sensitivity of the simulated flow to the specified value of k at the inflow boundary. Furthermore, he confirmed the positive feature of integration through the sublayer without damping functions, namely yielding a solution that is much less stiff numerically.

Contrary to the behaviour of the $k-\epsilon$ model, the $k-\omega$ model appears to encounter difficulties at the shear layer / freestream interfaces. Menter (1992a) observed a matched exponential decay of both k and ω with the square of the distance from the wall. Thus, the eddy-viscosity calculated using the standard $k-\omega$ definition remains constant when in reality it should approach zero at the viscous / inviscid interface. This effect tends to be

less severe in boundary-layer flows than in free shear layers since larger values of ω are calculated at the edge due to wall influence. It was also observed by Menter that adverse-pressure-gradient flows can have production / dissipation ratios notably larger than 1.0. On this basis he concluded that even if the correct values of turbulence kinetic energy and dissipation rate are computed, the standard equations would yield unrealistically high Reynolds stresses. A restriction on eddy-viscosity was therefore used in Menter's version of the k - ω model. Details of the equations are postponed until Chapter 3.

Figure 1.3.5.1.1.6 Separating adverse-pressure-gradient flow (C.S0) of Driver and Johnston, *BL*=Baldwin-Lomax, *JK*=Johnson-King, *BB*=Baldwin-Barth a) velocity profiles, b) Reynolds stresses, c) skin friction coefficient, C_f , d) static pressure coefficient, reproduced from Menter (1992b)

1.3.5.1.2 Zero Equation Models

The calculation of turbulence quantities with zero-equation models is much simpler than with one- or two-equation models. Instead of calculating turbulence length and time scales with partial differential equations, they are determined from algebraic expressions.

The value of eddy-viscosity is based on a mixing length such as the one proposed by Prandtl (1925). It was discussed in Section 1.3.5.1.1 that closure coefficients employed in two-equation models are based on the assumption of turbulence equilibrium. Although this limits the prediction of flows where non-equilibrium effects are significant, convection and diffusion effects are still captured by these models. This is not the case in algebraic models which theoretically cannot account for any *history* effects. Such models are thus severely restricted at a fundamental level in their ability to accurately predict flows of engineering interest.

1.3.5.1.3 One Equation Models

Classical one-equation models solve one partial differential equation, for instance for turbulence kinetic energy representing the velocity scale of turbulence, but employ an algebraic equation to calculate the remaining turbulence scale. Such models have been shown to often provide higher prediction accuracy than low-Re k - ϵ models. This result is due to poor performance of *PDE*-based dissipation rate calculations in certain flows, as discussed earlier.

More recently, one-equation models which use eddy-viscosity, rather than a more fundamental quantity such as k , as the primary variable have been regaining popularity. Figures 1.3.5.1.1.4 and 1.3.5.1.1.5 displayed the conical diffuser predictions of Dudek (1996) and Figure 1.3.5.1.1.6 displayed the separating adverse pressure gradient flow of Menter (1992b). Both tests included the one-equation model of Baldwin and Barth

(1991) which directly solves for the eddy-viscosity. The prediction capability of this model is seen to be generally worse than the two-equation models tested. It significantly over predicts displacement thickness, shape factor and hence the effect of adverse streamwise pressure gradient. Skin friction is significantly under predicted as well. Menter observed the highest performance in the C.S0 case by the Johnson-King (1985) “half- equation” model, *JK*. This is a half-equation model which is defined as solving one *ordinary* differential equation. Such models attempt to improve the prediction of non-equilibrium flows over algebraic models by solving an ordinary differential equation for the maximum value of Reynolds stress. A turbulence length scale must still be specified algebraically.

Numerical problems at the edge of shear layers are not restricted to the $k-\omega$ model described earlier. Analyses of the *BB* model by Menter (1994) and Spalart and Allmaras (1994) showed that the underlying equations are ill-conditioned in this region. As a result, prediction is sensitive to grid resolution near such interfaces. Spalart and Allmaras (1994) have developed a one-equation model (*SA*) that does not suffer from this shortcoming. In the *SA* formulation, a partial differential equation is solved for the transport of, $\bar{\nu}_t$, a modified form of eddy-viscosity. Three damping functions and eight closure coefficients are employed in this model.

Figure 1.3.5.1.3.1 displays results of C.S0 tests performed by Menter (1996a). The models used in this study were the Spalart-Allmaras one-equation model (*SA*), Menter's

Baseline zonal $k-\omega$ / $k-\epsilon$ model (1994), a zonal $k-\omega$ / $k-\epsilon$ version of the renormalisation group (*RNG*) model (e.g., Yakhot and Orszag, 1986) and Menter's (1994) Shear-Stress-Transport (*SST*) model. The highest accuracy results from the *SA* and *SST* models. The high performance of the *SA* one-equation model compared to the two-equation models further supports the discussion given above regarding modelling of the dissipation rate through a *PDE*. The use of one-equation models cannot be expected to capture all the physics.

Figures 1.3.5.1.3.1 Separating adverse pressure gradient flow C.S0 of Driver and Johnston, a) velocity, b) skin friction, c) static pressure reproduced from Menter (1996a)

1.3.5.1.4 Zonal Two-Equation Models

For the C.S0 test case presented in Section 1.3.5.1.3, Menter (1996a) observed the highest accuracy from the *SST* model. This model falls in the same category as the zonal versions of k - ϵ and *RNG* closures tested by Menter. These formulations take advantage of the relative strengths of various models at different distances from a wall surface. Each of the three models solve the k - ω model near the wall and k - ϵ elsewhere. The zonal k - ϵ and *RNG* models use k - ω in the sublayer only. The *SST* model employs k - ω from the wall all the way through the logarithmic layer, and blending with the k - ϵ formulation takes place in the wake region of the boundary layer. Formulation of the *SST* model equations is presented in Chapter 3.

1.3.5.2 Curved Ducts

In Section 1.3.2.5 it was explained how cross-stream pressure gradients are imposed on the fluid in curved ducts which results in secondary flow. Bradshaw (1987) stated that the effect of the induced streamwise vorticity is to rotate the turbulent eddies about a streamwise axis. He suggested that models based on Reynolds stress transport will have trouble reproducing the changes in shear stress magnitude in a boundary layer with crossflow, i.e., a three dimensional boundary layer.

In addition, the effect of streamline curvature on turbulence depends on whether the curvature is concave or convex. Researchers such as Bradshaw (1973), So and Mellor

(1973) and Barlow and Johnston (1988) have observed skin friction to increase on a concave surface, when compared to flat surfaces, due to the amplification of turbulence by the curvature. Conversely, it was shown that convex curvature tends to damp turbulence resulting in a decrease of skin friction coefficient. Muck et al. (1985) and Hoffman et al. (1985) suggested that these two effects have qualitatively different character. Concave curvature results in a reorganisation of the eddy structures, whereas convex curvature simply attenuates pre-existing eddies. Barlow and Johnston (1988) found that concave boundary layers displayed a range of behaviour from longitudinal Taylor-Görtler vortices to energetic large eddies. Luo and Lakshminarayana (1996) explain that the phenomenological modelling of turbulence for concave curvature has to be different than for convex curvature for these reasons.

These curvature effects tend to promote departure from isotropy which is an implicit assumption in standard eddy-viscosity models. Predictions of three turbulent curved-duct flows by Luo and Lakshminarayana (1996) are displayed in Figures 1.3.5.2.1 - 1.3.5.2.5. Figure 1.3.5.2.1 shows the prediction of 2-D flow in the 180° duct of Sandborn (1989) using the standard k - ϵ model and three different Reynolds Stress Models (*RSM*). Reynolds stress, or *second moment*, closures model the Reynolds stresses directly rather than assuming the eddy-viscosity approximation. Improved prediction of anisotropic turbulence is therefore made possible. The disadvantage of these models is that a partial differential equation must be solved for each Reynolds stress (i.e., six equations in 3-D flow) which results in substantial computational expense compared to eddy-viscosity

based models. In addition, closure with this type of model still requires the solution of a partial differential equation for turbulence dissipation rate, ϵ . Therefore, all the aforementioned problems associated with the ϵ equation are retained. Figures 1.3.5.2.2 - 1.3.5.2.4 display prediction of the 90° rectangular duct flow of Kim and Patel (1993). The turbulent boundary layer prediction on the concave 90° duct experiment performed by Johnson and Johnston (1989) is shown in Figure 1.3.5.2.5. The standard *RSM* of Gibson and Launder (1978) and *RSM*'s employing streamline curvature corrections as proposed by of Launder et al. (*LPS*) (1977) and Shih et al. (1995) are shown in these figures. All of the *RSM* models are observed to offer improved prediction accuracy compared to the standard k - ϵ model. However, due to their substantial computational overhead, these models do not provide a trade-off between accuracy and computation times that is acceptable for iterative design-cycle analyses. Thus, the following discussion concentrates on the trends observed for the k - ϵ model.

Figure 1.3.5.2.1 Prediction of 2-D flow in a 180° duct, $\theta=30^\circ$,
a- c) Reynolds stresses, d) streamwise velocity (H =duct height),
reproduced from Luo and Lakshminarayana (1996)

Figure 1.3.5.2.2 Prediction of 90° rectangular duct, $\theta=45^\circ$,
a-c) Reynolds stresses, d) streamwise velocity (H =duct height),
reproduced from Luo and Lakshminarayana (1996).

Figure 1.3.5.2.3 Prediction of 90° rectangular duct, 1/2 duct height (H) downstream
from bend, a-c) Reynolds stresses, d) streamwise velocity,
reproduced from Luo and Lakshminarayana (1996)

Figure 1.3.5.2.4 Prediction of 90° rectangular duct, a) inner (convex) wall skin friction
coefficient, b) outer (concave) wall skin friction coefficient (H =duct
height, s =streamwise coordinate), reproduced from Luo and
Lakshminarayana (1996)

Figure 1.3.5.2.5 Prediction of concave boundary layer development, $\theta=30^\circ$
a-c) Reynolds stresses, d) skin friction coefficient (H =duct height, R =surface curve radius, s =streamwise coordinate), reproduced from Luo and Lakshminarayana (1996)

In Figure 1.3.5.2.1 ($\theta=30^\circ$), near the inner wall of the 180° duct, turbulence has been damped by the convex curvature and flow acceleration, which results in very low values of measured turbulent shear stress. Large amplification of turbulence is observed near the concave wall. The standard k - ϵ model does not capture these effects and results in a much more uniform profile. The predicted velocity profiles, however, are seen to match the experimental data quite well. Luo and Lakshminarayana explain that this is due to the dominance of pressure forces over turbulent stresses at this station in the duct .

In Figure 1.3.5.2.2, displaying the results at 45° , the ratio of boundary layer thickness to radius of curvature is much lower than the previous station ($\delta/R_{\text{inner}}=0.05$, $\delta/R_{\text{outer}}=0.04$ versus $\delta/R_{\text{inner}}=1.0$, $\delta/R_{\text{outer}}=0.3$). The Reynolds shear stresses are once again significantly

under predicted near the concave wall. Results at the station one-half duct height downstream of the bend shown in Figure 1.3.5.2.3 are similar, although the discrepancy downstream of the concave surface is worse. This appears to have little effect on velocity profiles at either station however, and is again likely due to pressure gradients dominating the Reynolds stress effects. Finally, Figure 1.3.5.2.4 displays the resulting prediction of skin friction along the length of the duct where the $k-\epsilon$ model shows somewhat lower overall accuracy than the RSM . It is also noteworthy that the standard RSM suffers underprediction of Reynolds stresses near the concave wall just as the standard $k-\epsilon$ model, but to a lesser extent, and that the RSM 's with curvature correction yield improved results.

In order to isolate the effects of pressure gradient from streamline curvature, Johnson and Johnston (1989) contoured the inner wall of their 90° duct to minimise the streamwise pressure gradient on the outer wall where measurements were taken. The ratio of inlet boundary layer thickness to radius of curvature was $\delta/R_{outer}=0.055$. As seen in Figure 1.3.5.2.5, the predictions of the standard $k-\epsilon$ model are once again inferior to those of the RSM 's. It is evident from the comparison of the standard $k-\epsilon$ model, standard RSM and the RSM 's with streamline curvature correction, that prediction of flows of this type with high accuracy depends on two factors. The anisotropy of Reynolds stresses must be correctly represented and the PDE for turbulence dissipation rate must be sensitised to the effects of streamline curvature.

Rumsey et al. (1999) investigated the performance of three different turbulence closures for the 180° U-duct flow of Monson and Seegmiller (1992). The models tested were the Spalart-Allmaras (SA) one-equation model, the Menter Shear-Stress-Transport (SST) two-equation model, and an Explicit Algebraic Stress Model (EASM) developed by Gatski and Speziale (1993). The Algebraic Stress Model (ASM) is derived from the full Reynolds stress equations by reducing them to nonlinear algebraic equations through an approximation of the convective and turbulence transport (Rodi, 1976). The constitutive relations can become very complex and contain many closure coefficients. Wilcox (1994) noted that although the ASM provides significant improvement in prediction of flows with notable streamline curvature, it fails to properly account for sudden changes in mean strain rate. Results of these tests are shown in Figures 1.3.5.2.6 to 1.3.5.2.9.

Figure 1.3.5.2.6 Reynolds stress prediction of 2-D flow in a 180° U-duct, a) $x/H=-2$, b) 0° in bend, reproduced from Rumsey et al. (1999)

Figure 1.3.5.2.6 displays Reynolds shear stress profiles upstream of the bend and at the bend entrance. Interestingly, all models already begin to show significant differences from the experimental data at the bend entrance. This is indicative of significant elliptic influence in this region. Figure 1.3.5.2.7 shows predicted velocity, Reynolds stress and turbulence kinetic energy profiles at 90° into the bend. The previously-noted problems in calculating turbulence quantities in the presence of concave or convex curvature are once again observed here. None of the models predict near-complete absence of Reynolds

Figure 1.3.5.2.7 Prediction of 2-D flow in a 180° U-duct, 90° into the bend
a) streamwise velocity, b) Reynolds stress, c) turbulence kinetic energy, reproduced from Rumsey et al. (1999)

shear stress in the vicinity of the convex wall evident in the experimental data. In the concave wall region all models underpredict both the Reynolds stresses and the turbulence kinetic energy, although the discrepancy is smaller for the *EASM*. As before, although prediction of the turbulence properties is quite poor, the predicted time averaged velocity profile agrees well with the measurements. Figure 1.3.5.2.9a displays the velocity profile at one-half of the duct height downstream from the bend entrance. Rumsey et al. (1999) state that all models predict separation too far downstream, although the separation length is much better predicted. None of the models capture the correct maximum magnitude of reverse flow. It is stated by Rumsey et al. that each model exhibits recovery from separation that is too slow and that this is a common trait among most turbulence models currently in use. This trend is reflected to some extent in the static pressure and skin friction coefficient distributions through the bend on the concave and convex surfaces as shown in Figures 1.3.5.2.9 (b),(c) and 1.3.5.2.8.

Figure 1.3.5.2.8 Prediction of 2-D flow in a 180° U-duct,
a) inner (convex) C_f , b) outer (concave) C_f ,
reproduced from Rumsey et al. (1999)

Figure 1.3.5.2.9 Prediction of 2-D flow in a 180° U-duct, a) streamwise velocity at streamwise coordinate $s/H=0.5$ (separated region), b) inner (convex) surface pressure distribution, c) outer (concave) surface pressure distribution, reproduced from Rumsey et al. (1999).

1.3.5.3 Vortex-Generator Jets

Accurate prediction of the effects of turbulence in the flow of a Vortex-Generator Jet

(VGJ) dictates the treatment of several complicated three-dimensional phenomena. These include entrainment of the jet into the crossflow and diffusion of the subsequent longitudinal vorticity. The same problems that plague the accurate prediction of turbulence in diffusers and curved ducts have been found to affect the prediction of jets in

crossflow. Several studies draw the common conclusions that inherent isotropy of eddy-viscosity and failure to properly model the effect of curvature on the turbulence field limit the accuracy of several models in jet-in-crossflow applications. The discussions in this section will therefore serve to supplement the previous discussion on curved ducts.

Hassan et al. (1998) have employed Menter's zonal *Baseline* $k-\epsilon$ / $k-\omega$ and *SST* models, as well as the standard $k-\epsilon$ model in an investigation of a row of jets injected from a flat plate at compound angles. The configuration is shown in Figure 1.3.5.3.1.1. These results are compared with the data of Findlay (1998) in Figures 1.3.5.3.1.2 and 1.3.5.3.1.3. Figure 1.3.5.3.1.2 displays predictions of turbulence kinetic energy profiles in the near-hole region. Agreement of the three models with experiment vary depending on velocity ratio and downstream distance. Prediction of the standard $k-\epsilon$ model is generally poor while the *Baseline* and *SST* models offer significant improvement. The *Baseline* $k-\epsilon$ appears to give the best prediction of turbulence kinetic energy and jet-fluid concentration gradient C . The peak in turbulence kinetic energy observed in the data near the wall at $x/D=1.0$ is not captured by any model. They explain that either inadequate modelling of turbulence in the jet exit region or inadequate representation of the plenum and jet entrance are likely causes. A computational study by Kim and Benson (1992) determined that accurate prediction of the jet / crossflow interaction would necessitate inclusion of at least one jet diameter of the jet pipe, in the case of jet flow from a straight pipe, or otherwise inclusion of the jet plenum. A typical grid distribution in the near-hole region is displayed in Figure 1.3.5.3.1.4. Although there may be less uncertainty when the jet boundary conditions are

set by modelling the plenum and jet pipe, this option would increase the number of grid nodes substantially. Further discussion on this subject will be presented in the context of the current study in Chapter 5.

Figure 1.3.5.3.1.1 Configuration of compound-angle *VGJ*, reproduced from Hassan et al. (1998)

Figure 1.3.5.3.1.2 Turbulence kinetic energy for compound-angle *VGJ*, R =jet-to-freestream velocity ratio, x/D =streamwise distance normalised by jet diameter—*SST*, -- *Baseline $k-\epsilon$* / *$k-\omega$* , standard *$k-\epsilon$* , ● experiment, reproduced from Hassan et al. (1998)

Figure 1.3.5.3.1.3 Jet fluid concentration gradient, C , for compound-angle VGJ , $R = VR$, x/D = streamwise distance normalised by jet diameter
 $—$ SST, $--$ Baseline $k-\epsilon$ / $k-\omega$, standard $k-\epsilon$, $\bullet$ experiment, reproduced from Hassan et al. (1998)

a)

b)

Figure 1.3.5.3.2.1 Jet entrance grid distribution, a) perspective view, b) top view, reproduced from Kim and Benson (1992)

2.0 Description of Numerical Algorithm

A full Navier-Stokes solver (Flow Solver - *FS*) developed by Dr. M.I. Yaras is used in simulations of the flows presented in subsequent chapters. This chapter presents an overview of the algorithm employed in the *FS* code. In *FS*, the Reynolds / Favre' averaged governing equations, expressed in strong-conservation form, are discretised in space using the finite-volume technique. Iterative solution of the spatially discretised governing equations is based on a multistage Runge-Kutta pseudo-time integration scheme with implicit residual smoothing at alternate stages. Second- and fourth-order artificial dissipation terms are added to control odd / even spatial decoupling and oscillations in regions with high gradients in flow variables. Preconditioning of the discretised equations similar to the artificial compressibility technique is employed to allow prediction of flows without limitations on the flow Mach number.

2.1 Finite Volume Discretisation

The governing equations of fluid motion can be expressed in conservation-law vector form as:

$$\frac{\partial \mathbf{Q}}{\partial t} + \frac{\partial \mathbf{E}}{\partial x} + \frac{\partial \mathbf{F}}{\partial y} + \frac{\partial \mathbf{G}}{\partial z} = \mathbf{H} \quad (2.1.1)$$

where $\mathbf{Q}$ is the solution vector, or vector of conservative variables; $\mathbf{E}$, $\mathbf{F}$, $\mathbf{G}$ are the flux vectors; and $\mathbf{H}$ is the source term. For the incompressible regime only the mass and

momentum conservation equations need to be considered, hence the flux and solution vectors take the following forms:

$$\begin{aligned}
 \mathbf{E} &= \begin{bmatrix} \rho U \\ (\rho U^2 + P) - 2\mu \frac{\partial U}{\partial x} - \tau_{xx} \\ (\rho UV) - \mu \left(\frac{\partial V}{\partial x} + \frac{\partial U}{\partial y} \right) - \tau_{xy} \\ (\rho UW) - \mu \left(\frac{\partial W}{\partial x} + \frac{\partial U}{\partial z} \right) - \tau_{xz} \end{bmatrix}, & \mathbf{F} &= \begin{bmatrix} \rho V \\ (\rho UV) - \mu \left(\frac{\partial U}{\partial y} + \frac{\partial V}{\partial x} \right) - \tau_{yx} \\ (\rho V^2 + P) - 2\mu \frac{\partial V}{\partial y} - \tau_{yy} \\ (\rho VW) - \mu \left(\frac{\partial V}{\partial z} + \frac{\partial W}{\partial y} \right) - \tau_{yz} \end{bmatrix}, \\
 \mathbf{G} &= \begin{bmatrix} \rho W \\ (\rho UW) - \mu \left(\frac{\partial W}{\partial x} + \frac{\partial U}{\partial z} \right) - \tau_{zx} \\ (\rho VW) - \mu \left(\frac{\partial V}{\partial z} + \frac{\partial W}{\partial y} \right) - \tau_{yz} \\ (\rho W^2 + P) - 2\mu \frac{\partial W}{\partial z} - \tau_{zz} \end{bmatrix}, & \mathbf{Q} &= \begin{bmatrix} \rho \\ \rho U \\ \rho V \\ \rho W \end{bmatrix}
 \end{aligned} \tag{2.1.2a-d}$$

Integrating Equation 2.1.1 over a control volume of volume $V_{c.v.}$ yields:

$$\iiint_{V_{c.v.}} \left[\frac{\partial \mathbf{Q}}{\partial t} + \frac{\partial \mathbf{E}}{\partial x} + \frac{\partial \mathbf{F}}{\partial y} + \frac{\partial \mathbf{G}}{\partial z} \right] dV_{c.v.} = \iiint_{V_{c.v.}} \mathbf{H} dV_{c.v.} \tag{2.1.3}$$

Applying Gauss' divergence theorem to the spatial derivatives gives:

$$\iiint_{V_{c.v.}} \left(\frac{\partial \mathbf{Q}}{\partial t} \right) dV_{c.v.} + \iint_{A_{c.v.}} \left(\mathbf{E} \cdot \mathbf{n}_x + \mathbf{F} \cdot \mathbf{n}_y + \mathbf{G} \cdot \mathbf{n}_z \right) dA_{c.v.} = \iiint_{V_{c.v.}} \mathbf{H} dV_{c.v.} \tag{2.1.4}$$

Assuming a spatially uniform distribution of $\mathbf{Q}$ and $\mathbf{H}$ within the control volume, and

assuming piecewise uniform distribution of fluxes over the “surface faces” of this control volume:

$$\frac{dQ_{c.v.}}{dt} V_{c.v.} + \sum_{c.v. \text{ faces}} (EA_x + FA_y + GA_z) = H_{c.v.} V_{c.v.} \quad (2.1.5)$$

Although the equations presented here are in dimensional form, their nondimensional versions are solved in *FS* to minimise round-off errors. In this context, it should be noted that all float variables in *FS* are defined with single precision. Although this yields less desirable properties with respect to round off errors when compared with double precision, single precision was chosen for the present study for the smaller *CPU* time it requires in arithmetic operations. The rationale for this trade off relates to one of the main objectives of the present study noted in Section 1.2, namely establishing the feasibility of obtaining sufficiently short turnaround times suitable for an iterative design-cycle environment.

2.2 Computational Molecule and Control-Volume Geometry

In *FS* a vertex-centred control-volume structure is employed (e.g., Swanson and Radespiel, 1991) in which the control volumes are “centred” at grid nodes. The advantage of this method is that, unlike cell-centred schemes, special interpolation schemes at computational-domain boundaries or between grid levels in the application of multigrid acceleration are not required. Figure 2.2.1 displays the resulting control volume definition. The grid node at which the control volume is centred is denoted by a dot, and the vertices of the control volume are placed at the centres of the eight grid cells (four grid

cells in 2-D) sharing this node. *FS* provides the option of 7-grid node (5 nodes in 2-D) or 27-grid node (9 nodes in 2-D) computational molecules in determining the fluxes at the

Figure 2.2.1 Cell-vertex-centred control volume
a) 2D space, b) 3D space

faces of the control volume. The larger computational molecules allow nonuniform description of fluxes over each face of the control volume, which would be expected to translate into coarser grid requirements for grid-independent solutions. This, however, would come at the expense of a higher operation count associated with the larger molecule. Therefore, comparative studies must be performed to establish whether the use of the larger of the two computational molecules provides benefits in terms of reduction in overall computation time to obtain converged, grid independent solutions. Such comparisons were not performed for the present study, and thus the more

conventional 7-node molecule (5 nodes in 2-D) was used. This yielded piecewise uniform distribution of fluxes over each of the faces of the control volume, obtained by averaging the fluxes at the two nodes on either side of the face.

In *FS* control-volume surface areas and volumes are calculated using the generalised methods of Vinokur (1989) for nonorthogonal geometries. The surface areas of the four segments making up each of the 6 faces of a control volume, as well as the volumes of the eight segments making up the control volume are calculated individually. The same approach is followed in two-dimensional space as well.

2.3 Artificial Dissipation

An artificial dissipation term, D_a , is added explicitly into the finite volume formulation to suppress odd / even decoupling of nodal velocity and pressure distributions in space, and to damp oscillations in regions of severe spatial gradients such as shocks or stagnation points. The addition is made as shown:

$$\frac{dQ_{c.v.}}{dt} V_{c.v.} + \sum_{c.v. \text{ faces}} (EA_x + FA_y + GA_z) = H_{c.v.} V_{c.v.} + D_{a_{c.v.}} \quad (2.3.1)$$

Following the work of Jameson et al. (1981), this artificial dissipation term is constructed from second and fourth order differences of the conservation variables, which are respectively first- and third-order accurate on a uniform mesh. Odd / even point

decoupling is controlled by the fourth-order term whereas damping of oscillations due to severe localised pressure gradients is provided by the second-order term:

$$D_a = \left(d_{i+\frac{1}{2},j,k} - d_{i-\frac{1}{2},j,k} \right) + \left(d_{i,j+\frac{1}{2},k} - d_{i,j-\frac{1}{2},k} \right) + \left(d_{i,j,k+\frac{1}{2}} - d_{i,j,k-\frac{1}{2}} \right) \quad (2.3.2)$$

where,

$$\begin{aligned} d_{i+\frac{1}{2},j,k} = & \lambda_{i+\frac{1}{2},j,k} \varepsilon_{2,i+\frac{1}{2},j,k} \left(Q_{i+1,j,k} - Q_{i,j,k} \right) \\ & - \varepsilon_{4,i+\frac{1}{2},j,k} \left(Q_{i+2,j,k} - 3Q_{i+1,j,k} + 3Q_{i,j,k} - Q_{i-1,j,k} \right) \end{aligned} \quad (2.3.3)$$

and similarly for the remaining d terms. The artificial dissipation is scaled by the spectral radius λ of the inviscid flux Jacobian (e.g, Jameson et al., 1981). The coefficients ε_2 and ε_4 , which control activation of fourth-order dissipation in smooth flow regions and second-order dissipation in the vicinity of large spatial gradients, are defined as:

$$\begin{aligned} \varepsilon_{2,i+\frac{1}{2},j,k} &= \kappa_2 \max \left(v_{i+1,j,k}; v_{i,j,k} \right) \\ \varepsilon_{4,i+\frac{1}{2},j,k} &= \max \left(0; \left(\kappa_4 - \varepsilon_{2,i+\frac{1}{2},j,k} \right) \right) \end{aligned} \quad (2.3.4a-b)$$

where,

$$v_{i,j,k} = \frac{|P_{i+1,j,k} - 2P_{i,j,k} + P_{i-1,j,k}|}{P_{i+1,j,k} + 2P_{i,j,k} + P_{i-1,j,k}} \quad (2.3.5)$$

and κ_2, κ_4 are user-specified constants that must be kept as small as possible to minimise the extent of error in the predicted flow due to artificial dissipation. For the flow simulations contained in this study κ_2 was kept constant at 0.25, although this value is judged to have negligible impact on the results due to the low levels of local pressure gradients encountered in the present test cases.

2.4 Integration in Pseudo-time

The *FS* code uses Runge-Kutta integration (Jameson et al., 1981) to solve the discretised governing equations by marching in pseudo-time to a “steady state”. Rewriting Equation 2.3.1 such that convective / diffusive flux, source and artificial dissipation terms are collectively defined as the “residual,” R , yields:

$$V_{c.v.} \frac{\partial Q}{\partial \tau} = -R \quad (2.4.1)$$

Thus, the result of marching to a steady state in pseudo-time, t , amounts to seeking the conditions for $R=0$. A two-stage Runge-Kutta integration scheme proceeds as:

$$\begin{aligned} Q_{i,j,k}^{(0)} &= Q_{i,j,k}^{(n)} \\ Q_{i,j,k}^{(1)} &= Q_{i,j,k}^{(0)} - \alpha_1 \frac{\Delta t_{i,j,k}}{V_{c.v.}} R_{i,j,k}^{(0)} \\ Q_{i,j,k}^{(2)} &= Q_{i,j,k}^{(0)} - \alpha_2 \frac{\Delta t_{i,j,k}}{V_{c.v.}} R_{i,j,k}^{(1)} \\ Q_{i,j,k}^{(n+1)} &= Q_{i,j,k}^{(2)} \end{aligned} \quad (2.4.2a-d)$$

The solution vector associated with the “old” and “new” pseudo-time levels are indicated by n and $n+1$ respectively. Although use of four stages is more common in published *CFD* literature, for the flows considered in this report a two-stage scheme with $\alpha_1 = \alpha_2 = 1$ was found to be optimum for rapid, monotonic convergence. Additionally, evaluation of the artificial dissipation terms in only the first of the two stages was found to be sufficient. Implicit residual smoothing is performed at alternate stages (i.e., after the 2nd stage in this instance) to increase the allowable pseudo-time increment Δt . The smoothing operator

used in the present study, incorporated into the 2nd stage of the Runge-Kutta integration, is expressed as:

$$\begin{aligned} & \left(I - \beta_i \frac{\Delta t_{i,j,k}^2}{V_{c,v,i,j,k}^2} \lambda_i^2 \delta_i^2 \right) \left(I - \beta_j \frac{\Delta t_{i,j,k}^2}{V_{c,v,i,j,k}^2} \lambda_j^2 \delta_j^2 \right) \left(I - \beta_k \frac{\Delta t_{i,j,k}^2}{V_{c,v,i,j,k}^2} \lambda_k^2 \delta_k^2 \right) \Delta Q \\ &= \frac{-\alpha_2 \Delta t_{i,j,k}}{V_{c,v,i,j,k}} R_{i,j,k}^{(1)} \end{aligned} \quad (2.4.3)$$

where ΔQ is equal to $Q_{i,j,k}^{(2)} - Q_{i,j,k}^{(0)}$, λ is the spectral radius of the inviscid flux Jacobian, and the second order difference operator (δ_i^2) is defined as:

$$\delta_i^2 \Delta Q = (\Delta Q_{i+1} - 2\Delta Q_i + \Delta Q_{i-1}) \quad (2.4.4)$$

For the flows considered in the present study, a value of 1.0 for the coefficients β_i , β_j and β_k was found to work well, and no further optimization of these coefficients was attempted. In *FS*, Equation (2.4.3) is implemented in the framework of either the Alternating-Direction-Implicit (*ADI*) procedure (e.g., Pulliam, 1985, 1993) or the Strongly Implicit Procedure (*SIP*) of Stone (1968). *FS* uses the modified version of the *SIP* scheme, *MSIP*, as described by Schneider and Zedan (1981). Apart from the initial flat-plate benchmarks, all simulation results presented herein were obtained using the *MSIP* procedure, with a value of 0.4 for the coefficient, α , as described by Schneider and Zedan. This choice was based on improved stability of the pseudo-time marching process facilitated by the *MSIP* procedure.

2.5 Convergence Acceleration Techniques

As was pointed out in Section 1.2, use of *CFD* in a design cycle is only desirable if computation times are sufficiently short. Thus, it is essential that all available “acceleration techniques” be used in such *CFD* algorithms. The algorithm used in the present study benefits from several such techniques.

In addition to the use of implicit residual smoothing to increase the pseudo-time step in the Runge-Kutta integration, the time steps are evaluated locally for each control volume to maximise the rate of advancement of the solution in pseudo-time. The upper limit for the local time step is set by the convective and diffusive time-step restrictions as follows:

$$\frac{1}{\Delta t_{i,j,k}} \geq \left(\frac{1}{\Delta t_c} + \frac{1}{\Delta t_d} \right)_{i,j,k} = \left(\frac{\sum \lambda}{CFL V_{c.v.}} + \frac{1}{K_d V_{c.v.}^2} \sum \frac{\mu + \mu_t}{\rho} A_{c.v.}^2 \right)_{i,j,k} \quad (2.5.1)$$

where *CFL* and *K_d* are user-specified constants to control convective and diffusive time-step limits, respectively. The summations on the right-hand side of this equation are performed over the faces of the control volume. Although local time stepping yields transients that are warped in pseudo-time, this does not pose a problem since only the converged “steady-state” solution is of interest.

To further accelerate convergence to “steady state” with the pseudo-time marching procedure, the Full-Approximation-Storage (*FAS*) multigrid scheme (Brandt, 1977) is used. In *FS* the multigrid procedure utilises V-cycles and the coarser grid levels are

constructed by dropping every other node of the next finer grid in all grid directions. For the flow simulations presented herein, two grid levels were found to provide a good trade off between increased rate of convergence offered by the multigrid procedure and increased computational effort due to the multigrid “overhead” associated with restriction and prolongation operations between grid levels. Furthermore, for each multigrid V-cycle, one time step is performed on the fine-grid level at the beginning and end of the cycle and two time steps are taken on the coarser grid. Linear interpolation is used in prolongating corrections to the solution variables from the coarser grid and injection is used to interpolate (restrict) the fine grid solution to the coarser grid. Use of area-weighted interpolation for the restriction process is available as an option in *FS*, and this option was utilised in the first two benchmarks presented here. Although this yields improved interpolation accuracy for internal nodes, injection was found to be preferable to avoid spurious oscillations near periodic computational-domain boundaries. In light of the reduced spatial resolution on the coarser grid levels of a multigrid cycle, the physical and artificial diffusion terms were calculated on the fine-grid levels only.

2.6 Artificial Compressibility

Preconditioning similar to the concept of compressibility is described by Chorin (1967), and Rizzi and Eriksson (1985) is used in *FS* to allow simulation of flows ranging from incompressible to supersonic speeds. In the preconditioned form of the governing equations the pseudo-time derivative of the solution vector:

$$\frac{\partial \mathbf{Q}}{\partial \tau} = \begin{Bmatrix} \partial \rho / \partial \tau \\ \partial(\rho U) / \partial \tau \\ \partial(\rho V) / \partial \tau \\ \partial(\rho W) / \partial \tau \end{Bmatrix} \quad (2.6.1)$$

is replaced by,

$$\frac{\partial \mathbf{Q}}{\partial \tau} = \begin{Bmatrix} (1/C_{AD})^2 \frac{\partial P}{\partial \tau} \\ (a_1/C_{AD})^2 U \frac{\partial P}{\partial \tau} + \frac{\partial(\rho U)}{\partial \tau} \\ (a_2/C_{AD})^2 V \frac{\partial P}{\partial \tau} + \frac{\partial(\rho V)}{\partial \tau} \\ (a_3/C_{AD})^2 W \frac{\partial P}{\partial \tau} + \frac{\partial(\rho W)}{\partial \tau} \end{Bmatrix} \quad (2.6.2)$$

The $a_{1,3}$ terms are user-specified constants and C_{AD} is an *artificial compressibility* parameter that is typically scaled on the local flow velocity or on a reference velocity representative of the overall flow motion in question. In the former choice, a lower limit is set for C_{AD} . For the flows considered in this study, scaling of C_{AD} upon the local flow

velocity was found to provide better stability in pseudo-time integration. The constants α_1 to α_3 enable independent scaling of the preconditioning for each of the governing equations. Due to time constraints, however, individual optimisation of the extent of preconditioning for each of the governing equations was not feasible. The constants α_1 to α_3 were thus set to unity.

2.7 Procuring Convergence

Once a rapid, monotonic convergence of the iterative solution is established through means described in previous sections, it is equally important to establish sound criteria for declaring the solution to be “converged”. For all the simulations presented herein, convergence was judged on the basis of several criteria. As a minimum, these criteria included upper limits for maximum changes in nodal pressure, velocity and turbulence quantities over one iteration cycle, low nodal residuals for all governing equations being solved, as well as global conservation of mass to satisfactory levels (typically 1%).

3.0 Modelling of Turbulence

This chapter describes the implementation of five turbulence models, all of which are based on the eddy-viscosity hypothesis of Boussinesq (1877). Since features of the complex flows being studied herein include streamline curvature, three-dimensional separation and adverse pressure gradients, such models may seem inappropriate due to the likelihood of significant turbulence anisotropy. Eddy-viscosity based models have been chosen rather than more elaborate Reynolds Stress Models (*RSM*'s) because the substantial increase in computational expense of such closures is presently unacceptable for design-cycle analysis. Although Algebraic Stress Models (*ASM*'s) are less expensive than *RSM*'s, these were not chosen since they incorporate a large number of constants that appear to require calibration for individual flows. Thus, the purpose of the present study is to determine the level of prediction accuracy attainable using eddy-viscosity models for intake design.

The five turbulence models that are included in the present investigation are Chien's (1982) Low-*Re* k - ϵ model, a two-layer k - ϵ model (Rodi et al., 1988), a k - ω model (Wilcox, 1988), the Shear-Stress-Transport (*SST*) model of Menter (1994), and the one-equation model (*SA*) of Spalart and Allmaras (1992). The k - ω , *SST* and *SA* models were chosen due to their documented success in predicting a variety of related flows (e.g., Bardina et al., 1997; Heidegger et al., 1999; Hellsten 1996). Although the performance of k - ϵ models is known to be relatively poor in conditions of adverse streamwise pressure

gradient, these models were chosen as a baseline to compare with the performance of others, particularly the *SST* model which reduces to the *k-ε* model away from solid surfaces. The original closure constants have been used for each model, rather than calibrating them for each flow, in order to test the universality of the models in their standard form. The following sections describe model implementation, convergence issues and boundary conditions. The equations for the turbulence models are presented in terms of normalised variables denoted by a (*) superscript, and are given in terms of the same conservation-variable form as the mass and momentum conservation equations (Equation 2.1.1) in Section 2.1. The coupling between these equations and the “primary” set of governing equations (Equation 2.1.1) is provided by the Reynolds stress tensor:

$$\tau_{ij} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \left(\mu_t \frac{\partial U_l}{\partial x_l} + \rho k \right) \quad (3.1)$$

which appears in the flux terms of the momentum equations.

In *FS*, the turbulence equations are solved together with the mass and momentum conservation equations in a coupled manner. The artificial dissipation terms included in the discretised turbulence-model equations are of the same form as those for mass and momentum conservation. Due to the nature of the *PDE*'s for the turbulence models, no preconditioning (artificial compressibility) is needed. The turbulence equations make use of the same local time steps as the mass and momentum equations. *FS* provides the option

of using multigrid acceleration on the turbulence equations as well. However, for the simulations presented here, wherein the turbulence *PDE*'s are integrated to the wall, this approach was found to cause accuracy and stability problems near walls. Hence, multigrid acceleration was limited to the mass and momentum equations only.

3.1 The k - ϵ Turbulence Model

The standard k - ϵ turbulence model (Launder and Spalding, 1974) written in conservation-law form contains the following Q , H and flux vectors:

$$Q^* = \begin{Bmatrix} \rho^* k^* \\ \rho^* \epsilon^* \end{Bmatrix}, \quad H^* = \begin{Bmatrix} H_k^* \\ H_\epsilon^* \end{Bmatrix}, \quad E^* = \begin{Bmatrix} \rho^* U^* k^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_k} \right) \frac{\partial k^*}{\partial x^*} \\ \rho^* U^* \epsilon^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_\epsilon} \right) \frac{\partial \epsilon^*}{\partial x^*} \end{Bmatrix}, \quad (3.1.1a-e)$$

$$F^* = \begin{Bmatrix} \rho^* V^* k^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_k} \right) \frac{\partial k^*}{\partial y^*} \\ \rho^* V^* \epsilon^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_\epsilon} \right) \frac{\partial \epsilon^*}{\partial y^*} \end{Bmatrix}, \quad G^* = \begin{Bmatrix} \rho^* W^* k^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_k} \right) \frac{\partial k^*}{\partial z^*} \\ \rho^* W^* \epsilon^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_\epsilon} \right) \frac{\partial \epsilon^*}{\partial z^*} \end{Bmatrix}$$

where the source terms are defined as,

$$H_k^* = \underbrace{\frac{1}{\text{Re}_{ref}} \tau_{ij}^* \frac{\partial U_i^*}{\partial x_j^*}}_{\text{Production of } k} - \underbrace{\text{Re}_{ref} \rho^* \epsilon^*}_{\text{Destruction of } k} \quad (3.1.2)$$

$$H_\epsilon^* = \underbrace{C_{\epsilon 1} \frac{1}{\text{Re}_{ref}} \frac{\epsilon^*}{k^*} \tau_{ij}^* \frac{\partial U_i^*}{\partial x_j^*}}_{\text{Production of } \epsilon} - \underbrace{C_{\epsilon 2} \text{Re}_{ref} \rho^* \frac{\epsilon^{*2}}{k^*}}_{\text{Destruction of } \epsilon} \quad (3.1.3)$$

with the Reynolds stress tensor given as,

$$\tau_{ij}^* = \mu_t^* \left[\left(\frac{\partial U_i^*}{\partial x_j^*} + \frac{\partial U_j^*}{\partial x_i^*} \right) - \frac{2}{3} \frac{\partial U_l^*}{\partial x_l^*} \delta_{ij} \right] - \frac{2}{3} \rho^* k^* \delta_{ij} \quad (3.1.4)$$

In these expressions, Einstein's summation convention is used with the indices i, j, k and Re_{ref} is a Reynolds number based on the reference length, velocity, and viscosity scales chosen for nondimensionalising the equations. Eddy-viscosity is then related to k and ε through:

$$\mu_t^* = C_\mu \rho^* \frac{k^{*2}}{\varepsilon^*} \quad (3.1.5)$$

and the closure coefficients, consistent with the standard version of the k - ε model, are:

$$\begin{aligned} C_{\varepsilon 1} &= 1.44, \\ C_{\varepsilon 2} &= 1.92, \\ C_\mu &= 0.09, \\ \sigma_k &= 1.0, \\ \sigma_\varepsilon &= 1.3 \end{aligned} \quad (3.1.6a-e)$$

3.1.1 Linearisation of Source Terms

The source terms of the turbulence equations are often linearised to improve the stability characteristics of their iterative solution. This is because purely explicit treatment of the production and destruction terms contained in these source terms would result in severe time-step limits in regions of flow where they dominate convection and diffusion. Various forms of linearisation have been employed in the published literature (e.g., Mavriplis and

Martinelli, 1994; Zheng et al., 1997). The linearisation used in *FS* is as follows:

$$H^{*n+1} = \frac{\partial H^*}{\partial Q^*} \Delta Q^* + H^{*n} \quad (3.1.1.1)$$

where,

$$\frac{\partial H^*}{\partial Q^*} = \begin{bmatrix} -2 \text{Re}_{\text{ref}} \left\{ \frac{C_\mu \rho^*}{\mu_t^*} \right\} k^* & 0 \\ 0 & -\frac{3}{2} C_{\varepsilon 2} \text{Re}_{\text{ref}} \left\{ \frac{C_\mu \rho^*}{\mu_t^*} \right\}^{\frac{1}{2}} \varepsilon^{*\frac{1}{2}} \end{bmatrix} \quad (3.1.1.2)$$

In constructing the $\partial H^* / \partial Q^*$ matrix, those off-diagonal terms that contribute to maximising diagonal dominance, namely the destruction terms, are treated implicitly.

3.1.2 Chien's Low-Re k - ε Model

Low-*Re* modelling using the k - ε turbulence closure necessitates the application of damping functions as the equations are integrated to the wall. One such model was proposed by Chien (1982). This formulation replaces the standard ε variable by an alternative one, $\tilde{\varepsilon}$, such that a simple wall boundary condition is obtained for $\tilde{\varepsilon}$, namely $\tilde{\varepsilon}=0$. All other deviations from the standard k - ε model are limited to the source terms:

$$H_k^* = \underbrace{\frac{1}{\text{Re}_{\text{ref}}} \tau_{ij}^* \frac{\partial U_i^*}{\partial x_j^*}}_{\text{Production of } k} - \underbrace{\text{Re}_{\text{ref}} \rho^* (\tilde{\varepsilon}^* + \varepsilon_0^*)}_{\text{Destruction of } k} \quad (3.1.2.1)$$

$$H_\varepsilon^* = \underbrace{\frac{1}{\text{Re}_{\text{ref}}} C_{\varepsilon 1} \frac{\tilde{\varepsilon}^*}{k^*} \tau_{ij}^* \frac{\partial U_i^*}{\partial x_j^*}}_{\text{Production of } \varepsilon} - \underbrace{\text{Re}_{\text{ref}} C_{\varepsilon 2} f_2 \rho^* \frac{\tilde{\varepsilon}^{*2}}{k^*} + \frac{1}{\text{Re}_{\text{ref}}} \rho^* E^*}_{\text{Destruction of } \varepsilon} \quad (3.1.2.2)$$

where,

$$\varepsilon_0^* = \frac{2\mu^*}{\text{Re}_{\text{ref}}^2} \frac{k^*}{\rho^* y_{\text{wall}}^2} \quad (3.1.2.3)$$

$$\varepsilon^* = \tilde{\varepsilon}^* + \varepsilon_0^* \quad (3.1.2.4)$$

$$f_2 = 1 - 0.22e^{-(\text{Re}_t/6)^2} \quad (3.1.2.5)$$

$$\text{Re}_t = \frac{\rho^* k^{*2}}{\mu^* \tilde{\varepsilon}^*} \quad (3.1.2.6)$$

$$E^* = \frac{-2\mu^*}{\rho^*} \frac{\tilde{\varepsilon}^*}{y_{\text{wall}}^2} e^{-y^*/2} \quad (3.1.2.7)$$

and y_{wall} is the normal distance to the wall. In evaluating y_{wall} for each internal node, the code *FS* determines the wall node that is at the shortest distance to the node, and then calculates the projection of this distance onto the wall normal. To prevent this approach from mishandling certain nodes, such as a node that is located just upstream and above of the leading edge of a flat plate, the aforementioned projection onto the wall normal is performed only if the normal distance is within a user-specified fraction of the absolute distance; otherwise the y_{wall} value for the internal node is set to a large default value.

The original eddy-viscosity relationship to k and ε given in Equation (3.1.5) is modified for wall damping effects as:

$$\mu_t^* = C_\mu f_\mu \rho^* \frac{k^{*2}}{\tilde{\varepsilon}^*} \quad (3.1.2.8)$$

where

$$f_\mu = 1 - e^{-0.0115y^+} \quad (3.1.2.9)$$

and the closure coefficients, slightly different than the standard versions, are:

$$\begin{aligned} C_{\varepsilon 1} &= 1.35, \\ C_{\varepsilon 2} &= 1.80, \\ C_\mu &= 0.09, \\ \sigma_k &= 1.0, \\ \sigma_\varepsilon &= 1.3 \end{aligned} \quad (3.1.2.10a-e)$$

3.1.3 Two-Layer k - ε Model

Another method of addressing the near wall difficulties of the standard k - ε model is the two-layer approach which replaces the partial differential equation (*PDE*) for ε with an algebraic equation near the wall. This circumvents both the inherent numerical stiffness of the low- Re ε equation containing damping functions and the relatively poor accuracy of the *PDE* for ε known to prevail in the near-wall regions. Two similar versions of two-layer k - ε models were proposed by Chen and Patel (1988) and Rodi et al. (1988). The algebraic equation for ε used in these models is expressed as:

$$\varepsilon^* = \frac{1}{Re_{ref}} \left(\frac{k^{*3/2}}{\ell_\varepsilon^*} \right) \quad (3.1.3.1)$$

The two models differ primarily in the choice of length scale (ℓ_ε^*). Chen and Patel chose the one originally proposed by Wolfshtein (1969):

$$\ell_\varepsilon^* = \kappa C_\mu^{-3/4} y_{wall}^* \left[1 - e^{-Re_y/5.08} \right] \quad (3.1.3.2)$$

where,

$$Re_y = Re_{ref} \frac{\rho^* \sqrt{k^*} y_{wall}^*}{\mu^*} \quad (3.1.3.3)$$

whereas, Rodi et al. adopted the model of Norris and Reynolds (1975),

$$\ell_\varepsilon^* = \frac{\kappa C_\mu^{-3/4} y^*}{(1 + 5.3/Re_y)} \quad (3.1.3.4)$$

In the near-wall region where ε is obtained algebraically, eddy-viscosity is calculated from:

$$\mu_t^* = Re_{ref} \rho^* C_\mu \sqrt{k^*} \ell_\mu^* \quad (3.1.3.5)$$

where,

$$\ell_\mu^* = \kappa C_\mu^{-3/4} y^* \left[1 - e^{-Re_y/A_\mu} \right] \quad (3.1.3.6)$$

and

$$A_\mu \Big|_{Chen} = 70.0, A_\mu \Big|_{Rodi} = 50.5 \quad (3.1.3.7)$$

Both models require a switching criterion to determine the distance from the wall within which to apply the algebraic ε equation. Chen and Patel switch at a distance that corresponds to $Re_y = 200$, whereas Rodi et al. use a damping function value (the bracketed term in Equation 3.1.3.6) of 0.95. Although both models are available in the *FS* code, due to their similarity, the simulations of the present study were based on the model of Rodi et al. only.

3.2 The k - ω Turbulence Model

The k - ω turbulence model of Wilcox (1988) written in conservation-law form contains the following Q , H and flux vectors:

$$Q^* = \begin{Bmatrix} \rho^* k^* \\ \rho^* \omega^* \end{Bmatrix}, \quad H^* = \begin{Bmatrix} H_k^* \\ H_\omega^* \end{Bmatrix}, \quad E^* = \begin{Bmatrix} \rho^* U^* k^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_k} \right) \frac{\partial k^*}{\partial x^*} \\ \rho^* U^* \omega^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_\omega} \right) \frac{\partial \omega^*}{\partial x^*} \end{Bmatrix}, \quad (3.2.1a-e)$$

$$F^* = \begin{Bmatrix} \rho^* V^* k^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_k} \right) \frac{\partial k^*}{\partial y^*} \\ \rho^* V^* \omega^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_\omega} \right) \frac{\partial \omega^*}{\partial y^*} \end{Bmatrix}, \quad G^* = \begin{Bmatrix} \rho^* W^* k^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_k} \right) \frac{\partial k^*}{\partial z^*} \\ \rho^* W^* \omega^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_\omega} \right) \frac{\partial \omega^*}{\partial z^*} \end{Bmatrix}$$

where the source terms are defined as,

$$H_k^* = \underbrace{\frac{1}{\text{Re}_{ref}} \tau_{ij}^* \frac{\partial U_i^*}{\partial x_j^*}}_{\text{Production of } k} - \underbrace{\text{Re}_{ref} C_\mu \rho^* k^* \omega^*}_{\text{Destruction of } k} \quad (3.2.2)$$

$$H_\omega^* = \underbrace{\frac{C_{\omega 1}}{\text{Re}_{ref}} \frac{\omega^*}{k^*} \tau_{ij}^* \frac{\partial U_i^*}{\partial x_j^*}}_{\text{Production of } \omega} - \underbrace{\text{Re}_{ref} C_{\omega 2} \rho^* \omega^{*2}}_{\text{Destruction of } \omega} \quad (3.2.3)$$

Eddy-viscosity is then related to k and ω through:

$$\mu_t^* = \rho^* \frac{k^*}{\omega^*} \quad (3.2.4)$$

and the closure coefficients are,

$$\begin{aligned}
 C_{\omega 1} &= 0.553, \\
 C_{\omega 2} &= 0.075, \\
 C_{\mu} &= 0.09, \\
 \sigma_k &= 2.0, \\
 \sigma_{\omega} &= 2.0
 \end{aligned}
 \tag{3.2.5a-e}$$

As with the k - ϵ model, source terms are linearised as per Equation 3.1.1.1, where:

$$\frac{\partial \mathbf{H}^*}{\partial \mathbf{Q}^*} = \begin{bmatrix} -2 \text{Re}_{ref} C_{\mu} \omega^* & 0 \\ 0 & -2 \text{Re}_{ref} C_{\omega 2} \omega^* \end{bmatrix}
 \tag{3.2.6}$$

3.3 Menter's Shear-Stress-Transport Turbulence Model

The Shear-Stress-Transport (SST) turbulence model (Menter, 1994) applies the k - ω model near the wall and the k - ε model elsewhere. The weaknesses of each individual model is thus removed since k - ε is known to behave poorly near the wall whereas k - ω results tend to be sensitive to the prescribed freestream turbulence dissipation rate (Menter, 1992a). The model is formulated in a k - ω framework by transforming the k - ε equation.

The formation of the SST model is as follows:

$$Q^* = \begin{Bmatrix} \rho^* k^* \\ \rho^* \omega^* \end{Bmatrix}, H^* = \begin{Bmatrix} H_k^* \\ H_\omega^* \end{Bmatrix}, E^* = \begin{Bmatrix} \rho^* U^* k^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_k} \right) \frac{\partial k^*}{\partial x^*} \\ \rho^* U^* \omega^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_\omega} \right) \frac{\partial \omega^*}{\partial x^*} \end{Bmatrix}, \quad (3.3.1a-e)$$

$$F^* = \begin{Bmatrix} \rho^* V^* k^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_k} \right) \frac{\partial k^*}{\partial y^*} \\ \rho^* V^* \omega^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_\omega} \right) \frac{\partial \omega^*}{\partial y^*} \end{Bmatrix}, G^* = \begin{Bmatrix} \rho^* W^* k^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_k} \right) \frac{\partial k^*}{\partial z^*} \\ \rho^* W^* \omega^* - \frac{1}{\text{Re}_{ref}} \left(\mu^* + \frac{\mu_t^*}{\sigma_\omega} \right) \frac{\partial \omega^*}{\partial z^*} \end{Bmatrix}$$

where the source terms are defined as,

$$H_k^* = \underbrace{\frac{1}{\text{Re}_{ref}} \tau_{ij}^* \frac{\partial U_i^*}{\partial x_j^*}}_{\text{Production of } k} - \underbrace{\text{Re}_{ref} C_\mu \rho^* k^* \omega^*}_{\text{Destruction of } k} \quad (3.3.2)$$

$$H_{\omega}^* = \underbrace{\frac{C_{\omega 1}}{\text{Re}_{ref}} \frac{\omega^*}{k^*} \tau_{ij}^* \frac{\partial U_i^*}{\partial x_j^*}}_{\text{Production of } \omega} - \underbrace{\text{Re}_{ref} C_{\omega 2} \rho^* \omega^{*2}}_{\text{Destruction of } \omega} + \underbrace{\frac{2\rho^* (1-F_1)}{\text{Re}_{ref}} \frac{\partial k^*}{\sigma_{\omega} \omega^*} \frac{\partial \omega^*}{\partial x_j^*} \frac{\partial \omega^*}{\partial x_j^*}}_{\text{Cross-Diffusion term}} \quad (3.3.3)$$

and,

$$F_1 = \tanh \left\{ \left[\min \left(\max \left(\frac{1}{\text{Re}_{ref}} \frac{\sqrt{k^*}}{C_{\mu} \omega^* y_{wall}^*}, \frac{1}{\text{Re}_{ref}^2} \frac{500 \mu^*}{\rho^* \omega^* y_{wall}^{*2}} \right), \frac{\rho_{ref} V_{ref}^2}{L_{ref}^2} \frac{4 \rho^* k^*}{\sigma_{\omega b} CD_{k\omega}^* y_{wall}^{*2}} \right) \right]^4 \right\} \quad (3.3.4)$$

is a blending function with $CD_{k\omega}^*$ evaluated as

$$CD_{k\omega}^* = \max \left[\frac{\rho_{ref} V_{ref}^2}{L_{ref}^2} \left(\frac{2\rho^*}{\sigma_{\omega b} \omega^*} \frac{\partial k^*}{\partial x_j^*} \frac{\partial \omega^*}{\partial x_j^*} \right); 10^{-20} \right] \quad (3.3.5)$$

Values for $C_{\omega 1}$, $C_{\omega 2}$, σ_k and σ_{ω} are evaluated from the original $k-\omega$ and transformed $k-\varepsilon$ constants as follows:

$$\begin{pmatrix} C_{\omega 1} \\ C_{\omega 2} \\ \sigma_k \\ \sigma_{\omega} \end{pmatrix}_{SST} = F_1 \begin{pmatrix} C_{\omega 1} \\ C_{\omega 2} \\ \sigma_k \\ \sigma_{\omega} \end{pmatrix}_a + (1 - F_1) \begin{pmatrix} C_{\omega 1} \\ C_{\omega 2} \\ \sigma_k \\ \sigma_{\omega} \end{pmatrix}_b \quad (3.3.6)$$

where,

$$k - \omega: \quad C_{\omega 1a} = 0.553, \quad C_{\omega 2a} = 0.075, \quad \sigma_{ka} = 1.176, \quad \sigma_{\omega a} = 2.0 \quad (3.3.7a-d)$$

$$\text{transformed } k - \varepsilon: \quad C_{\omega 1b} = 0.440, \quad C_{\omega 2b} = 0.0828, \quad \sigma_{kb} = 1.0, \quad \sigma_{\omega b} = 1.168 \quad (3.3.8a-d)$$

and C_{μ} retains the same value as for the previous models.

In the linearisation of the source terms, the cross-diffusion term appearing in Equation 3.3.3 is treated explicitly in *FS*, resulting in an identical $\partial \mathbf{H}^* / \partial \mathbf{Q}^*$ matrix to that of the *k- ω* model given in Equation 3.2.6. The eddy-viscosity is calculated using:

$$\mu_t^* = \frac{0.31 \rho^* k^*}{\max \left(0.31 \omega^*; \frac{1}{\text{Re}_{ref}} \Omega^* F_2 \right)} \quad (3.3.9)$$

where,

$$F_2 = \tanh \left\{ \left[\max \left(\frac{1}{\text{Re}_{ref}} \frac{2\sqrt{k^*}}{C_\mu \omega^* y_{wall}^*}; \frac{1}{\text{Re}_{ref}^2} \frac{500 \mu^*}{\rho^* \omega^* y_{wall}^{*2}} \right) \right]^2 \right\} \quad (3.3.10)$$

and Ω^* is the magnitude of local vorticity.

This formulation of μ_t ensures that the conventional definition (Equation 3.2.4) is employed in zero and mild pressure gradients only. For significant adverse pressure gradient, Menter's so-called *shear-stress-transport limitation* is applied to remove the non-physical instantaneous response of Reynolds-stresses to changes in shear-strain rate implicit in the Boussinesq treatment of eddy-viscosity.

3.4 The Spalart-Allmaras Turbulence Model

The Spalart-Allmaras (SA) one-equation model (1992) solves for a modified eddy-viscosity ($\tilde{\nu}$) and the Q , H and flux terms of the conservation-law form of the model are as follows:

$$\begin{aligned} Q^* &= \{\rho^* \tilde{\nu}^*\}, & E^* &= \left\{ \rho^* U^* \tilde{\nu}^* - \frac{1}{\sigma \text{Re}_{ref}} (\mu^* + \rho^* \tilde{\nu}^*) \frac{\partial \tilde{\nu}^*}{\partial x^*} \right\}, \\ F^* &= \left\{ \rho^* V^* \tilde{\nu}^* - \frac{1}{\sigma \text{Re}_{ref}} (\mu^* + \rho^* \tilde{\nu}^*) \frac{\partial \tilde{\nu}^*}{\partial y^*} \right\}, \\ G^* &= \left\{ \rho^* W^* \tilde{\nu}^* - \frac{1}{\sigma \text{Re}_{ref}} (\mu^* + \rho^* \tilde{\nu}^*) \frac{\partial \tilde{\nu}^*}{\partial z^*} \right\}, \end{aligned} \quad (3.4.1a-e)$$

$$H^* = \underbrace{C_{b1} \tilde{\Omega}^* \rho^* \tilde{\nu}^*}_{\text{Production of } \tilde{\nu}^*} - \underbrace{\frac{C_{w1} f_w \rho^*}{\text{Re}_{ref}} \left(\frac{\tilde{\nu}^*}{y_{wall}^*} \right)^2}_{\text{Destruction of } \tilde{\nu}^*} + \underbrace{\frac{C_{b2} \rho^*}{\sigma \text{Re}_{ref}} \frac{\partial \tilde{\nu}^*}{\partial x_j^*} \frac{\partial \tilde{\nu}^*}{\partial x_j^*}}_{\text{additional diffusion term}}$$

Auxiliary equations for use with this source term are as follows:

$$\mu_t^* = \tilde{\nu}^* f_{v1} \rho^* \quad (3.4.2)$$

$$\tilde{\Omega}^* = \Omega^* + \frac{1}{\text{Re}_{ref}} \frac{\tilde{\nu}^*}{\kappa^2 y_{wall}^{*2}} f_{v2} \quad (3.4.3)$$

$$f_{v1} = \frac{\chi^3}{\chi^3 + C_{v1}^3}, \quad f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}} \quad (3.4.4a-b)$$

$$\chi = \frac{\tilde{\nu}^*}{\nu^*} \quad (3.4.5)$$

$$f_w = g \left[\frac{1 + C_{w3}^6}{g^6 + C_{w3}^6} \right]^{\frac{1}{6}} \quad (3.4.6)$$

$$g = r + C_{w2} (r^6 - r) \quad (3.4.7)$$

$$r = \frac{1}{\text{Re}_{\text{ref}}} \frac{\tilde{v}^*}{\tilde{\Omega}^* \kappa^2 y_{\text{wall}}^{*2}} \quad (3.4.8)$$

The closure coefficients are:

$$\begin{aligned} C_{b1} &= 0.1355, \\ C_{b2} &= 0.622, \\ \sigma &= 0.667, \\ \kappa &= 0.41, \\ C_{v1} &= 7.1, \\ C_{w2} &= 0.3, \\ C_{w3} &= 2.0 \end{aligned} \quad (3.4.9a-g)$$

and C_{w1} is determined from,

$$C_{w1} = C_{b1} / \kappa^2 + (1 + C_{b2}) / \sigma \quad (3.4.10)$$

Finally, linearisation of the source term is based on a $\partial H^* / \partial Q^*$ evaluated as:

$$\frac{\partial H^*}{\partial Q^*} = -2 \frac{C_{w1} f_w}{\text{Re}_{\text{ref}}} \frac{\tilde{v}^*}{y_{\text{wall}}^{*2}} \quad (3.4.11)$$

3.5 Procuring Convergence

In regions of high velocity gradients, two-equation models are known to sometimes produce turbulence at such an unphysically high rate that the solution diverges. Yoder et al. (1999) have observed this problem with Chien's k - ϵ model in flows as simple as a flat plate boundary layer. They found it necessary to limit the maximum values of k , ϵ and μ_t .

as well as restrict the amount by which these quantities are updated during each iteration. Bardina et al. (1997) utilised similar limiters in their simulations with two-equation models. For the three-dimensional flow simulations presented in Chapters 5 and 6, imposing an upper limit on μ_t as well as on the ratio production-to- destruction ratio of k (ρ_k/β_ρ) was found to be effective in promoting monotonic convergence.

In addition to the above-mentioned upper limits, it was also necessary to set lower limits for k , ε , ω and $\tilde{\nu}$ to prevent them from assuming negative values during pseudod-time stepping to a steady state.

3.6 Turbulence Boundary Conditions

At wall boundaries, k , $\tilde{\varepsilon}$, $d\varepsilon/dn$ and $\tilde{\nu}$ are set to zero. In the *SST* turbulence model the wall value of ω is determined using:

$$\omega_{wall}^* = \frac{\beta_{wall}}{Re_{ref}^2} \frac{6\mu^*}{0.075\rho^* y_1^{*2}} \quad (3.6.1)$$

A value of $\beta_{wall}=10$ was suggested by Menter (1994), whereas Hellsten (1998a,b) suggested 1.25. Sensitivity of the simulation results to the value of β_{wall} will be discussed in Section 4.1.3.

In the k - ω turbulence model the wall value of ω is obtained from:

$$\omega_{wall}^* = \frac{U_\tau^{*2} S_R}{\nu^*} \quad (3.6.2)$$

where,

$$S_R = \begin{cases} (50 / k_R^*)^2 & k_R^* \leq 25 \\ 100 / k_R^* & k_R^* > 25 \end{cases} \quad (3.6.3)$$

The parameter k_R^* was calculated as per the recommendation of Hellsten (1998a,b) for smooth walls:

$$k_R^* = 2.4 y_1^{+0.85} \quad (3.6.4)$$

At inflow boundaries all turbulence properties are represented through polynomials which may incorporate multiple patches over a given boundary to allow accurate representation of complicated distributions. Finally, at outflow boundaries the turbulence quantities are extrapolated from within the computational domain.

4.0 Verification of Turbulence Models

In this chapter, results from two verification tests are presented for the five turbulence models described in Chapter 3. The first case is an incompressible, zero-pressure-gradient boundary layer developing on a flat plate. Comparisons are made with the experimental data of Wiegardt and Tillmann (1969) as well as the data compiled by Patel et al. (1985). Sensitivities of the turbulence models to lower limits imposed on turbulence quantities, spatial resolution, as well as other simulation parameters are established in this simple flow to provide guidelines for subsequent simulations involving more complex flows. The second test case is an axisymmetric separating flow measured by Driver and Johnston (Case C.S0, 1990). This benchmark was deemed a useful precursor to the M2129 diffusing S-duct simulation since adverse streamwise pressure gradient and the presence of separation are key features of both flows. Additionally, guidelines determined from the flat plate study with respect to various simulation parameters are verified in the C.S0 flow of Driver and Johnston.

4.1 Simulations of a Flat Plate Boundary Layer

Simulations of a flat plate boundary layer were conducted in order to investigate the effects of lower limits imposed on turbulence properties, artificial compressibility (i.e., preconditioning for low-Mach-number conditions), spatial resolution, artificial dissipation, and the nature of inflow boundary conditions on convergence characteristics and prediction accuracy. The plate maximum Reynolds number based on length (L) was

4.5×10^6 and the freestream velocity was 33 m/s ($=V_{ref}$). The simulations were conducted with a computational domain of 15cm height and 2.1m length. A 10cm long “inviscid wall” boundary was placed upstream of the plate so as to include the plate leading edge into the simulations. Both the upper and aft boundaries of the computational domain were set as outflow boundaries with a fixed, spatially uniform static pressure. All flow variables other than pressure were extrapolated to the outflow boundaries from within the computational domain. At the inflow boundary uniform distributions of velocity (33m/s), flow direction and turbulence properties were imposed and static pressure was extrapolated from the interior nodes as part of the solution. Unless stated otherwise in the subsequent sections, the inflow turbulence properties were as follows:

Turbulence Model	Turbulence quantity
Rodi's k - ε model	$k = 1.3 \times 10^{-3}$ $\varepsilon = 7$
Chien's k - ε model	$k = 1.3 \times 10^{-3}$ $\varepsilon = 7$
k - ω model	$k = 1.8 \times 10^{-6}$ $\omega = 100$
Menter's SST model	$k = 1.8 \times 10^{-6}$ $\omega = 100$
SA model	$\chi = 0.77$

Table 4.1.1 Values of turbulence parameters specified at the inflow boundary

The initial velocity distribution throughout the computational domain was set to a uniform value of 1 m/s parallel to the plate surface. The static pressure was set to a uniform initial value matching the outflow pressure specifications. Although the FS code allows

specification of nonuniform initial distributions of flow variables within the computational domain, this option was not needed for the flat plate or for any of the remaining test cases considered in this study. Finally, the density, molecular viscosity were fixed at 1.2 kg/m^3 and $1.8 \times 10^{-5} \text{ Pa}\cdot\text{s}$, respectively.

Only limited results will be provided for flat-plate simulations with Chien's k - ϵ model since convergence difficulties were encountered with this turbulence model in most instances. More specific information on this problem is included in the next section.

4.1.1 Sensitivity to Freestream Turbulence Parameters

This section describes the results of a parametric study establishing sensitivity of the predicted boundary layer development to the freestream values of the turbulence parameters $(k, \epsilon, \tilde{\epsilon}, \omega, \tilde{\nu})$. For this purpose, the turbulence specifications at the inflow boundary of the computational domain were varied systematically. In order to draw accurate conclusions in such an exercise, it is important that the freestream values of the turbulence parameters remain constant along the length of the plate. During the course of pseudo-time marching to a "steady state" solution, it is plausible for the turbulence parameters to temporarily assume negative values. This nonphysical behaviour would tend to cause rapid divergence of the solution. In *FS*, this problem is avoided by imposing lower limits on the turbulence parameters that are being solved. Matching these lower limits to the turbulence specifications at the inflow boundary prevented decay of

freestream turbulence with downstream distance, hence provided the desired homogeneous distribution. Thus, in the discussions that follow, changes in the freestream turbulence settings are indicated by changes in these lower limits.

A somewhat conservative grid consisting of 273 nodes in the streamwise (x) direction, 129 nodes in the cross-stream (y) direction, and $y_1^+ = 1.25$ at $x/L = 0.6$ was used. Details of this grid are given in Appendix A.3. In what follows, the predicted skin friction coefficient and the velocity profile at $x/L = 0.6$ will be compared with the data of Wiegardt and Tillmann (1969), whereas the data compiled by Patel et al. (1985) will be used as a reference for the predicted k profile at the same streamwise position. The band of variation in the k data of Patel et al. is quite large as seen in Figure 4.1.1.1. Thus, comparison of predicted values with this turbulence data will be of qualitative nature. The choice of $x/L = 0.6$ as the streamwise location for analysis of the results was arbitrary since the trends at this location are similar to those observed at other streamwise positions.

Figure 4.1.1.1 Experimental flat plate k data, reproduced from Patel et al. (1985)

Results of Rodi's k - ε model for several freestream ($=\varepsilon_{min}$) turbulence dissipation rates are given in Figures 4.1.1.2a-c. Key simulation parameters used in these cases are listed in Table 4.1.1.1. The $k_{min}=0.0013$ value was chosen based on an assumption of 0.1% freestream turbulence intensity. The velocity profile as well as the streamwise distribution of C_f are predicted quite accurately, while the peak in the experimental k profile is not captured at all. These results would imply that the inaccuracy in the k distribution is offset by underestimated ε values, thereby yielding μ_t values which translate into the correct distribution for the Reynolds shear stress. Sensitivity of k , velocity and C_f distributions to ε_{min} variations is noted to be negligible. Predicted velocity profiles show slight sensitivity at the boundary layer edge when ε_{min} is increased to 100. Based on these results, values for ε_{min} on the order of 10, which yield $(\mu_{t,min}/\mu)$ ratios on the order of 0.001 are recommended. This is in agreement with suggested values in the published literature (e.g. Menter, 1994 and Bardina et al., 1997).

Another test was run with $k_{min}=1.8 \times 10^{-5}$ while maintaining $\mu_{t,min}/\mu=0.001$. Convergence was not stable in this case because this combination of k_{min} and $\mu_{t,min}$ yields freestream dissipation rates in the range where roundoff errors seem to become significant. Another noteworthy observation was the dependence of convergence rate on the initial value of k . It was found that low $k_{initial}$ values resulted in convergence towards laminar flow before arriving at the turbulent solution, which increases the required CPU time. Use of $k_{initial}=0.11$, corresponding to $\mu_{t,initial}/\mu=10$, avoided this transient pattern.

Unrealistically high turbulence production rates were encountered at the plate leading edge with Chien's k - ϵ model, which caused the solution to diverge. As will be noted in Section 3.5, this problem can be remedied by enforcing a limit on the production-to-destruction ratio for k and/or μ_t . However, for the purposes of this benchmark, it was deemed sufficient to conduct simulations based on Chien's k - ϵ model with the inflow boundary of the computational domain placed downstream of the leading edge, the results of which are presented in Section 4.1.4.

C_{AD}	CFL	K_d	κ_s	k_{min}	$k_{initial}$	$\varepsilon_{initial}$	$\mu_{t,max}/\mu$	$\rho_H/\rho_{L,max}$
$\max[1.0; 1.4V_{local}]$	1.8	0.5	0.01	0.0013	0.11	7	-	-

Table 4.1.1.1 Simulation parameters for the results shown in Figure 4.1.1.2a-c

Figure 4.1.1.2 Flat-plate prediction based on Rodi's k - ε model - sensitivity to ε_{min}

a) C_f distribution, b) velocity profile at $(x/L=0.6)$,
c) k profile at $(x/L=0.6)$

Parametric studies were performed for the k - ω and Menter's SST turbulence models with $\mu_{t,min}$, ω_{min} and k_{min} systematically adjusted such that the effects of each of these parameters were established in isolation. The conclusion was that within the ranges considered ($\omega_{min} = 100$ to 5000 ; $k_{min} = 1.8 \times 10^{-6}$ to 1.8×10^{-3} and $\mu_{t,min}/\mu = 2 \times 10^{-4}$ to 1×10^{-1}), the only noticeable sensitivity was with respect to the value of ω_{min} . What follows is a selection of the results of these parametric studies that highlights this sensitivity.

Selective results for the k - ω model are shown in Figure 4.1.1.3a-c. The predicted skin friction distribution is noted to shift to lower values as ω_{min} is increased. The velocity and k profiles are noted to be influenced by ω_{min} as well. Based on these levels of sensitivity, the extent to which the predictions agree with the experimental data is fairly consistent with the results of Rodi's k - ϵ model. Very close to the leading edge of the plate, the prediction accuracy for C_f is less than with Rodi's k - ϵ model; further downstream the accuracies are comparable when the right freestream ω value is used. The k - ω model is noted to be even less effective than Rodi's k - ϵ model in following the peaky k profile; it is also less accurate with respect to k below y^+ of 10, regardless of the freestream ω value. Between the two values of ω_{min} tested, the smaller one is clearly preferable in providing more accurate C_f information. In addition, the discontinuity in velocity derivative near the boundary layer edge, a well-known deficiency of the k - ω model, is not evident at this ω_{min} value. Finally, a tendency of the solution to converge to a laminar flow was observed with ω_{min} values not much higher than 1000. Based on all these results, a freestream value of 100 seems to be the right choice for ω within the context of the k - ω turbulence model.

C_{AD}	CFL	K_d	κ_d	$k_{initial}$	$\omega_{initial}$	$\mu_{t, max}/\mu$	$\rho_\ell/\rho_{t, max}$
$\max[1.0; 1.4V_{local}]$	1.8	0.5	0.01	$=k_{min}$	$=\omega_{min}$	-	-

Table 4.1.1.2 Simulation parameters for the results shown in Figure 4.1.1.3a-c

Figure 4.1.1.3 Flat-plate prediction based on k - ω model - sensitivity to ω_{min}
a) C_f -distribution, b) velocity profile at $(x/L=0.6)$,
c) k profile at $(x/L=0.6)$

The results for Menter's *SST* turbulence model are given in Figures 4.1.1.4a-c. In contrast with the *k- ω* model, the *SST* predictions for velocity and C_f distributions are noted to be independent of freestream ω in the 100 to 1000 range. The velocity profile is captured slightly more accurately than with either Rodi's *k- ϵ* or the *k- ω* models. C_f distribution is predicted almost as accurately as with Rodi's *k- ϵ* model, except for a slight underestimation in close vicinity of the leading edge. For a freestream ω value of 5000, relaminarization of the boundary layer is noted beyond $Re_x=10^6$.

Finally, sensitivity of the predictions with the *SA* model to freestream eddy viscosity is shown in Figures 4.1.1.5a-c. Variation of $\chi_{min} = \tilde{\nu}_{min}/\nu$ from 0.24 to 1.4 was considered which corresponds to a $\mu_{t,min}/\mu$ range from 1×10^{-5} to 1×10^{-2} . The changes in the predicted velocity and C_f distributions over this range of freestream turbulence variations are noted to be negligible. The velocity profile at ($x/L=0.6$) is captured as accurately as with Rodi's *k- ϵ* model. The C_f predictions are also as accurate over the rear portion of the plate, whereas near the leading edge the values fall somewhat short of the experimental data. All models except Rodi's *k- ϵ* are noted to underestimate C_f close to the leading edge while the prediction accuracy seems to be independent of the turbulence model further downstream. This trend may be tied to the fact that with the type of grid used for these simulations, in which clustering of the nodes towards the wall is identical at all streamwise positions, y^+ values are higher while relatively fewer nodes exist in the boundary layer near the leading edge (refer to Appendix A.3 for details on the grid). As will be shown later, each of the turbulence models responds differently to the spatial resolution of the boundary layer.

C_{AD}	CFL	K_d	κ_d	k_{min}	$k_{initial}$	$\omega_{initial}$	$\mu_{t, max}/\mu$	$\rho/\beta_{t, max}$
$\max[1.0; 1.4V_{local}]$	1.8	0.5	0.01	$1.8e-05$	$=k_{min}$	$=\omega_{min}$	-	-

Table 4.1.1.3 Simulation parameters for the results shown in Figure 4.1.1.4a-c

Figure 4.1.1.4 Flat-plate prediction based on Menter's SST model - sensitivity to ω_{min}
a) C_f -distribution, b) velocity profile at ($x/L=0.6$), c) k profile at ($x/L=0.6$)

C_{AD}	CFL	K_d	κ_d	χ_{min}	$\chi_{initial}$	$\mu_{t\ max}/\mu$
$\max[1.0; 1.4V_{local}]$	1.8	0.5	0.01	0.77	0.77	-

Table 4.1.1.4 Simulation parameters for the results shown in Figure 4.1.1.5a-c

Figure 4.1.1.5 Flat-plate prediction based on the *S4* model - sensitivity to $\chi_{min} = \tilde{v}_{min}/\nu$
a) C_f -distribution, b) velocity profile at $(x/L=0.6)$, c) χ profile at $(x/L=0.6)$

4.1.2 Sensitivity to Artificial Compressibility

In order to establish optimum setting for the preconditioning of the governing equations, parametric studies on the artificial compressibility factor, C_{AD} , were performed for each of the turbulence models. Non-optimum C_{AD} values were identified as those which resulted in either solution instability or significant reduction in convergence rate. Instability was found to occur when C_{AD} was too low, with the solution persistently oscillating with iterations in pseudo-time. Based on analysis of the pseudo-transient flow development, this oscillatory behaviour appears to be related to pressure differences being communicated across the computational domain out of phase with the velocity field. High C_{AD} values, on the other hand, yielded lower convergence rates. It may be argued that this is a result of slower convection of pressure transients across the computational domain, since the C_{AD} parameter can be viewed as an artificial wave speed, replacing the standard speed of sound when the governing equations are solved in true compressible form without any preconditioning. An optimum C_{AD} of $\max[1.0; 1.4V_{local}]$ was observed for all four models. Acceptable levels of convergence were achieved with values of C_{AD} ranging from $\max[1.0; 1.4V_{local}]$ to $\max[1.0; 2.2V_{local}]$.

It should be noted that, for the $y_l^+ = 1.25$ value used in these simulations, the minimum flow velocity at all of the nodes off the surface of the plate remained above 2m/s. Hence, the lower C_{AD} threshold of 1 m/s was not invoked anywhere in the computational domain.

4.1.3 Sensitivity to Spatial Resolution, Artificial Dissipation and Wall Boundary Conditions

A total of five grids were tested to establish sensitivity of the predictions to the location of the first node off the plate surface and to the number of nodes within the boundary layer. The first grid consisted of 273 nodes in the streamwise direction and 129 nodes in the cross-stream, with 82 nodes within the maximum boundary layer thickness of 4cm at the trailing edge of the plate. The node count within the boundary layer was determined on the basis of a boundary layer edge location identified with an edge velocity that is 99% of the freestream value. The distance of the first node from the plate surface corresponded to $y_1^+ = 1.25$ at $x/L = 0.6$. This was the finest grid tested and was chosen to suit the y_1^+ as well as boundary layer resolution requirements of all four models based on the information available in the literature (e.g. Bardina et al., 1997; Heidegger et al., 1999; Hellsten, 1998a). Subsequent grids were reduced in resolution, and the distance of the first node from the wall was varied in an attempt to identify the coarsest grid with the largest y_1^+ that provided C_f values within 5% of the experimental data. The 5% threshold was chosen as an acceptable level of accuracy for C_f in design-cycle analysis. The streamwise number of nodes in these coarser grids was 97. Cross-stream resolution was reduced to 45 and 23 nodes, yielding 31 and 15 nodes, respectively, within the boundary layer thickness at the trailing edge of the plate. The distance of the first node from the wall was varied with these two coarse grid resolutions. Values of approximately $y_1^+ = 3$ and $y_1^+ = 5$ were specified at $x/L = 0.6$. The basic information for each grid is summarised in Table 4.1.3.1 below and details of each grid are given in Appendix A.3.

Grid	$N_i \times N_j$	$y_1^+(x/L=0.6)$	Max. number of nodes in <i>BL</i>
A	273×129	1.25	82
B	97×45	2.8	31
C	97×23	5.0	36
D	97×45	5.0	15
E	97×23	3.0	19

Table 4.1.3.1 Grids tested in flat plate simulations

Results of the tests with these grids are displayed in Figures 4.1.3.1 to 4.1.3.4. For the $k-\omega$ and *SST* turbulence models the results are shown for two of the grids only since the discrepancy between the predicted and experimental C_f data was well beyond the 5% threshold with the remaining grids. In the plots, symbols are included to correspond with node locations on the prediction curves so as to provide a visual impression of the boundary layer resolution in the context of the simulation results. Rodi's $k-\epsilon$ model displays the least amount of dependence on boundary-layer resolution and y_1^+ , and is closely followed by the *SA* model. Both models produce reasonably accurate results for the C_f and velocity distributions on the coarsest grid considered ($y_1^+=5$ at $x/L=0.6$; 15 nodes in the *BL* at $x/L=1.0$). Menter's *SST* model and the $k-\omega$ model, on the other hand, already yield prediction errors on the second finest grid ($y_1^+=2.8$ at $x/L=0.6$; 31 nodes in the *BL* at $x/L=1.0$) that are somewhat greater than those with the $k-\epsilon$ and *SA* models on the coarsest grid. These results were used as guidelines in the construction of the grids in near-wall regions of the remaining test cases in this study.

In the flat-plate simulations the fourth-order artificial dissipation term was dominant since the pressure gradients, which were only present at the leading edge, were not steep enough to yield noticeable magnitudes for the second-order term. The effect of this artificial dissipation on the prediction accuracy was evaluated through simulations with two different values of the scaling coefficient κ_4 , namely 0.005 and 0.01. Differences were apparent only on the coarser grids shown in Figures 4.1.3.5a-c, whereas the results for the finer grids essentially overlapped. This dependence of artificial dissipation on grid resolution is expected since node spacings dictate the distances over which the 4th order difference appearing in the artificial dissipation term is evaluated. This conclusion drawn in the context of the $k-\omega$ model was confirmed to be true for the remaining models as well; for brevity those results are not included herein. The choices of κ_4 for the previously discussed freestream turbulence and grid sensitivity analyses were such that the trends observed in these contexts were not influenced by the amount of artificial dissipation.

In Section 3.6.1 the importance of the β_{wall} parameter in the calculation of ω_{wall} for the *SST* model was indicated. Hellsten (1998b) noted that an increase of β_{wall} above 1.25 tends to amplify the dependence of the *SST* model predictions on grid resolution. Variation of this parameter was studied and the results are displayed in Figures 4.1.3.6a-c. It is evident that prediction accuracy is very sensitive to changes in β_{wall} and that a value of 1.25 does indeed provide the best result.

C_{AD}	CFL	K_d	κ_4	k_{min}	ε_{min}	$k_{initial}$	$\varepsilon_{initial}$	$\mu_{t\ max}/\mu$	$\rho_{t\ max}/\rho$
$\max[1.0; 1.4V_{local}]$	1.8	0.5	0.005	0.0013	7	0.11	$=\varepsilon_{min}$	-	-

Table 4.1.3.2 Simulation parameters for the results shown in Figure 4.1.3.1a-c

Figure 4.1.3.1 Flat-plate prediction based on Rodi's k - ε model - sensitivity to grid resolution a) C_f distribution, b) velocity profile at ($x/L=0.6$), c) k profile at ($x/L=0.6$)

C_{AD}	CFL	K_d	κ_s	k_{min}	ω_{min}	$k_{initial}$	$\omega_{initial}$	$\mu_{t, max}/\mu$	$\rho_k/\rho_{t, max}$
$\max[1.0; 1.4V_{local}]$	1.8	0.5	0.005	$1.8e-6$	100	$=k_{min}$	$=\omega_{min}$	-	-

Table 4.1.3.3 Simulation parameters for the results shown in Figure 4.1.3.2a-c

Figure 4.1.3.2 Flat-plate prediction based on the $k-\omega$ model - sensitivity to grid resolution a) C_f distribution, b) velocity profile at $(x/L=0.6)$, c) k profile at $(x/L=0.6)$

C_{AD}	CFL	K_d	κ_d	k_{min}	ω_{min}	$k_{initial}$	$\omega_{initial}$	μ_{∞}/μ	$\rho_k/\rho_{k,max}$
$\max[1.0; 1.4V_{local}]$	1.8	0.5	0.005	$1.8e-6$	100	$=k_{min}$	$=\omega_{min}$	-	-

Table 4.1.3.4 Simulation parameters for the results shown in Figure 4.1.3.a-c

Figure 4.1.3.3 Flat-plate prediction based on Menter's SST model - sensitivity to grid resolution
a) C_f distribution, b) velocity profile at ($x/L=0.6$) c) k profile at ($x/L=0.6$)

C_{AD}	CFL	K_d	κ_4	χ_{min}	$\chi_{initial}$	$\mu_{t, max}/\mu$
$\max[1.0; 1.4V_{local}]$	1.8	0.5	0.005	0.77	$=\chi_{min}$	-

Table 4.1.3.5 Simulation parameters for the results shown in Figure 4.1.3.4a-c

Figure 4.1.3.4 Flat-plate prediction based on the SA model - sensitivity to grid resolution

a) C_f distribution, b) velocity profile at $(x/L=0.6)$,
c) χ profile at $(x/L=0.6)$

C_{AD}	CFL	K_d	κ_4	k_{min}	ω_{min}	$k_{initial}$	$\omega_{initial}$	$\mu_{t, max}/\mu$	$\rho_k/\beta_{k, max}$
$\max[1.0; 1.4V_{local}]$	1.8	0.5	0.005	$1.8e-6$	100	$=k_{min}$	$=\omega_{min}$	-	-

Table 4.1.3.6 Simulation parameters for the results shown in Figure 4.1.3.5a-c

Figure 4.1.3.5 Flat-plate prediction based on the $k-\omega$ model - sensitivity to κ_4

(97×45 nodes, $y_1^+ = 3$)

a) C_f distribution, b) velocity profile at ($x/L=0.6$), c) k profile at ($x/L=0.6$)

C_{AD}	CFL	K_d	κ_4	k_{min}	ω_{min}	$k_{initial}$	$\omega_{initial}$	$\mu_{t, max}/\mu$	$\rho_k/\delta_{t, max}$
$\max[1.0; 1.4V_{local}]$	1.8	0.5	0.01	$1.8e-5$	1000	$=k_{min}$	$=\omega_{min}$	-	-

Table 4.1.3.7 Simulation parameters for the results shown in Figure 4.1.3.6a-c

Figure 4.1.3.6 Flat-plate prediction based on Menter's SST model - sensitivity to β_{wall} (273×129 nodes, $y_1^+=1.25$)
a) C_f distribution, b) velocity profile at $(x/L=0.6)$, c) k profile at $(x/L=0.6)$

4.1.4 Flat-Plate Boundary Layer with Nonuniform Inflow Boundary Conditions

In the final phase of the flat plate study, the inflow boundary was placed downstream of the leading edge, requiring specification of nonuniform profiles of inflow properties. This test was conducted to provide guidance for the subsequent more elaborate simulations, where the desire to minimize the grid-node count dictates the smallest possible computational domain size, which often requires the use of such nonuniform inflow boundary conditions. Profiles of flow variables (e.g., velocity, flow direction, turbulence properties) over boundaries are specified in the *FS* code through polynomial curves / surfaces (up to 10th order in 2-D and 5th order in 3-D). The distributions of flow properties in the boundary-layer portion of the inflow boundary ($Re_\theta=2760$) were obtained from the previous simulations where the boundary layer was predicted over the full length of the plate. Outside the boundary layer, the inflow boundary conditions were the same as in the previous tests. It was found that accurate polynomial fitting to the resultant distribution of velocity and turbulence properties required the inflow boundary to be subdivided into as many as 7 smaller patches.

Figures 4.1.4.1 to 4.1.4.5 display the test results. As shown in Tables 4.1.4.2 to 4.1.4.6, a higher minimum artificial compressibility factor of $C_{AD\ min}=15$ was required for stable convergence with the new inflow boundary condition than the previous value of $C_{AD\ min}=1.0$.

In all cases an overprediction of C_f is noted in the vicinity of the inflow boundary. This resulted from an error of less than 0.5% of the freestream velocity at the first node off the wall, that was caused by the polynomial fitting process. However, the flow recovers from this problem over a relatively short streamwise distance, resulting in velocity and turbulent kinetic energy profiles at $x/L=0.6$ that are almost identical to the simulations that included the leading edge. These simulations highlight the importance of specifying inflow boundary conditions very accurately.

Convergence problems with Chien's k - ϵ model, caused by unrealistic amounts of production of turbulence energy at the leading edge, was noted in Section 4.1.1. Figures 4.1.4.2a-c show the results of this model with the new computational domain configuration that excludes the leading edge. It is noted that, while the prediction accuracy is comparable to those of other models, a lower viscous-time-step limit (controlled by K_d) was required. Interestingly, this model is the only one that predicted the expected peak in k near the wall, albeit only qualitatively.

C_{AD}	CFL	K_d	κ_d	k_{min}	ε_{min}	$k_{initial}$	$\varepsilon_{initial}$	$\mu_{t, max}/\mu$	$\rho_\mu/\rho_{\mu, max}$
$\max[15; 1.4V_{local}]$	1.8	0.5	0.005	0.0013	7	0.11	$=\varepsilon_{min}$	-	-

Table 4.1.4.1 Simulation parameters for the results shown with solid lines in Figures 4.1.4.1a-c

Figure 4.1.4.1 Flat-plate prediction based on Rodi's k - ε model - sensitivity to inflow boundary location (dashed lines correspond to earlier simulations that include the leading-edge region)

a) C_f distribution, b) velocity profile at $(x/L=0.6)$, c) k profile at $(x/L=0.6)$

C_{AD}	CFL	K_d	κ_4	k_{min}	ε_{min}	$k_{initial}$	$\varepsilon_{initial}$	$\mu_{t, max}/\mu$	$\rho_w/\rho_{t, max}$
$\max[15; 1.4V_{local}]$	1.8	0.5	0.005	0.0013	7	0.11	$=\varepsilon_{min}$	-	-

Table 4.1.4.2 Simulation parameters for the results shown Figures 4.1.4.2a-c

Figure 4.1.4.2 Flat-plate prediction based on Chien's k - ε model - sensitivity to inflow boundary location

a) C_f distribution, b) velocity profile at ($x/L=0.6$),

c) k profile at ($x/L=0.6$)

C_{AD}	CFL	K_d	κ_4	k_{min}	ω_{min}	$k_{initial}$	$\omega_{initial}$	$\mu_t max/\mu$	$\rho_k/\rho_{t max}$
$\max[15; 1.4V_{local}]$	1.8	0.5	0.005	$1.8e-6$	100	$=k_{min}$	$=\omega_{min}$	-	-

Table 4.1.4.3 Simulation parameters for the results shown with solid lines in Figure 4.1.4.3a-c

Figure 4.1.4.3 Flat-plate prediction based on the k - ω model - sensitivity to inflow boundary location (dashed lines correspond to earlier simulations that include the leading-edge region) a) C_f distribution, b) velocity profile at $(x/L=0.6)$, c) k profile at $(x/L=0.6)$

C_{AD}	CFL	K_d	κ_4	k_{min}	ω_{min}	$k_{initial}$	$\omega_{initial}$	$\mu_{t, max}/\mu$	$\rho_k/\rho_{k, max}$
$\max[15; 1.4V_{local}]$	1.8	0.5	0.005	$1.8e-6$	100	$=k_{min}$	$=\omega_{min}$	-	-

Table 4.1.4.4 Simulation parameters for the results shown with solid lines in Figure 4.1.4.4a-c

Figure 4.1.4.4 Flat-plate prediction based on the Menter's SST model - sensitivity to inflow boundary location (dashed lines correspond to earlier simulations that include the leading-edge region)
a) C_f distribution, b) velocity profile at $(x/L=0.6)$,
c) k profile at $(x/L=0.6)$

C_{AD}	CFL	K_d	κ_d	χ_{min}	$\chi_{initial}$	$\mu_{i, max}/\mu$
$\max[15; 1.4V_{local}]$	1.8	0.5	0.005	0.77	$=\chi_{min}$	-

Table 4.1.4.5 Simulation parameters for the results shown with solid lines in Figure 4.1.4.5a-c

Figure 4.1.4.5 Flat-plate prediction based on the $S4$ model - sensitivity to inflow boundary location (dashed lines correspond to earlier simulations that include the leading-edge region)
a) C_f distribution, b) velocity profile at $(x/L=0.6)$,
c) χ profile at $(x/L=0.6)$

4.2 Simulations of Separating Flow in an Annular Diffuser

The axisymmetric, separating, adverse-pressure-gradient flow of Driver and Johnston (Case C.S0, 1990) constituted the second test case of the present study. In this experiment Driver and Johnston used an annular diffuser which was formed by aligning a cylinder longitudinally in a wind-tunnel test section with diverging walls. Boundary layer suction was applied at the test section walls such that separation occurred on the cylinder surface only. Prediction of the flow in this test case is particularly challenging since the separation bubble is not constrained in the axial direction. The size of the computational domain was minimised by prescribing a boundary at the larger radius that followed a stream-surface rather than extending the domain to the test-section walls. The shape of this stream-tube was defined on the basis of mass conservation. Figure 4.2.1 gives a cross-sectional view of the resulting axisymmetric computational domain.

Figure 4.2.1 Computational domain used for Driver and Johnston's C.S0 test case

The domain was selected to be 1.26m long. The distance $(R_s - R_0)$ between the surface of the inner cylinder and the outer slip-boundary ranged from a minimum of 37.1mm at the inlet to a maximum of 71.1mm in the region of separation. The *FS* code was configured to solve the governing equations in a form suitable for axisymmetric flows, thus allowing

the use of a “2-D” grid. Grid dependence tests revealed the minimum y_1^+ and node-count requirements as listed in Table 4.2.1 for each model. These results are noted to be quite consistent with the observations in the flat-plate simulations.

Turbulence Model	Minimum number of nodes in the boundary layer.	y_1^+
<i>SA</i> model	15	5
Rodi's $k-\epsilon$ model	15	5
$k-\omega$ model	15	1.3
Menter's <i>SST</i> model	15	1.3

Table 4.2.1 Minimum turbulence model requirements based on simulations of the flat-plate and Driver & Johnston's C.S0 flows

For consistency, however, comparison of the predictions using different turbulence models with experimental data was based on a single grid that satisfies the minimum requirements of the most stringent turbulence model. A conservative grid was thus designed consisting of 89 and 61 nodes in the streamwise and radial directions, respectively, with $y_1^+ = 1.28$ and about 40 nodes in the boundary layer at the inflow boundary. The variation of cross-stream clustering of the nodes with streamwise distance was chosen to ensure similar resolution of the boundary layer elsewhere. Detailed information on this grid is provided in Appendix A.3.

Profiles of velocity and turbulence quantities were specified at the inflow boundary. The experimental data corresponding to the location of the inflow boundary of the computational domain ($x=-0.4572\text{m}$) were found to agree well with the profiles of a simulated flat-plate boundary layer with the same Re_θ of 2760. Since the experimental profiles were relatively sparse, and not all of the turbulence quantities were part of the measured data set, the simulated flat-plate boundary layer data were used to specify the inflow boundary conditions.

The freestream velocity ($=V_{ref}$) at the inflow boundary was set to 30 m/s as per the experimental conditions. The values used for the turbulence quantities in the freestream portion of the inflow boundary are listed in Table 4.2.2.

Turbulence Model	Turbulence quantity
Rodi's k - ε model	$k = 1.3 \times 10^{-3}$ $\varepsilon = 7$
Chien's k - ε model	$k = 1.3 \times 10^{-3}$ $\varepsilon = 7$
k - ω model	$k = 1.8 \times 10^{-6}$ $\omega = 100$
Menter's SST model	$k = 1.8 \times 10^{-6}$ $\omega = 100$
S A model	$\chi=0.77$

Table 4.2.2 Inflow boundary freestream turbulence settings

At the “slip” boundary following a streamline, all flow properties were extrapolated from within the computational domain. At the outflow boundary the static pressure was set to a uniform value and all other flow properties were extrapolated from internal nodes

adjacent to the outflow boundary. The initial velocity distribution was set to a uniform value of 1 m/s pointing in the x-direction (diffuser axis) and the initial pressure distribution was set to the value specified at the outflow boundary. The density and molecular viscosity were fixed at 1.2 kg/m³ and 1.8×10⁻⁵ Pa·s respectively.

It is well known that fixing of static pressure at an outflow boundary results in reflection of transient pressure waves that evolve in the computational domain during pseudo-time marching, rather than allowing them to leave the domain. The *FS* code provides a less reflective alternative boundary condition, originally proposed by Rudy and Strikwerda (1980):

$$\frac{\partial P}{\partial t} - \rho \alpha_s \frac{\partial V_{local}}{\partial t} + \alpha_p (P - P_{BC}) = 0 \quad (4.2.1)$$

where α_p is a constant (set to 1/8 for the present test case), P_{BC} is the pressure value set at the outflow boundary and α_s , which is normally the speed of sound in a purely compressible flow solver, is evaluated as:

$$\alpha_s = \sqrt{(V_{local})^2 + (C_{AD})^2} \quad (4.2.2)$$

to reflect the preconditioned nature of the governing equations being solved by the present algorithm. The pressure at any of the outflow boundary nodes is obtained from Equation 4.2.1 during iterations. When “steady state” is reached at the end of the pseudo-time marching process, the time-dependent terms in this equation become zero, yielding $P=P_{BC}$ as required.

Although the use of this less reflective outflow pressure boundary in the C.S0 test case substantially reduced the magnitude of the pressure oscillations during pseudo-time stepping, ultimate convergence to a solution was still not possible with two-grid-level multigrid cycles. After a sufficient number of multigrid cycles to provide a good initial estimate of the flow, extensive single grid iterations were needed to complete the solution. These convergence difficulties encountered with this flow were absent in the simpler flat-plate test case, as well as the complex three-dimensional flows to be discussed in subsequent chapters. As was reviewed in Chapter 1, separating flows in diffusing ducts such as the C.S0 case tend to be unsteady. Even if a steady state is achieved in a well-controlled environment, which seems to be the case in Driver and Johnston's experiments, this state may easily be perturbed by small disturbances. The difficult-to-control transient behaviour observed numerically during pseudo-time stepping may be merely a reflection of this nature of the flow.

Finally, Table 4.2.3 displays the simulation parameter settings and Table 4.2.4 summarises the initial and threshold values used for the turbulence quantities. These values were chosen based on the flat plate sensitivity studies reported in Section 4.1. It is noted that C_{AD} and K_d needed some adjustments from the flat plate settings to optimise the rate of convergence of the solution.

C_{AD}	CFL	K_d	κ_d
$\max[10.0; 1.0V_{local}]$	1.8	0.1	0.005

Table 4.2.3 Simulation parameters employed in the C.S0 test case,
(same settings used for all models)

	$k_{initial}$	$\varepsilon_{initial}$	$\omega_{initial}$	$\chi_{initial}$	k_{min}	ε_{min}	ω_{min}	χ_{min}	$\mu_{t,max}/\mu$	ρ_k/δ_k
<i>SA</i>	-	-	-	0.77	-	-	-	0.77	-	-
Rodi's $k-\varepsilon$	0.11	7	-	-	1.3×10^{-3}	7	-	-	-	-
Chien's $k-\varepsilon$	0.11	7	-	-	1.3×10^{-3}	7	-	-	-	-
$k-\omega$	1.8×10^{-6}	-	100	-	1.8×10^{-6}	-	100	-	-	-
<i>SST</i>	1.8×10^{-6}	-	100	-	1.8×10^{-6}	-	100	-	-	-

Table 4.2.4 Turbulence-model initial and threshold setting employed in the C.S0 test case

The simulation results are shown in Figures 4.2.2 to 4.2.8. It is evident from Figure 4.2.2(a) that all models reproduce the correct pressure distributions upstream and downstream of the separation bubble. The *SA* and *SST* models more accurately capture the flat portion of the C_p curve in the separated region. However, all model calculations result in same over-prediction of static pressure in the region of separation. Predictions of C_f are shown in Figure 4.2.2(b). Separation is not predicted by Chien's $k-\varepsilon$ model at all, and no flow reversal is captured by Rodi's $k-\varepsilon$ model. The poor prediction of C_f by Chien's model was found to be related to periodic variations of streamwise velocity along the length of the domain which existed along the first three grid lines next to the wall. It was speculated that this could be caused by insufficient spatial resolution, hence another test was run with $y_1^+ = 0.25$. No change in this trend was observed. Similar C_f trends have been observed by Dudek et al. (1996) and Shih (1998) indicating that this may be a

fundamental deficiency of Chien's $k-\epsilon$ formulation. As presented in Figures 4.2.3 to 4.2.5, Menter's *SST* model is most successful in capturing the trends in the velocity field followed by the *SA* and $k-\omega$ models. Locations of separation and reattachment, summarised in Table 4.2.5, are also most accurately predicted by the *SST* model. Predictions of k profiles, shown in Figures 4.2.6 to 4.2.8, indicate a common trend for all models to underestimate the peak value. This trend is consistent with the results obtained for the flat plate boundary layer.

Location	Experiment	<i>SA</i>	Rodi's $k-\epsilon$	$k-\omega$	<i>SST</i>
Separation (x/R_0)	0.6	-0.5	0.5	0	0.5
Re-attachment (x/R_0)	3.2	4.4	0.5	1.7	3.1

Table 4.2.5 Predicted separation and reattachment points compared with Driver and Johnston's experimental data

Figure 4.2.2 Comparison of predictions with Driver and Johnston's experimental data, a) C_p distribution, b) C_f distribution

Figure 4.2.3 Comparison of predicted velocity profiles with Driver and Johnston's experimental data, a) $x/R_0 = -0.181$, b) $x/R_0 = 0.726$

Figure 4.2.4 Comparison of predicted velocity profiles with Driver and Johnston's experimental data, a) $x/R_0 = 2.177$, b) $x/R_0 = 3.266$

Figure 4.2.5 Comparison of predicted velocity profiles with Driver and Johnston's experimental data ($x/R_0 = 4.354$)

Figure 4.2.6 Comparison of predicted k profiles with Driver and Johnston's experimental data, a) $x/R_0 = -0.181$, b) $x/R_0 = 0.726$

Figure 4.2.7 Comparison of predicted k profiles with Driver and Johnston's experimental data, a) $x/R_0 = 2.177$, b) $x/R_0 = 3.266$

Figure 4.2.8 Comparison of predicted k profiles with Driver and Johnston's experimental data ($x/R_0 = 4.354$)

5.0 Simulations of Vortex-Generator-Jet Flow

Extensive simulations presented in Chapter 4 demonstrated the numerical behaviour and predictive capability of the five turbulence models in question for two-dimensional and axisymmetric flows. These test cases will now be used to provide guidance for the simulation of more complicated three-dimensional flows. This chapter presents simulation results for the flow field of a Vortex-Generator Jet (*VGJ*). The predictions are compared to Findlay's (1998) vectored-jet-in-crossflow experiment. This particular test case was selected due to high-resolution and accurate measurements of velocity and turbulence in the near-field of the jet. Findlay tested several combinations of θ (skew angle) and ϕ (pitch angle). The configuration that is chosen as a test case herein ($\theta=90^\circ$, $\phi=30^\circ$) is not optimum with respect to boundary-layer control. Nonetheless, this configuration was chosen since it allowed the use of a fully orthogonal computational grid, thereby eliminating potential inaccuracies due to grid skewness, and the resultant turbulence levels in the wake of the jet were significantly higher than for the other configurations.

5.1 Description of the Test Case

The goal of this test was to determine the prediction capability of Rodi's $k-\varepsilon$, Wilcox's $k-\omega$, Menter's *SST* two-equation models, and Spalart and Allmaras' one-equation model for *VGJ* flows in the context of aircraft engine intake design optimisation. Chien's $k-\varepsilon$ model was excluded from this study due to its poor numerical behaviour and predictive capability demonstrated in previous tests involving simpler flows. As was discussed extensively in

the literature survey (Section 1.3), the predictive capability of state-of-the-art turbulence models in *VGJ* flows is of interest to S-duct engine intake designers since *VGJ*'s can be used to prevent development of excessive distortion, as well as to manage already-developed distortion that exists due to flow phenomena prevailing at the intake entrance or upstream of the intake. The fundamental assumption of isotropy of turbulence inherent in standard eddy-viscosity-type models cannot be fully justified for flows such as that prevailing in the near-wake of a vortex-generator jet. On the other hand, the requirement of short turn-around cycles in a design optimisation context prevents the use of higher order turbulence models. Thus, it is important to establish whether one- or two-equation eddy-viscosity models provide an acceptable trade-off between accuracy and computational times, which is the question that this study aims to answer.

The streamwise vortices generated by the *VGJ*'s are the key mechanisms in providing control of the intake flow. Thus, in the context of flow control, it is particularly important to accurately predict the cross-stream transport of momentum facilitated by these vortices which, in turn, requires precise prediction of their streamwise trajectory, concentration and diffusion rates. Therefore, the analysis of simulation results presented herein focuses on these aspects of the *VGJ* flow field.

The simulated jet, depicted schematically in Figure 5.1.1, has an exit-plane cross-section of $1D \times 2D$ from which the jet exits at a pitch angle, Φ , of 30° and a skew angle, θ ,

	VR	ϕ	θ	lateral spacing of jets
Findlay (1998)	1.5	30°	90°	3D
Barberopoulos and Garry (1998)	0.8 - 1.4	45°	60°	single jet
Compton and Johnston (1992)	1	45°	90°	single jet
Henry & Pearcey (1994)	0.5 - 2	30°	60°	18D
Johnston and Nishi (1990)	0.8 - 1.0	45°	$\pm 90^\circ$	8D
Khan (1999)	1	30°	60°	single jet
Zhang (1993, 1998)	0.5 - 1.5	45°	45°	5D, single jet
Zhang (1995)	1	45°	60° - 90°	3 - 7 D
Zhang and Rona (1999)	0.5 - 1.5	45°	45°	4 - 11D
Zhang and Collins (1997)	1	30°	60°	single jet

Table 5.1.1 Summary of VGJ parameters for BL control studied by several researchers

The studies listed in this table all focus on boundary layer (BL) control, which, in the context of aircraft engine intakes, relates to prevention of the development of flow distortion. Hence, in a strict sense, the present simulations based on Findlay's test case are limited to this type of VGJ flow control in aircraft engine intake applications. VGJ 's that are optimized for the control of pre-existing distortion in aircraft intakes tend to have significantly higher velocity ratios (VR) such that the resultant streamwise vortex penetrates well beyond the edge of the local boundary layer. A detailed data set for a VGJ optimised for this application does not appear to be available in the published literature. Nonetheless, the fundamental features of such VGJ 's are not expected to differ from those

tailored for *BL* control. Thus, the conclusions drawn in this chapter with respect to grid optimisation and turbulence modelling should apply to both types of *VGJ*'s.

5.1.1 Boundary and Initial Conditions

As shown in Figure 5.1.1, the computational domain was chosen to extend $5D$ upstream of the jet exit and $11D$ in the downstream direction. The first dimension relates to the desire to place the inflow boundary for the crossflow far enough upstream of the jet where the crossflow is still undisturbed. The location of the outflow boundary, on the other hand, was chosen such that the region of flow for which measurements are available is included into the computational domain.

The freestream velocity at the inflow boundary for the crossflow ($x=-5D$) was set to 3.84m/s as per Findlay's experimental conditions. The flow direction was aligned with the x -axis and the turbulence parameters were set to the values given in Table 5.1.1.1.

Turbulence model	Turbulence quantity
Rodi's $k-\epsilon$	$k = 1.3 \times 10^{-3}$ $\epsilon = 7$
$k-\omega$	$k = 1.8 \times 10^{-6}$ $\omega = 100$
Menter's SST	$k = 1.8 \times 10^{-6}$ $\omega = 100$
SA	$\chi=0.77$

Table 5.1.1.1 Inflow boundary freestream turbulence settings

The profiles of velocity and turbulence kinetic energy in the crossflow boundary layer ($x=-5D$; $Re_\delta=445$, $\delta=2D$) were obtained from polynomial curve fits to the experimental data, and the remaining turbulence properties (ε , ω and χ) were obtained from separately-performed flat-plate simulations. The static pressure, which is the only quantity not specified at this inflow boundary, was extrapolated from within the computational domain, as was the case in the previous flows considered in the present study.

At the top outflow boundary located at $z/D=7$, static pressure with a uniform distribution was specified and all other quantities were extrapolated from within the domain. The *FS* code also provides the option of fixing the static pressure at only a single node at an outflow boundary. In this case, the cross-stream pressure gradients in the outflow plane are matched to those in the cross-stream grid-plane adjacent to this outflow boundary. However, such treatment of the top outflow boundary in the *VGJ* case adversely affected the rate of convergence to a solution. Thus, the pressure was uniformly fixed at this boundary, which required it to be placed sufficiently above the jet exit so as to prevent it from interfering with the natural development of vortex roll-up and subsequent diffusion.

Establishing the top boundary too far away from the jet exit goes against the desire to keep the node count of the computational domain low, which is critical to the use of *CFD* in the design cycle of aircraft engine intakes. Thus, as part of the present simulations,

several locations were considered for this boundary to establish sensitivity of the predicted flow field to the size of the computational domain.

Upon setting the pressure at the top outflow boundary, it was found that no pressure specification was required at the aft outflow boundary located at $x/D=11$. Thus, all flow properties were extrapolated to this boundary from within the computational domain.

The lateral (i.e., in the y direction) extent of the computational domain was based on the spacing of the multiple jets in Findlay's experiments ($=3D$). In the simulation, periodic conditions were then specified at the side boundaries of the computational domain. The *FS* code provides options for the quantities that are to be matched at the corresponding nodes of periodic boundaries. For the present simulations, this matching was imposed on the primitive variables (velocity components, pressure and the turbulence quantities) and on the flux vectors E , F , G (refer to Chapter 2 for descriptions of these vectors).

The remaining boundary condition that requires some consideration is the inflow boundary associated with the jet. It is well known that the distributions of velocity and turbulence quantities at this boundary are generally far from being uniform. A number of simulation studies have included the jet pipe leading to the jet-exit plane, and even the plenum feeding the jet pipe as part of the computational domain. Although, there have also been studies that completely ignored the nonuniformity of the flow at the jet-exit plane. Table 5.1.1.2 summarises the range of jet-exit treatments encountered in the published literature.

Reference	Jet exit boundary conditions
Alvarez and Jones (1993)	uniform
Demuren (1993)	included the jet pipe
Findlay (1998)	included the jet pipe
Hassan et al. (1998)	included the jet pipe
Henry and Pearcey (1994)	uniform
Kim and Benson (1993)	included $1D$ of the jet pipe
Tam and Gruber (1999)	nozzle included + uniform
Zhang (1993)	uniform

Table 5.1.1.2 Treatments of jet-exit boundary in published literature

In the context of design optimisation of aircraft-engine intakes utilizing *VGJ*'s, inclusion of the jet pipes, and possibly the plenums feeding these pipes, into the computational domain is undesirable since its impact on computational times would be significant. Thus, in the present study the jet boundary conditions were specified right at the exit plane of the jet. The ultimate objective was to simply impose uniform flow conditions at this inflow boundary, as was done by several other researchers. However, to justify this approach, simulations were also performed with nonuniform flow distribution at the jet-exit plane for comparison.

For the nonuniform jet-exit-plane conditions, polynomial chi-square surface fits (algorithm was adopted from Press et al., 1992) were performed to the velocity magnitude, flow direction, and turbulence kinetic energy data of Findlay (1998).

In the absence of experimental data for ε , this quantity was determined using:

$$\varepsilon = \frac{k^{3/2}}{0.3D_h} \quad (5.1.1.1)$$

where D_h is the hydraulic diameter of the jet exit plane. This approach of evaluating ε has been used by several researchers (e.g., Zhang, 1993; Henry and Pearcey 1994). Various methods for estimating ε in jet flows were evaluated by Nikjoo and Mongia (1998), who found the above-noted relation to perform equally well as more elaborate alternatives. Upon establishing the ε distributions, determining the ω and $\tilde{\nu}$ distributions for use with the k - ω , SST and SA turbulence models is straight forward.

In order to obtain accurate polynomial fits to the distributions of velocity, flow direction and turbulence quantities, it was necessary to subdivide the jet-exit-plane into smaller patches. The subdivision involved a core patch surrounded by four patches along the perimeter of the orifice, with each of these rim patches having a width of $0.16D$. This dimension was based on the location of the data points closest to the edges of the orifice. Typically, polynomial surfaces of the order of 5 were needed in each patch. As shown in Figure 5.1.1.1, the polynomial fits provided quite reasonable representations of the actual distributions. Surface fits for flow direction were of similar quality, and are not shown here since the variation in flow direction was quite small (less than 10 degrees). For simulations with uniform flow conditions at the jet exit plane, average values of velocity magnitude, direction and turbulence kinetic energies were used, and Equation 5.1.1.1 was used to obtain ε and thus ω and $\tilde{\nu}$.

Figure 5.1.1.1 Distribution of V_i and k at the jet exit plane, a) Contours, b) Curves $\circ$ Findlay expt. (1998), — Surface fit

To initialise the simulations, the static pressure in the computational domain was set to a uniform value matching the fixed pressure at the top outflow boundary, and the flow velocity was set to 1 m/s pointing in the x direction. The density and molecular viscosity were fixed at 1.2 kg/m^3 and $1.8 \times 10^{-5} \text{ Pa}\cdot\text{s}$. The initial values and the thresholds of the turbulence quantities were set as indicated in Table 5.1.1.3.

Model	$k_{initial}$	$\varepsilon_{initial}$	$\omega_{initial}$	$\chi_{initial}$	k_{min}	ε_{min}	ω_{min}	χ_{min}	$\mu_{t,max}/\mu$	$(\rho_k/\delta_k)_{max}$
<i>SA</i>	-	-	-	0.8	-	-	-	0.8	-	-
Rodi's k - ε	0.1	7	-	-	1.3×10^{-3}	7	-	-	-	20
k - ω	1.8×10^{-6}	-	100	-	1.8×10^{-6}	-	100	-	-	-
Menter's <i>SST</i>	1.8×10^{-6}	-	100	-	1.8×10^{-6}	-	100	-	-	-

Table 5.1.1.3 Turbulence-model initial and threshold settings

It is noted that for Rodi's k - ε model the production-to-destruction ratio of k (ρ_k/δ_k) had to be limited to 20 to prevent divergence of the solution. The unbounded growth of this ratio tended to take place in the shear layer between the jet and crossflow, upstream of the roll-up process. Finally, the simulation parameters affecting artificial diffusion, artificial compressibility, and local time-step sizes are summarised in Table 5.1.1.4.

C_{AD}	CFL	K_d	κ_d
$\max[3.0; 2.2V_{local}]$	1.8	0.1	0.005

Table 5.1.1.4 Simulation parameters employed for the *VGJ*

5.2 Description of the Flow Physics

Prior to discussions of prediction sensitivity to the choice of grid, boundary conditions and turbulence-models, a typical set of results of these simulations is presented in this section to provide the reader with sufficient background on the physics of the flow. The main features of the *VGJ* flow field predicted with Menter's *SST* model are shown in Figure 5.2.1. The velocity, streamwise vorticity (Ω_x), and turbulent kinetic energy (k) fields, as well as the surface C_f distribution in the near-field of the jet, accompanied by streamlines at the first node off the wall are displayed.

A significant spanwise velocity component, with peak magnitudes comparable to the initial jet velocity, is observed to persist $8D$ downstream of the jet orifice, which is the result of the shallow pitch angle and spanwise orientation of the jet combined with the relatively large jet-to-crossflow velocity ratio (VR). A single dominant streamwise vortex is noted to develop but is observed to be relatively weak to start with, and loses its strength rapidly with downstream distance. Presence of this vortex combined with the small initial pitch angle of the jet are most likely responsible for the collective reorientation of the cross-stream velocity component from the initial 30° towards 0° within about $1.5D$ downstream distance. Before studying the crossflow velocity and streamwise vorticity distributions further, examination of the vector field topology (part d of Figure 5.2.1) would help to shed light on the overall flow development.

Figure 5.2.1 Features of the V/GJ flow field, based on Menter's SST model
 a) Cross-flow velocity vectors (V, W) and U contours,
 b) Q_x contours (negative values indicated by dashed lines),
 c) k contours, d) C_f and streamlines at first node off the wall

This type of figure has already been introduced in Section 1.3.4.3, and the reader is referred to that section for any background information.

The stagnation point in front of the jet is indicated by the saddle of separation $Ss1$. The location of this point is biased toward the negative y direction relative to the centre of the jet orifice due to the large jet skew angle. The same trend was observed by Barberopoulos and Garry (1998) for a jet at 60° skew angle as was shown in Figure 1.3.4.3.7 (b). Negative bifurcation lines ($NBL1,2$), or three-dimensional lines of separation, extend out from $Ss1$ in the negative and positive y directions providing an indication of the extent of the separated (recirculating) region in front of the jet. Due to the close proximity of the adjacent jets in the present test case, both of the NBL 's lead into separation saddles ($Ss2$) at the jet orifice. Thus, instead of wrapping around the jet and augmenting focus nodes $F1$ and $F2$, as is the case for a single jet which was shown in Figure 1.3.4.3.7, the NBL 's are blocked. This would be expected to produce a weaker streamwise vortex behind the orifice and also explains why horseshoe vortices cannot be discerned from the velocity or vorticity plots downstream of the jet orifice. A node of attachment ($Na1$) follows $Ss1$ and feeds two focus nodes of separation ($F1$ and $F2$) at the corner $x/D=0.5$, $y/D=-1.0$ which constitute the starting point of the dominant streamwise vortex. The close proximity of the two focus nodes to each other is due to the high skew angle of the present jet. This pattern has also been observed by Barberopoulos and Garry for a jet at 60° (Figure 1.3.4.3.7b).

Having identified the origin of the dominant streamwise vortex from the surface vector topology, the remaining details of the vortex development can be discerned from parts (a), (b) and (c) of Figure 5.2.1. At plane A_1 the negative vorticity field associated with this vortex is seen to be partially annihilated on the upwash side by positive vorticity convected away from the boundary layer. Further downstream, in planes A_2 , A_3 and A_4 , this partially annihilated region is seen to contribute to the spanwise smearing of the negative vorticity field as it rotates and stretches. This smearing effect is seen to be reflected in the cross-stream distribution of k . These observations are supported by the results of Khan (1999) who noted lower vorticity magnitudes and higher deformation of the vorticity field at a skew angle (θ) of 90° compared to $\theta=60^\circ$ which he deemed to be the optimum value for boundary layer control at $\Phi=30^\circ$ and $VR=1.0$.

5.3 Grid Optimisation Studies

Successive refinement in all three directions of the grid to formally undertake Richardson extrapolation analysis, and thus establish grid independence, would have resulted in unreasonably high *CPU* times for this study. Instead, a series of grids were chosen that separately provided local refinement in carefully selected portions of the computational domain where spatial gradients in the flow field were relatively high.

Figure 5.3.1 Grids employed for the V/GJ computational domain,
a) Grid A ($N_x=47 \times N_y=37 \times N_z=29$), b) Grid B ($59 \times 37 \times 29$),
c) Grid C ($65 \times 37 \times 43$), d) Grid D ($69 \times 61 \times 43$)

As is evident from the grids shown in Figure 5.3.1, emphasis was placed on establishing sufficient resolution in the vorticity field downstream of the jet orifice, in the core region of the jet, and along the perimeter of the jet. In close proximity of the wall surface ($z/D=0.0$), resolution in the z direction was fixed based on the requirement of $y_1^*=1.25$ and a minimum of 15 nodes in the boundary layer, established in the previous benchmarks. In addition to the node count, locations of the top and aft outflow boundaries were also varied to establish sensitivity of the predicted flow to the size of the computational domain. The ultimate objective of such a “node count and domain size optimisation” exercise is to identify the best combination of these parameters that would allow accurate prediction of the *VGJ* flow in the shortest time. As has been mentioned several times throughout the previous chapters, computation time is as important a factor as prediction accuracy in dictating whether or not *CFD* analysis of *VGJ*'s can be used as a tool in the iterative design optimisation of aircraft engine intakes containing *VGJ*'s. It must be mentioned that the current optimisation is specific to the *VGJ* configuration being considered with respect to θ , Φ , VR and lateral jet spacing. Nonetheless, it should be possible to extrapolate from the present results to establish first-order estimates of minimum grid / domain size requirements of alternative *VGJ* configurations.

5.3.1 Classification of Grids

Details of each of the four grids are summarised in Table 5.3.1.1 and comparison with the grids of several other researchers is made in Table 5.3.1.2. Grid A had the fewest number of nodes with $47 \times 37 \times 29$ in the i , j and k directions, respectively. The top and aft outflow boundary locations of $z/D=5$ and $x/D=8$, respectively, were chosen based on inspection of

Grid	N_i	N_j	N_k	Number of nodes in:			Largest grid cell	Top outflow (z/D)	Aft outflow (x/D)
				inflow BL	jet core*	jet shear layer*			
A	47	37	29	20	11	7	$1D$	5	8
B	59	37	29	20	9	5	$1/2D$	5	8
C	65	37	43	20	9	5	$1/2D$	7	11
D	69	61	43	20	15	9	$3/4D$	7	11

Table 5.3.1.1 Grid details for the *VGJ*

* these node counts relate to flow region just above the jet orifice

the flow field with respect to spatial gradients in velocity and pressure. The core of the jet was resolved with 11 nodes in the streamwise direction and a proportional number of nodes in the spanwise direction, and 7 nodes were distributed across the shear layer along the perimeter of the jet orifice. Transition from the high jet resolution to maximum inter-node distances of about $1D$ close to the inflow and aft outflow boundaries was realised with a maximum cell stretch rate of 60%. The resolution of the jet in vicinity of the orifice was coarsened in Grid B and resolution of the relatively coarser portions of the domain was increased such that the maximum inter-node distance was $1/2D$. This grid consisted of $59 \times 37 \times 29$ nodes. The core of the jet contained 9 nodes in the streamwise direction and

5 nodes were located within the shear layer. In Grid C, sensitivity to the locations of the outflow boundaries was tested by relocating these boundaries to $x/D=11$ and $z/D=7$.

Reference	Domain size / D ($x \times y \times z$)	Jet type	VR	Number of nodes ($N_i \times N_j \times N_k$)	Turbulence models
Present study	$13 \times 3 \times 5$	spanwise	1.5	$47 \times 37 \times 29$	Rodi's $k-\epsilon$, $k-\omega$, SST , SA
Alvarez & Jones (1993)	$11 \times 3 \times 6$	normal	2.3	$162 \times 88 \times 44$	$k-\epsilon$, RSM (wall fn.)
Demuren (1993)	$22 \times 6 \times 11$	normal	0.5 & 2	$86 \times 38 \times 38$	RSM (wall fn.)
Findlay (1998)	$45 \times 3 \times 30$	spanwise	0.5 to 1.5	$60 \times 24 \times 40$	$k-\epsilon$ (wall fn.)
Hassan et al. (1998)	$35 \times 3 \times 25$	spanwise	0.5 to 1.5	$34 \times 31 \times 27$	$k-\epsilon$ (wall fn.), BSL , SST
Henry & Pearcey (1994)	$14 \times 7 \times 7$	compound angle	0.5 to 2	$38 \times 35 \times 25$	$k-\epsilon$ (wall fn.)
Kim & Benson (1993)	$17 \times 2 \times 4$	confined normal	2.3	$165 \times 59 \times 83$	$k-\epsilon$, MS
Tam & Gruber (1999)	$17 \times 8 \times 14$	supersonic normal	1.7	$159 \times 105 \times 117$	BSL , SST , $k-\omega$
Zhang (1993)	$72 \times 8 \times 5$	compound angle	0.5 to 1.5	$62 \times 30 \times 28$	$k-\epsilon$, RSM (wall fn.)

Table 5.3.1.2 Comparison of domain sizes and node counts for several *VGJ* studies

The spatial resolution of Grid B was maintained, resulting in $65 \times 37 \times 43$ nodes. Finally, in Grid D the core of the jet contained 15 nodes in the streamwise direction and 9 nodes were placed across the jet shear layer. While resolution of the jet flow in the vicinity of the jet orifice was the highest in this grid, the resolution of the coarser portions of the

domain was between those of A and B. Finally, as is evident from the comparisons in Table 5.3.1.2, the current selections of computational domain size and node counts are significantly lower than the typical choices in the published literature. This reflects the thrust of the present study in attempting to simultaneously minimise computation times and retain high prediction accuracy.

5.3.2 Results of the Grid Sensitivity Study

Figures 5.3.2.1 to 5.3.2.5 summarise the simulation results based on the four grids, all of which were conducted using Rodi's k - ϵ model. Since away from the walls gradients in k and ϵ are typical of those encountered in other turbulence quantities such as ω and $\bar{v}$, optimisation of the grid structure based on a single turbulence model is justified. Although the spatial resolution and domain size vary substantially between the four grids, for the most part only small sensitivity to these parameters is evident in the velocity and k distributions in Figures 5.3.2.1 to 5.3.2.5. The same is true for the value and location of maximum streamwise vorticity shown in Figures 5.3.2.6(a) and 5.3.2.7(a). Maximum vorticity values are almost identical for all grids, and the location of this peak is matched well up to $x/D=3.0$. Beyond this point the substantial distortion of the vorticity field and relatively low magnitude of the peak increases the uncertainty, hence inter-grid comparison of the results is not very reliable. Based on these results, one can thus conclude that the Grid-A configuration is sufficient to obtain results nearly independent of spatial resolution and domain size.

Figure 5.3.2.1 Predicted velocity profiles (plane A₁) based on Rodi's $k-\epsilon$ model, sensitivity to grid resolution
 • Findlay expt. (1998), - - - Grid A, - - - - Grid B,
 - - - - Grid C, ——— Grid D

Figure 5.3.2.2 Predicted velocity profiles (plane A_2) based on Rodi's $k-\epsilon$ model, sensitivity to grid resolution
 • Findlay expt. (1998), ---- Grid A, -.-.-.- Grid B,
 Grid C, ——— Grid D

Figure 5.3.2.3 Predicted velocity profiles (plane A_3) based on Rodi's $k-\epsilon$ model, sensitivity to grid resolution
 ◦ Findlay expt. (1998), --- Grid A, -.-.- Grid B,
 Grid C, ——— Grid D

Figure 5.3.2.4 Predicted k profiles based on Rodi's k - ϵ model, sensitivity to grid resolution (planes A_1 , A_2 , A_3)
 ◦ Findlay expt. (1998), ---- Grid A, - · - · - Grid B,
 · · · · Grid C, ——— Grid D

Figure 5.3.2.5 Comparison of velocity and k profiles (based on Rodi's k - ϵ model) with experimental data, a) W and b) k across shear layer, c) W and d) k across streamwise vorticity field - sensitivity to grid resolution

Figure 5.3.2.6 Predicted maximum streamwise vorticity in the jet-induced vorticity field, a) sensitivity to grids, b) effect of turbulence models (based on Grid D), c) sensitivity to jet boundary conditions (based on Grid D)

Figure 5.3.2.7 Predicted location of max. streamwise vorticity in the jet-induced vorticity field, a) comparison of grids, b) comparison of models (based on Grid D), c) comparison of jet boundary conditions (based on Grid D)

Since the amount of artificial dissipation affecting the solution is related to the spatial resolution, it is essential that the grid optimisation study be performed in this context. Simulations were thus performed with values of 0.005 and 0.01 for the scaling factor for 4th order artificial dissipation, κ_4 . The sensitivity was noted to be small, and only became noticeable for the coarser grids (grids A, B, C). Table 5.3.2.1 summarises the maximum variations noted for several quantities due to changes in spatial resolution and artificial dissipation. The percentage values given for vorticity and k are with respect to the maximum values of the respective quantities encountered within the computational domain, whereas for the velocity components $\bar{V}_j$ is used as the reference. In light of the reduced sensitivity of the simulation results to artificial dissipation observed with the finer spatial resolution, comparisons of turbulence models and alternative considerations of jet-orifice boundary conditions presented in subsequent sections are based on the more conservative spatial resolution and domain size of Grid D.

Variable	Max. variation due to spatial resolution	Max. variation due to κ_4
U	$\pm 6.5\%$	$\pm 2.5\%$
V	$\pm 7.0\%$	$\pm 2.5\%$
W	$\pm 4.5\%$	$\pm 1.0\%$
k	$\pm 6.0\%$	$\pm 6.0\%$
Ω_x	$\pm 4.5\%$	$\pm 3.5\%$

Table 5.3.2.1 Uncertainties due to spatial resolution and artificial diffusion

5.4 Effect of Turbulence Models on Simulation Results

Prediction results for the turbulence models in question are shown for velocity and k in Figures 5.4.1 to 5.4.4, and for vorticity in Figures 5.3.2.6 (b), (c) and 5.3.2.7 (b), (c). At each streamwise location, the value of U is underpredicted in close vicinity of the wall which may be in part due to underestimation of the cross-stream transfer of momentum by the streamwise vortex developing with downstream distance. Generally all models appear to produce similar distributions, although Rodi's k - ϵ model is outperformed by the remaining models in capturing the trends in the turbulence kinetic energy, the velocity component V , and to a lesser extent, the velocity component W . The value of maximum streamwise vorticity (Figure 5.3.2.6 b) is predicted reasonably well with Rodi's k - ϵ and the k - ω models, while the SA and SST models yield very good results. As far as the location of this peak vorticity, shown in Figure 5.3.2.7 (b), all models predict a trajectory that is slightly further away from the wall surface than the measured one, with the discrepancy increasing further downstream, and the lateral convection of the vortex is slightly overestimated.

In summary, the difference between the prediction accuracies of the four turbulence models is not very large. There is consistent evidence, however, that the predictions with Rodi's k - ϵ model are slightly inferior to those with the remaining models in certain respects.

Specifically, all models are capable of capturing the rate of diffusion of the streamwise vortex with downstream distance and the trajectory of this vortex, but predictions of the overall velocity field, hence the extent of cross-stream mass and momentum transfer are not as accurate, with Rodi's k - ϵ model trailing the other ones in this respect.

Figure 5.4.1 Predicted velocity profiles (plane A₁, based on Grid D)
 ◦ Findlay expt. (1998), --- SA model, - · - · - Rodi's $k-\epsilon$ model,
 · · · · · $k-\omega$ model, — Menter's SST model

Figure 5.4.2 Predicted velocity profiles (plane A_2 , based on Grid D)
 ◦ Findlay expt. (1998), --- SA model, - · - · - Rodi's $k-\epsilon$ model,
 - · - · - $k-\omega$ model, — Menter's SST model

Figure 5.4.3 Predicted velocity profiles (plane A₃, based on Grid D)
 ◦ Findlay expt. (1998), --- SA model, - · - · - Rodi's k - ϵ model,
 · · · · · k - ω model, — Menter's SST model

Figure 5.4.4 Predicted k profiles (planes A_1 , A_2 , A_3 , based on Grid D)
 • Findlay expt. (1998), --- Rodi's k - ϵ model,
 - · - - k - ω model, — Menter's SST model

5.5 Alternative Treatments of Jet-Orifice Boundary Conditions

Although the simulations presented so far were based on an accurate description of the flow nonuniformity at the jet exit plane, this approach is not practical since the type of nonuniformity would depend significantly on the jet configuration, the details of the jet pipe and plenum design, as well as the crossflow conditions. For the present approach of specifying inflow boundary conditions at the jet exit plane to be practical, a more generic distribution must suffice. Sensitivity of the *VGJ* flow field to this jet boundary condition was therefore tested by comparing uniform distributions of velocity and turbulence quantities to the original, nonuniform results. The velocity magnitude was adjusted to match the mass flow of the nonuniform case, hence maintaining the same jet-to-crossflow velocity ratio. As for the nonuniform case, a thin boundary layer ($\delta=0.16D$) was specified along the perimeter of the jet orifice, so as to avoid sharp velocity gradients that may have triggered solution instability due to excessive levels of shear. An average k value was calculated from the nonuniform distribution and was specified along with a uniform ε value calculated from this value using Equation 5.1.1.1. Figures 5.5.1 to 5.5.5 display the results for velocity and k distributions. Results for maximum vorticity value and location are shown in Figures 5.3.2.6(c) and 5.3.2.7(c).

Distinct variations are noted in these figures due to uniform versus nonuniform treatment of the jet-exit-plane conditions. However, these variations are noted to be generally within the margin of error of the simulations with respect to experimental results. Thus, it

appears that uniform boundary conditions may be imposed at the jet-exit plane without notable changes in prediction accuracy when one of the present turbulence models is used. However, it should be noted that this statement is based on tests with a relatively high VR , and there is evidence that it may not be applicable to lower VR values. For example, Findlay (1998) observed higher flow nonuniformity at the jet exit plane with decrease in VR , and Khan (1999) noted that the turning of the jet flow was already initiated before leaving the jet hole for a VGJ with $VR=1.0$.

Figure 5.5.1 Comparison of velocity profiles (plane A_1) with experimental data, sensitivity to jet boundary condition (results are based on Grid D)
 ◦ Findlay expt. (1998), --- SA model, - · - · - Rodi's k - ϵ model, · · · · · k - ω model, — Menter's SST model
 (red curves: uniform jet-exit flow; black curves: nonuniform jet-exit flow)

Figure 5.5.2 Comparison of velocity profiles (plane A₂) with experimental data, sensitivity to jet boundary condition (results are based on Grid D)

◦ Findlay expt. (1998), --- SA model, - · - · - Rodi's $k-\epsilon$ model, · · · · · $k-\omega$ model, — Menter's SST model
(red curves: uniform jet-exit flow; black curves: nonuniform jet-exit flow)

Figure 5.5.3 Comparison of velocity profiles (plane A_3) with experimental data, sensitivity to jet boundary condition (results are based on Grid D)
 • Findlay expt. (1998), --- SA model, - - - Rodi's k - ϵ model, $\cdots$ k - ω model, — Menter's SST model
 (red curves: uniform jet-exit flow; black curves: nonuniform jet-exit flow)

Figure 5.5.4 Comparison of k profiles (planes A_1 , A_2 , A_3) with experimental data, sensitivity to jet boundary condition (results are based on Grid D)
 ◦ Findlay expt. (1998), - - - Rodi's k - ϵ model,
 k - ω model, — Menter's SST model
 (red curves: uniform jet-exit flow; black curves: nonuniform jet-exit flow)

Figure 5.5.5 Comparison of velocity and k profiles (Grid D) with experimental data;
a) W and b) k across shear layer, c) W and d) k across streamwise
vorticity field - sensitivity to jet boundary condition
◦ Findlay expt. (1998), --- SA model, - · - · - Rodi's $k-\epsilon$ model,
- · - · - $k-\omega$ model, — Menter's SST model
(red curves: uniform jet-exit flow; black curves: nonuniform jet-exit flow)

6.0 Simulations of the M2129 Diffusing S-Duct Flow

In the previous chapter the prediction capabilities of several turbulence models were established for the flow field of *VGJ*'s, and the minimum grid / computational-domain-size requirements were identified for proper simulation of these flows. The last step towards completing the evaluation of *CFD* simulations as a viable tool for design optimisation of aircraft-engine-intakes is to examine prediction capabilities and computing requirements for a typical intake duct. This chapter presents the simulation results for such a test case.

6.1 Description of the Test Case

The M2129 S-duct experiment of Yu (1991) was chosen as a reference for the present simulations. The M2129 geometry is shown in Figure 6.1.1, and key dimensions are given in Tables 6.1.1 and 6.1.2. The centreline offset and variation of duct diameter with centreline distance are defined in Equations 6.1.1 and 6.1.2. It should be noted that parts (a) and (c) of these equations describe extensions upstream and downstream of the original M2129 duct geometry, that were included in the computational domain to simplify the specifications of inflow and outflow boundary conditions.

Figure 6.1.1 Computational domain for the M2129 diffusing S-duct

Plane	s/ℓ	γ (deg.)
Inflow	-0.28	0
A_1	0	0.0°
A_2	0.23	17.6
A_3	0.53	25.1
A_4	0.76	17.7
A_5	1	0
Outflow	1.49	0

Table 6.1.1 Location and angle of planes used in data comparison and boundary settings
* orientations of all planes are referenced to plane A_1

$AR = D_2^2/D_1^2$	ℓ/D_1	$ \Delta z_{CL}/\ell $	ℓ (Yu, 1991)	D_1 (Yu, 1991)
1.4	3.55	0.3	170.4mm	48mm

Table 6.1.2 Basic dimensions of the M2129 diffusing S-duct

$$-0.28 \leq s/\ell \leq 0.00 \quad z_{CL}/\ell = 0 \quad (6.1.1a)$$

$$0.00 \leq s/\ell \leq 1.00 \quad \frac{z_{CL}}{\ell} = \frac{\Delta z_{CL}}{2\ell} \left(1 - \cos \left(\pi \frac{s}{\ell} \right) \right) \quad (6.1.1b)$$

$$1.00 \leq s/\ell \leq 1.49 \quad z_{ct}/\ell = \Delta z_{ct}/\ell \quad (6.1.1c)$$

$$-0.28 \leq s/\ell \leq 0.00 \quad D = D_1 \quad (6.1.2a)$$

$$0.00 \leq s/\ell \leq 1.00 \quad \frac{D - D_1}{D_2 - D_1} = 3 \left(1 - \frac{s}{\ell}\right)^4 - 4 \left(1 - \frac{s}{\ell}\right)^3 + 1 \quad (6.1.2b)$$

$$1.00 \leq s/\ell \leq 1.49 \quad D = D_2 \quad (6.1.2c)$$

A more popular test case used for *CFD* studies of diffusing S-ducts has been the M2129 experiments of Willmer et al. (1981). Although the geometries employed in each test are identical in non-dimensional terms, two important differences exist between these experiments. The conditions of Willmer et al.'s experiments were given in Section 1.3.3.2. In these experiments, inlet Mach numbers of 0.412 and 0.794, and Reynolds numbers (based on inlet velocity and diameter) of 1.158×10^6 and 1.848×10^6 were tested to provide more realistic aircraft flight conditions. Yu's experiments, on the other hand, were performed at an inlet Mach number less than 0.2 and a Reynolds number of 4.3×10^4 . Clearly, the flow regime studied by Yu represents unrealistic aircraft speeds. Nonetheless, inspection of simulated results of either test case (e.g., Anderson et al., 1993, Yu and Lee, 1994) showed them to contain very similar flow features. Therefore, since the goal of the present study is to evaluate turbulence model prediction capability of the three-dimensional duct flow field, the differences in Mach numbers and Reynolds numbers do not detract from the suitability of Yu's data as a benchmark. Yu's data was chosen over Willmer et al.'s due to the high-resolution flow field data collected inside the duct as well as at the entrance. Two-component laser-doppler anemometer measurements of velocity

and turbulence were taken in addition to wall static pressure. The experiments of Willmer et al. consisted only of wall static pressure and exit plane total pressure measurements. Finally, it should be noted that the lower Reynolds number of Yu's experiments means that, relative to Willmer's test conditions, the boundary layer thickness along the duct walls is greater, hence adequate resolution of the flow in the cross-sectional plane is feasible with fewer number of grid nodes. The conclusions of this chapter with respect to computational times must therefore be interpreted in this context.

6.1.1 Boundary and Initial Conditions

The inflow boundary was placed one D_1 upstream of the original entrance (i.e., beginning of the bent portion) of the M2129 S-duct. This allowed specification of unidirectional flow conditions at this boundary. The freestream flow at this boundary was set to 15.3 m/s and the area averaged velocity was 13.4 m/s. This together with the chosen values of $\rho=1.2 \text{ kg/m}^3$ and $\mu=1.8 \times 10^{-5} \text{ Pa}\cdot\text{s}$ corresponded to the Reynolds number of Yu's experiment. The freestream turbulence kinetic energy was also scaled from Yu's experimental data, and the remaining turbulence properties were estimated based on a dissipation rate calculated as:

$$\varepsilon = \frac{k^{3/2}}{0.3D_1} \quad (6.1.1.1)$$

The freestream turbulence conditions at the inflow boundary are summarised in Table 6.1.1.1. It is noted that unlike the previous test cases, the freestream turbulence specified at the inflow boundary is relatively high in this instance ($Tu \approx 3\%$).

Model	Turbulence quantity
Rodi's k - ε model	$k = 0.36$ $\varepsilon = 16.5$
k - ω model	$k = 0.36$ $\omega = 477$
Menter's SST model	$k = 0.36$ $\omega = 477$
SA model	$\chi = 44.7$

Table 6.1.1.1 Freestream values of turbulence parameters specified at the inflow boundary

The radial profiles of streamwise velocity and turbulence kinetic energy at the inflow boundary were scaled from the experimental data. The k data set was relatively sparse very close to the duct wall, hence this region was supplemented with simulated flat plate boundary layer data corresponding to a similar value of $Re_\theta = 875$ and freestream turbulence. The profiles of ε , ω and $\chi = \bar{v}/\nu$ were also obtained from these flat plate simulations. Upon specification of these flow variations in the radial direction through 2-D polynomial fits, the results were mapped to all circumferential positions (a feature provided by the preprocessor of the FS code) since the flow condition in Yu's experiments were axisymmetric. As in the previous test cases, static pressure was not fixed at the inflow boundary, and was obtained as part of the solution through extrapolation from within the computational domain. Static pressure was specified at the outflow boundary.

A uniform pressure distribution was found to offer better rates of solution convergence than extrapolation of cross-stream pressure gradients from within the domain. The outflow boundary was placed sufficiently downstream of the exit plane of the bend ($1.75D_i$) to prevent this uniform cross-stream pressure specification at the outflow boundary from affecting the natural development of the flow within the bend. When solutions, originally converged with uniform outflow pressure distribution, were further iterated in pseudo-time with allowance for nonuniform pressures to develop at the outflow boundary, no changes were observed in the flow field within the curved portions of the duct.

To initialise the simulations, static pressure was set to a uniform value throughout the domain, matching the outflow boundary value, and the flow velocity was set to 1 m/s pointing in the x -direction.

The initial values and thresholds of the turbulence quantities are given in Table 6.1.1.2. These values correspond to the optimum settings determined in Section 4.1. In addition to the lower limits, the same upper limit of 20 used in the *VGJ* study (Chapter 5) for the production-to-destruction ratio of k was found to be necessary for stable convergence of the turbulence field with the two-equation models. Furthermore, the $k-\omega$ and Menter's *SST* models benefited from an upper limit that was placed on the eddy-viscosity as well. Without this limit, localised regions of high eddy-viscosity would tend to appear near the duct walls at various streamwise and circumferential positions during transient

development of the flow in pseudo-time. Examination of the converged flow field indicated peak values of μ_t/μ not exceeding 200. Hence the upper limit setting of $\mu_t/\mu=350$ did not artificially influence the converged solution.

	$k_{initial}$	$\varepsilon_{initial}$	$\omega_{initial}$	$\chi_{initial}$	k_{min}	ε_{min}	ω_{min}	χ_{min}	$\mu_t max/\mu$	$P_k/S_t max$
SA	-	-	-	0.77	-	-	-	0.77	-	-
Rodi's $k-\varepsilon$	0.1	7	-	-	0.001	7	-	-	-	20
$k-\omega$	2e-06	-	100	-	2e-06	-	100	-	350	20
SST	2e-06	-	100	-	2e-06	-	100	-	350	20

Table 6.1.1.2 Turbulence-model initial conditions and threshold settings

Finally, the settings of artificial compressibility artificial diffusion and pseudo-time step limits are summarised in Table 6.1.1.3.

C_{AD}	CFL	K_d	κ_4
$\max[10; 3.2V_{local}]$	1.8	0.1	0.005

Table 6.1.1.3 Simulation parameters employed for the M2129 diffusing S-duct

6.2 Description of the Flow Physics

Before discussions on grid optimisation and turbulence-model comparisons, this section introduces the physics of the flow to provide the reader with sufficient background information for the discussions that follow. The important flow features are identified and discussed using results from the *SA* turbulence model summarised in Figure 6.2.1.

Figure 6.2.1 M2129 diffusing S-duct flow predicted with the *SA* model; ($\bar{U}=13.4$ m/s)
a) streamwise velocity, b) streamwise vorticity, c) total pressure

As noted in Fig. 6.2.1(b), between planes A_1 and A_2 a lobe of positive streamwise vorticity appears adjacent to the side wall ($t_1/D=0.5$, $t_2/D=0.0$). This early stage of the development of a classical streamwise vortex arises from the rotation of the incoming boundary layer vorticity lines due to the flow turning in the first bend. In between this region of positive vorticity and the duct wall lies a thin layer of negative vorticity attributed to the induced secondary flows. The imposed cross-stream momentum near the wall producing this sense of wall vorticity would oppose the generation of secondary flow (which would be of opposite sign) in the second bend. Nonetheless, limited development of such secondary flows is still present in the second bend, as displayed by the vorticity contours in plane A_3 and the outflow plane.

Between Planes A_2 and A_3 the lobe of positive vorticity generated in the first bend develops into a well-defined streamwise vortex. The cross-stream convection of this vortex is noted to be consistent with the corresponding induced velocity field near the walls as shown in Figure 6.2.1a.

Further insight is gained by examining the vector-field topology through streamline traces next to the duct wall as shown in Figure 6.2.2. Between planes A_2 and A_3 such limiting streamlines are shown to be reoriented from the “streamwise” direction, briefly pointing upstream and then concentrating into a pair of focus nodes. This phenomenon has been referred to by such authors as Anderson et al. (1993) as *vortex lift-off*. It may be argued

Figure 6.2.2 Vortex lift-off in the M2129 diffusing S-duct

that this flow pattern is the response of the low momentum fluid, swept into this region by the streamwise vortices of the first bend, to the prevailing adverse streamwise pressure gradients. As the streamwise vortex pair continuously feeds low momentum fluid into this region, and this fluid is opposed by the pressure forces in the streamwise direction, it lifts off the surface in the form of a new pair of tornado-like vortices orthogonal to the wall.

Once reoriented in the the streamwise direction by the streamwise momentum of the flow further away from the wall, the generated “lift-off” vortices would have the same sense of rotation as the streamwise vortex pair generated in the first bend, and thus the two pairs most likely merge shortly after lift-off. Evidence of this merging of the vortex pairs is seen in Figure 6.2.1(b) where the streamwise vortices of the first bend are noted to loose strength between planes A_2 and A_4 , a trend that is reversed between planes A_4 and A_5 .

Finally, performance of this diffuser is depicted in Figure 6.2.3. The static pressure recovery can be seen to fall far short of the ideal value due to a combination of viscous losses and flow distortion. The development of the pressure loss is fairly gradual with downstream distance which suggests that the aforementioned lift-off phenomena and subsequent development of the vorticity field are not significant loss-generation mechanisms in mass-averaged terms.

Figure 6.2.3 Total pressure losses and static-pressure recovery performance of the M2129 diffusing S-duct

6.3 Grid Optimisation Studies

Main features of each grid tested in this study are summarised in Table 6.3.1 and the grids themselves are displayed in Figure 6.3.1. Each grid was generated based on the guidelines established in Sections 4.1 and 4.2. At the inflow plane, the first node off the wall was set to $y_1^+ = 1.2$ and a minimum of 20 nodes were placed within the boundary layer. This type of node distribution at the inflow boundary yielded a minimum resolution of 15 nodes in the boundary layer and a maximum y_1^+ value of 1.3 anywhere within the duct.

In order to confirm grid independence of the results, refinements of Grid A were considered with the additional nodes placed in the cross-stream directions (Grid B), the streamwise direction (Grid C), and in both cross-stream and streamwise directions (Grid D). To put the current choices of grids in context, the grids employed in some of the published M2129 simulation studies are given in Table 6.3.2. It is noted that Grid A contains noticeably fewer nodes particularly in the cross-stream plane than the grids in any of these studies. This choice was intentional, since one of the objectives of the present study is to establish compatibility of the most competitive computation times for such flows with typical turnaround times desired in a design / optimisation environment.

Figure 6.3.1 Grids tested for the M2129 diffusing S-duct
a) Grid A ($N_i=57 \times N_j=25 \times N_k=53$), b) Grid B ($57 \times 31 \times 61$),
c) Grid C ($97 \times 25 \times 53$), d) Grid D ($97 \times 31 \times 61$)

Grid	number of nodes in the:					y_1^+ at the inflow boundary
	i direction	j direction	k direction	cross-plane	inflow BL	
A	57	25	53	1325	20	1.2
B	57	31	61	1891	23	1.2
C	97	25	53	1325	20	1.2
D	97	31	61	1891	23	1.2

Table 6.3.1 Grid details for the M2129 diffusing S-duct

Reference	number of nodes in the:			
	streamwise direction	radial direction	circum. direction	cross-plane
Anderson & Gibb (1992)	61	49	49	2401
Lee & Yu (1994)	123	50	50	2500
Marquis & Ong (1995)	87	41	49	2009

Table 6.3.2 Comparison with grids of published M2129 simulation studies

Mapping of a structured grid onto a circular geometry, without the aid of subdividing the grid into multiple “blocks,” results in local regions of substantial skewness as shown in Figure 6.3.2. This inherent grid skewness, combined with the need to cluster the grid nodes towards the duct walls so as to resolve the boundary layer, yields a local layer of grid cells with internal angles as small as 2° ! To alleviate this problem to some extent, as many such cells as possible were blocked off, i.e., were excluded from the computational

domain. Two $6 \text{ cell} \times 6 \text{ cell}$ portions of the computational domain were blocked off on grids A to D as shown in the *computational space* inset of Figure 6.3.2. As indicated in the *physical space* inset of Figure 6.3.2, this resulted in a maximum geometry deviation of $0.1\% D_i$ from a perfectly circular wall, which was deemed acceptable. Despite this process of blocking off cells, there still remained considerable grid skewness at two circumferential positions of the cross-sectional plane. Examination of the simulation results, however, did not indicate any noticeable differences in the boundary layer structure in these skewed regions compared to elsewhere along the duct perimeter.

Figure 6.3.2 Highly-skewed region of the computational grid for the M2129 S-duct

The velocity distributions obtained with the four different grids are compared to each other in Figures 6.3.3 to 6.3.7. From these figures it is evident that the spatial resolution offered by Grid A is sufficient to obtain essentially grid independent results.

As was discussed in the context of the *VGI* simulations, the extent of artificial dissipation affecting the simulation results is a topic that must be addressed in conjunction with

optimisation of the spatial grid. This is because the explicit artificial dissipation terms contained in the present algorithm are obtained from 2nd and 4th order spatial differences of the solution vector, Q . Comparison of simulation results with values of 0.005 and 0.01 for the scaling coefficient of the dissipation term, κ_4 , indicated very small sensitivity to such variations in κ_4 with Grid D, whereas the sensitivity was noticeable with the remaining, coarser grids. Based on the good agreement between the results obtained with all four grids as presented in Figures 6.3.3 to 6.3.7, which utilised $\kappa_4=0.005$, it appears that this κ_4 value is sufficiently low for all of the grids considered here. Nonetheless, the comparisons of turbulence models presented in the next section were based on Grid D to eliminate any possibility of increased uncertainty due to small artificial dissipation effects on the coarser grids.

Figure 6.3.3 Grid sensitivity of simulation results in Plane A₁ ($s/l = 0.0$)
a) distribution along t_1 , b) distribution along t_2

Figure 6.3.4 Grid sensitivity of simulation results in Plane A₂ ($s/l = 0.24$)
a) distribution along t_1 , b) distribution along t_2

Figure 6.3.5 Grid sensitivity of simulation results in Plane A₃ ($s/l = 0.54$)
a) distribution along t_1 , b) distribution along t_2

Figure 6.3.6 Grid sensitivity of simulation results in Plane A₄ ($s/l = 0.76$)
a) distribution along t_1 , b) distribution along t_2

Figure 6.3.7 Grid sensitivity of simulation results in Plane A_3 ($s/l = 1.0$)
a) distribution along t_1 , b) distribution along t_2

6.4 Effect of Turbulence Models on Simulation Results

For all four turbulence models the predicted streamwise and cross-stream velocity distributions are compared to experimental data in Figures 6.4.1 to 6.4.5, and wall static pressure comparisons are given in Figure 6.4.6. Due to the availability of experimental data, only the “ t_2 ” component of cross-stream velocity is presented. Nonetheless, this should be sufficient to provide an overall indication of the prediction accuracy for the secondary motions in the duct. All four turbulence models are noted to capture the general distribution of velocity reasonably well. It is noteworthy, however, that the prediction error is significantly higher in the cross-stream motion if the results are viewed in the context of the respective magnitudes of streamwise and cross-stream velocity components. As for the streamwise velocity distribution, the one area where all four

models are in mutual agreement, yet fail to follow the experimental trend is along the side portion of the duct ($t_1=D/2$, $t_2=0$). The measured V_z distribution at this location is indicative of a notably thicker streamwise boundary layer than the predicted ones, and this trend is persistent at all five cross-stream planes. This discrepancy may be the combined result of the accuracy with which the streamwise pressure gradient is predicted along the walls (Figure 6.4.6) as well as the prediction accuracy for the cross-stream motion, since both of these factors influence the streamwise momentum of the local flow..

The variations amongst the turbulence models do not appear to be substantial with the exception of the region of vortex lift-off and further downstream of this region, as observed in the comparisons within planes A_3 , A_4 and A_5 . Particularly, Rodi's $k-\epsilon$ model is noted to be consistently outperformed by the remaining turbulence models in this portion of the duct.

The wall static pressure distributions at the "top," "bottom" and "side" of the duct (Figure 6.4.6) display a generally acceptable agreement between the predictions and experimental data. Amongst the four turbulence models, the prediction of Menter's *SST* model are noted to be consistently more accurate at all three circumferential positions. The relatively poor performance of Rodi's $k-\epsilon$ model is consistent with the model's lesser ability in capturing the vortex lift-off about halfway through the duct.

Figure 6.4.1 Predictions of the velocity field in Plane A₁ ($s/l = 0.0$) compared with the experimental data of Yu (1991)
a) distribution along t_1 , b) distribution along t_2

Figure 6.4.2 Predictions of the velocity field in Plane A₂ ($s/l = 0.24$) compared with the experimental data of Yu (1991)
a) distribution along t_1 , b) distribution along t_2

Figure 6.4.3 Predictions of the velocity field in Plane A₃ ($s/l = 0.54$) compared with the experimental data of Yu (1991)
a) distribution along t_1 , b) distribution along t_2

Figure 6.4.4 Predictions of the velocity field in Plane A₄ ($s/l = 0.76$) compared with the experimental data of Yu (1991)
a) distribution along t_1 , b) distribution along t_2

Figure 6.4.5 Predictions of the velocity field in Plane A_s (s/≠ 1.0) compared with the experimental data of Yu (1991)
a) distribution along t_1 , b) distribution along t_2

Figure 6.4.6 Predicted wall static pressure compared with the experimental data of Yu (1991)
 a) streamwise distribution along $t_1 = 0, t_2 = D/2$
 b) streamwise distribution along $t_1 = D/2, t_2 = 0$
 c) streamwise distribution along $t_1 = 0, t_2 = -D/2$

7.0 Conclusions

Five turbulence models have been evaluated for prediction accuracy, numerical robustness and computational efficiency based on test cases ranging from a two-dimensional equilibrium boundary layer to three-dimensional separated flows. The objective of these evaluations has been to establish the feasibility of using a *Navier-Stokes* solver in the design-cycle optimisation of high-performance aircraft engine intakes. The choices of the more complex test cases included in this study, namely a diffusing S-duct representing a typical engine intake with centerline curvature and a vortex-generator jet for flow control, reflect this objective of the study.

The group of turbulence models included in this study consists of the low- Re k - ϵ model of Chien (1982), the two-layer k - ϵ model of Rodi et al. (1988), the k - ω model of Wilcox (1988), the two-equation shear-stress-transport model of Menter (1994), and the one-equation eddy-viscosity model of Spalart and Allmaras (1992). All of these models were integrated to the surface at wall boundaries rather than employing wall functions which are known to be incompatible with separated flows. Despite the fundamental shortcomings of lack of turbulence anisotropy and absence of Reynolds-stress relaxation in response to sudden changes in the strain field associated with such eddy-viscosity models, they are substantially more efficient than higher order Reynolds stress closures. Thus, the current study was started with the hypothesis that such *PDE* based eddy-viscosity type of models are more likely to yield an acceptable combination of computational efficiency and

accuracy which are equally important factors in an iterative design optimisation framework. The following observations resulting from the extensive simulations contained in this study allow evaluation of this hypothesis:

- All turbulence models were found to provide satisfactory development of the boundary layer over a flat plate in terms of velocity and surface shear stress distributions. At the same time, only Chien's k - ϵ model was able to reproduce the trends in the profile for turbulence kinetic energy, albeit only qualitatively.
- All but Chien's k - ϵ model were fairly successful in capturing the surface pressure and skin friction distributions in an axisymmetric separating flow. A slight overprediction of the static pressure in the separated zone was common for the models. Menter's *SST* model was the most successful in capturing the trends in the velocity profiles followed by the *SA* and k - ω models; Rodi's k - ϵ model performed rather poorly in this respect. Those models that solve for the turbulence kinetic energy, k , failed to capture the peak k value in the boundary layer, which is consistent with the observations in the flat plate flow.
- In terms of minimum grid resolution requirements for both the flat plate and the axisymmetric separating flows, Rodi's k - ϵ and the *SA* models showed the best performance, requiring a maximum of $y_1^+ = 5$ and at least 15 nodes within the boundary layer for acceptable prediction accuracy. The required minimum node count was about the same for the k - ω and *SST* models, whereas the upper limit for y_1^+ had to be set at about 1.3 for comparable accuracy.

- Simulations were performed for a periodic vortex-generator-jet (VGJ) flow of 30 deg. pitch, 90 deg. skew, a jet-to-crossflow velocity ratio of 1.5, and an approach boundary layer thickness of about twice the streamwise dimension of the jet orifice (D). For this configuration, node counts of 47, 37 and 29 in the streamwise, spanwise and vertical directions were found to be sufficient to yield essentially grid-independent results with the computational domain being $13D$ long, $5D$ high and $3D$ wide. Accounting for the flow nonuniformity at the jet orifice inflow boundary was found to have a noticeable effect on the prediction accuracy. However, the extent of this effect was observed to be comparable to the typical difference between measured data and predictions based on any one of the turbulence models.
- The difference between the prediction accuracies of the four turbulence models (Rodi's $k-\epsilon$, Menter's SST , $k-\omega$, and SA) in the VGJ flow was not very significant. However, the predictions with Rodi's $k-\epsilon$ model were noted to be consistently less accurate than with the remaining models in certain respects. Specifically, all four models were able to reproduce the rate of diffusion of the streamwise vortex with downstream distance and the trajectory of this vortex reasonably well, but predictions of the overall velocity field, hence the extent of cross-stream mass and momentum transfer were not as accurate, with Rodi's $k-\epsilon$ model trailing the other models in this respect.
- Finally, grid optimisation studies for a diffusing S-duct, with a geometry that is typical of a longitudinally curved aircraft engine intake, revealed that nearly grid

cross stream plane and about 60 nodes in the streamwise direction, with any one of the four turbulence models. All four models were able to capture the overall streamwise and secondary flow development reasonably well. Rodi's k - ϵ model was noted to be particularly less accurate in the region of vortex lift off. This observation is consistent with the Rodi k - ϵ results in the *VGJ* flow where vortex lift off just downstream of the jet orifice is also a key feature of the overall flow development.

A byproduct of the present study is the evaluation of a particular *Navier-Stokes (NS)* algorithm for three-dimensional separated flows. It has been demonstrated that iterative solution of the *NS* equations through explicit pseudo-time marching based on *Runge-Kutta* integration, combined with local sizing of pseudo-time steps and a multigrid procedure as acceleration schemes, and implicit residual smoothing as a stability enhancement mechanism, is an efficient and robust algorithm for such flows. Additionally, explicit control over artificial dissipation terms facilitated by the present algorithm has been shown to be very effective in being able to suppress the development of spurious spatial oscillations, without introducing excessive numerical diffusion into the solution that would undermine the predictive capability of the turbulence models. Finally, the capability of a certain type of preconditioning of the mass and momentum equations to allow monotonic and efficient convergence in the incompressible regime has been demonstrated, and appropriate settings for the relevant preconditioning parameters have been provided for the present flows.

8.0 Recommendations for Future Studies

The present study has established the minimum computational requirements and predictive capabilities of several turbulence models for diffusing-S-duct and vortex-generator-jet (*VGJ*) flows. The next step should be to consider the ultimate case of an S-duct furnished with one or more rows of *VGJ*'s for the purpose of flow control (hereafter referred to as the S-duct+*VGJ* configuration). As became evident from the results presented in Chapters 5 and 6, the S-duct and *VGJ* flows have many dominant features in common, thus it may be reasonable to expect the turbulence models to yield similar accuracy in the S-duct+*VGJ* configuration. Nonetheless, it would be very desirable to confirm this postulate through comparisons with detailed experimental data. Therefore, the next step in this line of study would have to be detailed measurements of the velocity and turbulence field in an S-duct utilising *VGJ* flow control.

The remaining issue is then one of computational efficiency, i.e. to efficiently combine the minimum grid requirements for the S-duct and the *VGJ*. For the purpose of boundary layer control to prevent separation and minimise cross-stream motion, an intake design could be expected to incorporate around 10 *VGJ*'s evenly distributed along the wall of each symmetric half (Gibb and Anderson, 1995). To provide the required resolution for the duct and the individual jets, without placing nodes where they are not required, a multiblock arrangement such as the one shown in Figure 8.1 could be used. In this configuration, the finer spatial resolution required for the individual jets may be specified

within a ring-shaped block (Block B) while maintaining the optimum number of nodes elsewhere in the duct. Based on the arguments made in Section 1.3.3, the *VGJ* setup tailored to counteract pre-existing distortion in the duct, rather than to control the local boundary layer, would incorporate fewer jets with higher velocity ratio since the generated vortices would have to penetrate well beyond the edge of the local boundary layer. In place of a narrow ring as shown in Figure 8.1, such an installation would then necessitate multiple, larger blocks of grid spread along the circumference of the duct at a selected streamwise position, each encompassing one *VGJ* flow region.

Figure 8.1 Schematic of a conceptual multiblock S-duct+*VGJ* computational domain

It is also noted that the multiblock approach would enable the use of orthogonal grids along the perimeter of the S-duct as opposed to the locally skewed type that was presented in Chapter 6. Figure 8.2 displays a typical cross-sectional grid-node distribution for the S-duct based on the multiblock concept.

Figure 8.2 Achievable level of grid orthogonality using the multiblock approach

Listed in Table 8.1 are the optimum grids for the *VGJ* and M2129 diffusing S-duct as presented in Chapters 5 and 6, together with the estimated node count for the S-duct+*VGJ* installation.

Case	Total number of nodes
<i>VGJ</i>	~50000
S-duct	~75000
projected S-duct+ <i>VGJ</i>	~630000

Table 8.1 Node counts for *VGJ*, S-duct and S-duct+*VGJ*

As was noted in Chapter 6, the Reynolds number of the S-duct simulations included in the present study was significantly lower than the typical values that would be expected on an aircraft installation. Thus, the actual node-count requirements for the duct, hence the projected value for the S-duct+*VGJ* configuration, would be somewhat higher than the value given in Table 8.1. It is clear that, for today's standard computational resources, the

resources, the resultant grid size is too large for such simulations to be integrated into an iterative design optimisation process.

Segmenting the computational domain through the multiblock approach makes this simulation a good candidate for coarse-grain parallel computing (e.g., Hall et al., 1995). The implementation of such parallel computing for the S-duct+VGJ configuration would make design-cycle analysis practical since the computing time could easily be manipulated through adjustments of the number of *CPU*'s. The *FS* code is capable of simulations based on a multiblock grid and has also been tailored for easy extension into a parallel-computing environment. Trial simulations of an S-duct utilising *VGJ* flow control using a network of Pentium computers is planned for the near future.

One problem with applying the standard multiblock approach arises when it is desired to test various streamwise locations for the *VGJ*'s as part of a design optimisation process. In the standard approach, essentially the entire grid-block structure in the S-duct would have to be updated. The implementation of a *chimera* grid would solve this problem (e.g., Chan and Buning, 1995). In this case, the blocks for the *VGJ*'s would simply overlap the block(s) of the "base" grid associated with the S-duct, hence a set of jet blocks could be readily moved around within the duct without requiring modification of the base grid.

Acknowledgments

The author would like to thank Dr. M.I. Yaras for his supervision of the research described in this thesis. In particular, the author is grateful to Dr. Yaras for introducing him to CFD and his teachings on state-of-the-art numerical methods and turbulence modelling.

Without the generalized, numerically robust and computationally efficient *FS* code developed by Dr. Yaras the extensive study of such complex flows would not have been possible. Financial support from the National Research Council of Canada under contract no. 31184-8-8847/001/ST, and the assistance of Dr. T. Currie in setting up this contract are hereby gratefully acknowledged. Dr. Currie made this extremely interesting graduate research topic possible and provided valuable literature at the beginning of the project.

Thanks are also due to Dr. I.S. Gartshore and Dr. M.J. Findlay for making their experimental data on the inclined-jet-in-crossflow available. Without such a detailed data set the *VGJ* prediction evaluation would not have been possible. The author is grateful to Dr. J. Goldak for his moral support and valuable technical discussions throughout the graduate program. Dr. H.I.H. Saravanamuttoo is thanked for introducing the author to gas turbine propulsion and revealing the many aspects of propulsion system design while making this topic extremely enjoyable to learn. Thanks are also due to Dr. R.J. Kind who generously provided valuable reference material. The author's friends and colleagues: M.Benner, M.Conlon, M.Kermani, V.Kulkarni, R.Lee and A.Mahallati are thanked for their moral support and valuable technical discussions. Finally, the author owes a great debt to his family for their undying support which made his education possible.

A.1 References

Advisory Group for Aerospace Research and Development (AGARD), 1991, "Air Intakes for High Speed Vehicles," Fluid Dynamics Panel Working Group 13, AR-270

AIR International, July 1998, pp. 18-23.

Alvarez, J., Jones, W.P., 1993, "Computation of a Jet Discharging into a Cross-Flow with a Second Moment Closure and a Low-Diffusive Convection Discretisation Scheme," *Engineering Turbulence Modelling and Experiments 2*, Edited by Rodi, W., and Martelli, F., Elsevier Science, pp. 239-249.

Alvarez, J., Jones, W.P., Seoud, R., 1993, "Predictions of Momentum and Scalar Fields in a Jet in Cross-Flow using First and Second Order Turbulence Closures," AGARD-CP-534, AGARD Fluid Dynamics Panel Symposium on 'Computation and Experimental Assessment of Jets in Cross Flow,' Winchester, U.K., pp.24-1 - 24-10.

Anabtawi, A.J., Blackwelder, R.F., Lissaman, P.B.S, Liebeck, R.H., 1999a, "An Experimental Investigation of Boundary Layer Ingestion in a Diffusing S-Duct with and without Passive Flow Control," AIAA Paper 99-0739.

Anabtawi, A.J., Blackwelder, R.F., Lissaman, P.B.S, Liebeck, R.H., 1999b, "An Experimental Study of Vortex Generators in Boundary Layer Ingesting Diffusers with a Centerline Offset," AIAA Paper 99-2110.

Andersen, P.S., Kays, W.M., Moffat, R.J., 1972, "The Turbulent Boundary Layer on a Porous Plate: An Experimental Study of the Fluid Mechanics for Adverse Free-Stream Pressure Gradients," Rept. No. HMT-15, Stanford University Dept. Mech Eng.

Anderson, B.H., 1991, "The Aerodynamic Characteristics of Vortex Ingestion for the F/A-18 Inlet Duct," AIAA Paper 91-0130.

Anderson, B.H., Gibb, J., 1992, "Application of Computational Fluid Dynamics to the Study of Vortex Flow Control for the Management of Inlet Distortion," AIAA Paper 92-3177.

Anderson, B.H., Gibb, J., 1993, "Study on Vortex Generator Flow Control for the Management of Inlet Distortion," J. Propulsion and Power, Vol. 9, pp. 422-430.

Anderson, B.H., Reddy, D.R., Kapoor, K., 1993, "A Comparative Study of Full Navier-Stokes and Reduced Navier-Stokes Analyses for Separating Flows Within a Diffusing Inlet S-Duct," AIAA Paper 93-2154.

Anderson, B.H., Gibb, J., 1996, "Vortex Generator Installation Studies on Steady State and Dynamic Distortion," AIAA Paper 96-3279.

Anderson, B.H., Gibb, J., 1998, "Vortex-Generator Installation Studies on Steady-State and Dynamic Distortion," J. Aircraft, Vol. 35, pp. 513-520.

Anderson, B.H., Miller, D.N., Yagle, P.J., Truax, P.P., 1999, "A Study on MEMS Flow Control for the Management of Engine Face Distortion in Compact Inlet Systems," ASME Paper FEDSM99-6920.

Armfield, S.W., Fletcher, C.A.J., 1989, "Comparison of k - ϵ and Algebraic Reynolds Stress Models for Swirling Diffuser Flow," Int. J. Num. Methods in Fluids, Vol. 9, pp. 987-1009.

Armfield, S.W., Cho, N-H., Fletcher, C.A.J., 1990, "Prediction of Turbulence Quantities for Swirling Flow in Conical Diffusers," AIAA Journal, Vol. 28, pp. 453-460.

Aviation Week & Space Technology (AW&ST), 1998, "Future Fighter Mission Unchanged: Dominate the Air," pp. 38-42, August 3.

Baldwin, B.S., Barth, T.J., 1991, "A One-Equation Turbulence Transport Model for High Reynolds Number Wall-Bounded Flows," AIAA Paper 91-0610.

Baldwin, B.S., Lomax, H., 1978, "Thin-Layer Approximation and Algebraic Model for Separated Turbulent Flows," AIAA Paper 78-257.

Ball, W.H., 1984, "Tests of Wall Blowing Concepts for Diffuser Boundary Layer Control," AIAA Paper 84-1276.

Bansod, P., Bradshaw, P., 1972, "The Flow in S-shaped Ducts," Aeronautical Quarterly, pp. 131- 140.

Barberopoulos, A.A., Garry, K.P., 1998, "The Effect of Skewing on the Vorticity Produced by an Airjet Vortex Generator," The Aeronautical J., Paper No. 2275, pp. 171-177.

Bardina, J.E., Huang, P.G., Coakley, T.J., 1997, "Turbulence Modeling Validation, Testing, and Development," NASA TM-110446.

Barlow, R.S., Johnston, J.P., 1988, "Structure of a Boundary Layer on a Concave Surface," J. Fluid Mech. Vol. 191, pp.137-175.

Barret, R., Farokhi, S., 1993, "On the Aerodynamics and Performance of Active Ramp Vortex Generators," AIAA Paper 93-3447.

Barret, R., Farokhi, S., 1996, "Subsonic Aerodynamics and Performance of a Smart Vortex Generator System," J. Aircraft, Vol. 33, pp. 393-398.

Bender, E.E., Anderson, B.H., Yagle, P.J., 1999, "Vortex Generator Modeling for Navier-Stokes Codes," ASME Paper FEDSM99-6919.

Bingaman, D.C., 1992, "Performance Study for Inlet Installations," NASA Contractor Report CR-189714.

Boussinesq, J., 1877, "Théorie de l'Écoulement Tourbillant," Mem. Présentés par Divers Savants Acad. Sci. Inst. Fr., Vol. 23, pp. 46-50.

Bowditch, D.N., Coltrin, R.E., 1983, "A Survey of Inlet Engine Distortion Capability," NASA Technical Memorandum, TM-83421.

Bradshaw, P., 1973, "Effects of Streamline Curvature on Turbulent Flow," AGARD AG 169.

Bradshaw, P., 1987, "Turbulent Secondary Flows," Ann. Rev. Fluid. Mech., Annual Reviews Inc., pp. 53-74.

Brandt, A., 1977, "Multi-Level Adaptive Solutions to Boundary-Value Problems," Math. Comput. Vol. 31, pp. 333-390.

Brown, A.C., Nawrocki, H.F., Paley, P.N., 1968, "Subsonic Diffusers Designed Integrally with Vortex Generators," AIAA J. Aircraft, Vol. 5, pp. 221-229.

Bushnell, D.M., 1992, "Longitudinal Vortex Control - Techniques and Applications," Aeronautical Journal, pp. 293-312.

Chan, W.M., Buning, P.G., 1995, "Zipper Grids for Force and Moment Computation on Overset Grids," AIAA Paper 95-1681.

Chang, H.C., 1942, "Aufrollung Eines Zylindrischen Strales Durch Querwind," Doctoral Dissertation, Univ. of Gottingen, 1942. Also available in English as "The Roll-up of a Cylindrical Jet in a Cross Flow," translated and edited by Nagaraja, K.S. and Schrade Herbert O., USAF-ARL 73-0131, Sept. 1973.

Chen, H.C., Patel, V.C., 1988, "Near-Wall Turbulence Models for Complex Flows Including Separation," *AIAA Journal*, Vol. 26, pp. 641-648.

Chien, K-Y., 1982, "Prediction of Channel and Boundary Layer Flows with a Low-Reynolds-Number Turbulence Model," *AIAA J.*, Vol. 20, pp. 33-38.

Chiu, S.H., Roth, K.R., Margason, R. and Tso, J., 1993, "A Numerical Investigation of a Subsonic Jet in Crossflow," AGARD-CP-534, AGARD Fluid Dynamics Panel Symposium on 'Computation and Experimental Assessment of Jets in Cross Flow,' Winchester, U.K.

Chorin, A.J., 1967, "A Numerical Method for Solving Incompressible Viscous Flow Problems," *J. Comp. Phys.*, Vol. 2, pp. 12-26.

Cockrell, D.J., Diamond, M.J., Jones, G.D., 1965, "The Diffuser Inlet Flow Parameter," *J. Royal Aeronautical Society*, Vol. 69, pp. 350-352.

Coles, D.E., Hirst, E.A., 1969, "Computation of Turbulent Boundary Layers," *Proceedings, AFOSR-IFP Stanford Conference 1968*, Stanford University.

Compton, D.A., Johnston, J.P., 1992, "Streamwise Vortex Production by Pitched and Skewed Jets in a Turbulent Boundary Layer," *AIAA J.*, Vol. 30, pp. 640-647.

Cumptsy, N.A., 1989, "*Compressor Aerodynamics*," Longman.

Cutler, A.D., Johnston, J.P., 1981, "The Effects of Inlet Conditions on the Performance of Straight-Walled Diffusers at Low Subsonic Mach Numbers -- A Review," Report PD-26, Stanford University.

**DFRC, 1995, NASA Dryden Flight Research Center, Internet Address:
www.dfrc.nasa.gov.**

Dolan, F.X., Runstadler, P.W., 1973, "Pressure Recovery Performance of Conical Diffusers at High Subsonic Mach Numbers," NASA Contractor Report, NASA CR-2299.

Driver, D.M., Johnston, J.P., 1990, "Experimental Study of a Three-Dimensional Shear-Driven Turbulent Boundary Layer with Streamwise Adverse Pressure Gradient," NASA TM 102211.

**Dudek, J.C., 1996, "NPARC Validation--Subsonic Turbulent Diffusing Pipe Flow," The NPARC Alliance, Internet Address:
<http://info.arnold.af.mil/nparc/archive/model/subsdif/>**

Dudek, J.C., Georgiadis, N.J., Yoder, D.A., 1996, "Calculation of Turbulent Subsonic Diffuser Flows Using NPARC Navier-Stokes Code," AIAA Paper 96-0497.

Emmons, H.W., Pearson, C.E., Grant, H.P., 1955, "Compressor Surge and Stall Propagation," Trans. ASME, Vol. 79, pp. 455-469.

Findlay, M., 1998, "An Experimental and Numerical Investigation of Inclined Jets in a Crossflow," Ph. D. Thesis, University of British Columbia.

Fox R.W., Kline S.J., 1962, "Flow Regimes in Curved Subsonic Diffusers," ASME J. Basic Engineering, Vol. 84, pp. 303-316.

- Gal-Or, B., 1990, *Vectored Propulsion Supermaneuverability and Robot Aircraft*, Springer-Verlag.
- Gatski, T.B., Speziale, C.G., 1993, "On Explicit Algebraic Stress Models for Complex Turbulent Flows," ASME J. Fluid Mech., Vol. 254, pp. 59-78.
- Gerlach, C.R., Schroeder, E.C., 1969, "Study of Minimum Pressure Loss in High Velocity Duct Systems," Interim Technical Report No. 1, NASA Contractor Report CR-102499
- Gibb, J., Anderson, B.H., 1995, "Vortex Flow Control Applied to Aircraft Intake Ducts," High Lift and Separation Control Conference , University of Bath, UK.
- Gibson, M.M., Launder, B., 1978, "Ground Effects on Pressure Fluctuations in the Atmospheric Boundary Layer," ASME J. Fluid Mech., Vol. 86, p. 491.
- Greitzer, E.M., 1980, "Review- Axial Compressor Stall Phenomena," ASME J. Fluids Engineering, Vol. 102, pp. 134-151.
- Greitzer, E.M., 1981, "The Stability of Pumping Systems-The 1980 Freeman Scholar Lecture," ASME J. Fluids Engineering, Vol. 103, pp. 193-242.
- Hall, M., Murphy, B., Amarasinghe, S., Liao, S., Lam, M., 1995, "Interprocedural Analysis for Parallelization," Proc. 8th Int. Workshop on Languages and Compilers for Parallel Computing, Springer-Verlag.
- Hall, E.J., Topp, D.A., Heidegger N.J., McNulty, G.S., Weber, K.F., Delaney, R.A., 1996, "Task 7-Endwall Treatment Inlet Flow Distortion Analysis and Final Report," NASA Contractor Report CR-195468.

Hassan, I., Findlay, M., Salcudean, M., Gartshore, I., 1998, "Prediction of Film Cooling with Compound-Angle Injection Using Different Turbulence Models,"

Proceeding of CFD 98, CFD Society of Canada, pp.x-1 - x-6.

Heidegger, N.J., Hall, E.J., Delaney, R.A., 1999, "Follow-on Low Noise Fan Aerodynamic Study, Task 15-Final Report," NASA CR-1999-206599.

Hellsten, A., 1996, "Implementation of a One-Equation Turbulence Model into the FINFLO Flow Solver," Report No B-49 Series B, Helsinki University of Technology Laboratory of Aerodynamics, ISBN 951-22-3219-7.

Hellsten, A., 1998a, "Some Improvements in Menter's $k-\omega$ SST Turbulence Model," AIAA Paper 98-2554.

Hellsten, A., 1998b, "On the Solid-Wall Boundary Condition of ω in the $k-\omega$ -Type Turbulence Models," Report No B-50, Series B, Helsinki University of Technology, Finland.

Henry, F.S., Pearcey, H.H., 1994, "Numerical Model of Boundary-Layer Control Using Air-Jet Generated Vortices," AIAA J., Vol. 32, pp. 2415-2425.

Herbst, W.B., 1990, "X-31 At First Flight," *Flying Qualities*, AGARD CP-508, pp. 29-1-29-8.

Hoffmann, J.A., 1981, "Effects of Free-Stream Turbulence on Diffuser Performance," J. Fluids Engineering, Vol. 103, pp. 385-390.

Horlock, J.H., Lewis, R.I., 1961, "Shear Flows in Straight-Sided Nozzles and Diffusers," International J. of Mechanical Science, Vol. 2, pp. 251-266.

Hunt J.C.R., Abell C.J., Peterka J.A., Woo, H., 1978, "Kinematical Studies of the Flows Around Free or Surface-Mounted Obstacles; Applying Topology to Flow Visualization," *J. Fluid Mech.*, Vol. 86, pp. 179-200.

Jameson, A., Schmidt, W., Turkel, E., 1981, "Numerical Solution of the Euler Equations by Finite Volume Methods Using Runge Kutta Time Stepping Schemes," *AIAA Paper 81-1259*.

Jane's, *All The World's Aircraft* 1987, Jane's Publishing Inc., 1987-1988.

Jansson, L.S., Davidson, L., 1996, "Numerical Study of Effusion Cooling in a Double-Row Discrete-Hole Configuration Using a Low-Re Reynolds Stress Transport Model," *Engineering Turbulence Modelling and Experiments 3*, Edited by Rodi, W., and Bergeles, G., Elsevier Science, pp. 731-741.

Johnson, P.L., Johnston, J.P., 1989, "The Effects of Grid-Generated Turbulence on Flat and Concave Turbulent Boundary Layers," Report MD-53, Dept. Mech. Eng., Stanford University.

Johnston, J.P., Nishi, M., 1990, "Vortex Generator Jets-Means for Flow Separation Control," *AIAA J.*, Vol. 28, pp. 989-994.

Jones, W.P., Launder, B.E., 1972, "The Prediction of Laminarization with a Two Equation Model of Turbulence," *International J. of Heat & Mass Transfer*, Vol. 15, pp. 301-314.

Kaiser, K.F., McDonald A.T., 1980, "Effect of Wake-Type Nonuniform Inlet Velocity Profiles on First Appreciable Stall in Plane-Wall Diffusers," *ASME J. Fluids Engineering*, Vol. 102, pp. 283-289.

Kelso, R.M., Lim, T.T., Perry A.E., 1996, "An Experimental Study of Round Jets in Cross-Flow," *J. Fluid Mech.*, Vol. 306, pp. 111-144.

Khan, Z.U., 1999, "On the Dominant Vortex Created by a Pitched and Skewed Jet in Crossflow," Ph. D. Thesis, Stanford University.

Kim, S.-W., Benson, T.J., 1992, "Calculation of a Circular Jet in Crossflow with a Multiple-Time-Scale Turbulence Model," *Int. J. Heat Mass Transfer*, Vol. 35, No. 10, pp. 2357-2365.

Kim, S.-W., Benson, T.J., 1993, "Fluid Flow of a Row of Jets in Crossflow - A Numerical Study," *AIAA Journal*, Vol. 31, pp. 806-811.

Kim, W.J., Patel, V.C., 1993, "An Experimental Study of Boundary Layer Flow in a Curved Rectangular Duct," *ASME Fluids Engineering Conference, FED* Vol. 146, pp. 13-28.

Klein, A., 1981, "Review: Turbulent Developing Pipe Flow," *ASME J. Fluids Engineering*, Vol. 103, pp. 250-257

Kline, S.J., Abbot, D.E., Fox, R.W., 1959, "Optimum Design of Straight-Walled Diffusers," *ASME J. Basic Engineering*, Vol. 81, pp. 321-329.

Kline, S.J., Cantwell, B.J., Lilley, G.M., Editors, 1982, *Proceedings, AFOSR-HTTM Stanford Conference on Complex Turbulent Flows, 1980-1981*, Stanford University.

Kolmogorov, A.N., 1942, "Equations of Turbulent Motion of an Incompressible Fluid," *Izvestia Academy of Sciences, USSR, Physics*, Vol. 6, pp. 56-58.

Kunik, W.G., 1986, "Application of a Computational Model for Vortex Generators in Subsonic Internal Flows," AIAA Paper 86-1458.

Launder, B.E., Spalding, D.B., 1974, "The Numerical Computation of Turbulent Flows," Computer Methods in Applied Mechanics and Engineering, North Holland, Vol. 3, pp. 269-289.

Launder, B.E., Pridden, C.H., Sharma, B., 1977, "The Calculation of Turbulent Boundary Layers on Spinning and Curved Surfaces," J. Fluids Eng., Vol. 98, pp. 231-239.

Lee, C.C., Boedicker, C., 1985, "Subsonic Diffuser Design and Performance for Advanced Fighter Aircraft," AIAA Paper 85-3073.

Lee, K.M., Yu, S.C.M., 1994, "Computational Studies of Flows in the RAE 2129 S-shaped Diffusing Duct," AIAA Paper 94-0658.

Legendre, R., 1956, "Séparation de l'écoulement Laminaire Tridimensionnel," Rech. Aéronaut., Vol. 54, pp. 3-8.

Livesey, J.L., Turner, J.T., 1964, "The Effect of Velocity Profile Decay on Shear Flow in Diffusers," International J. Mechanical Science, Vol. 6, pp. 371-379.

Luo, J., Lakshminarayana, B., 1996, "Analysis of Streamline Curvature Effects on Wall-Bounded Turbulent Flows," *Engineering Turbulence Modelling and Experiments 3*, Edited by Rodi, W., and Bergeles, G., Elsevier Science, pp. 59-69.

Margason, R.J., 1993, "Fifty Years of Jet in Cross Flow Research," AGARD-CP-534, AGARD Fluid Dynamics Panel Symposium on 'Computation and Experimental Assessment of Jets in Cross Flow,' Winchester, U.K.

Margason, R.J., 1969, "Analytic Description of Jet-Wake Cross Sections for a Jet Normal to a Subsonic Free stream," NASA SP-218, pp. 131-140.

Marquis, A.J., Ong, L.Y., 1995, "The Computation of Flow in a RAE M2129 Diffuser Using Low Reynolds Number k - ϵ Turbulence Models," Proc. Ninth Int. Conf. on Num. Methods in Laminar and Turbulent Flows, pp. 841-851.

Masuda, S., Ariga I., Watanabe, I., 1971, "On the Behavior of Uniform Shear Flow in Diffusers and Its Effects on Diffuser Performance," ASME J. of Engineering for Power, Vol. 93, pp. 377-385.

Masuda, S., Ariga I., Watanabe, I., 1975, "Uniform Shear Flow Within Rectangular Parallel-Walled Diffusers," ASME Joint Fluids Engineering and Lubrication Conference, 75-FE-34.

Mavriplis, D.J., Martinelli, L., 1994, "Multigrid Solution of Compressible Turbulent Flow on Unstructured Meshes Using a Two-Equation Model," Int. J. Num. Methods in Fluids, Vol. 18, pp. 887-914.

Mayer, D.W., Anderson, B.H., Johnson, T.A., 1998, "3D Subsonic Diffuser Design and Analysis," AIAA Paper 98-3418.

McDonald A.T., Fox, R.W., Van Dewoestine, R.V., 1971, "Effects of Swirling Flow on Pressure Recovery in Conical Diffusers," AIAA Journal, Vol. 9, pp. 2014-2018

Menter, F.R., 1992a, "Influence of Freestream Values on k - ω Turbulence Model Predictions," AIAA J., Vol. 30, pp. 1657-1659.

Menter, F.R., 1992b, "Performance of Popular Turbulence Models for Attached and Separated Adverse Pressure Gradient Flows," AIAA J., Vol. 30, pp. 2066-2072.

Menter, F.R., 1994, "Eddy Viscosity Transport Models and their Relation to the k - ϵ Model," *NASA Technical Memorandum TM-108854*.

Menter, F.R., 1996a, "A Comparison of Some Recent Eddy-Viscosity Turbulence Models," *ASME J. of Fluids Engineering*, Vol. 118, pp. 514-519.

Menter, F.R., 1996b, "On the Connection Between One- and Two- Equation Models of Turbulence," *Engineering Turbulence Modelling and Experiments 3*, Edited by Rodi, W., and Bergeles, G., Elsevier Science, pp. 131-140.

Monson, D.J., Seegmiller, H.L., 1992, "An Experimental Investigation of Subsonic Flow in a Two-Dimensional U-Duct," *NASA Technical Memorandum TM-103931*.

Moore, C.A., Kline S.J., 1955, "Some Effects of Vanes and of Turbulence on Two-Dimensional Wide-Angle Subsonic Diffusers," *NACA TN 4080*.

Muck, K.C., Hoffmann, P.H., Bradshaw, P., 1985, "The Effect of Convex Curvature on Turbulent Boundary Layers," *ASME J. Fluid Mech.*, Vol. 161, p. 347.

Nikjoo, M., Mongia, H., 1998, "Study of Non-Linear k - ϵ Model for Turbulent Swirling Flows," *AIAA Paper 98-3984*.

Norris, L.H., Reynolds, W.C., 1975, "Turbulent Channel Flow with a Moving Wavy Boundary," Rept. No. FM-10, Stanford University, Dept. Mech. Eng.

Panton, R.L., 1996, *Incompressible Flow*, 2nd ed., Wiley & Sons.

Patel, V.C., Rodi, W., Scheuerer, G., 1985, "Turbulence Models for Near-Wall and Low-Reynolds Number Flows: A Review," *AIAA Journal*, Vol. 23, pp. 1308-1319.

Pearcey, H.H., Lachmann, G.V., (editor), 1961, "Shock-induced Separation and its Prevention," *Boundary Layer and Flow Control, It's Principles and Application*, Vol. 2, Pergamon.

Pearcey, H.H., Rao, K., Sykes, D.M., 1993, "Inclined Air-Jets Used as Vortex Generators to Suppress Shock-Induced Separation," AGARD-CP-534, AGARD Fluid Dynamics Panel Symposium on 'Computation and Experimental Assessment of Jets in Cross Flow,' Winchester, U.K.

Perry A.E., Chong M.S., 1987, "A Description of Eddying Motions and Flow Patterns Using Critical-Point Concepts," *Ann. Rev. Fluid Mech.*, Vol. 19, pp. 125-155.

Prandtl, L., 1925, "Über die ausgebildete Turbulenz," *ZAMM*, Vol. 5, pp. 136-139

Press, W.H., Teukolsky, S.A., Vetterling, W.T., Flannery, B.P., 1992, *Numerical Recipes in C; the art of scientific computing*, 2nd ed., Cambridge University Press.

Pulliam, T.H., 1985, *Von Karman Institute, Lecture Series* 1985-03.

Pulliam, T.H., 1993, "Time Accuracy and the Use of Implicit Methods," AIAA Paper 93-3360-CP.

Rae, W.H., Pope, A., 1984, *Low-Speed Wind Tunnel Testing*, 2nd ed., Wiley-Interscience.

Reneau, L.R., Johnston, J.P., Kline S.J., 1964, "Performance and Design of Straight Two-Dimensional Diffusers," Report PD-8, Stanford University.

Rizzi, A., Eriksson, L., 1985, "Computation of Inviscid Incompressible Flow With Rotation," *J. Fluid Mech.*, Vol. 153, pp. 275-312.

Rodi, W., 1976, "A New Algebraic Relation for Calculating Reynolds Stresses," *ZAMM*, Vol. 56, p. 219.

Rodi, W., Celik, I., Demuren, A.O., Scheuerer, G., Shirani, I., Leschziner, M.A., and Rastogi, A.K., 1982, Proceedings, *AFOSR-HTTM Stanford Conference on Complex Turbulent Flows*, 1980-1981, Stanford University, Kline, S.J., Cantwell, B.J., Lilley, G.M., Editors.

Rodi, W., Scheuerer, G., 1986, "Scrutinizing the k - ϵ Turbulence Model Under Adverse Pressure Gradient Conditions," *ASME J. Fluids Engineering*, Vol. 108, pp. 174-179.

Rodi, W., 1988, "Recent Developments in Turbulence Modeling," Proc. 3rd Int. Symp. on Refined Flow Modeling and Turbulence Measurements, Y. Iwasa, N. Tamai, and A. Wada, eds., Tokyo, Japan.

Rosenhead, L., 1932, "The Formation of Vortices From a Surface of Discontinuity," *Proceedings of Royal Society of London, Series A*, Vol. 134.

Rudy, D.H., Strikwerda, J.C., 1980, "A Nonreflecting Outflow Boundary Condition for Subsonic Navier-Stokes Calculations" *J. Comp. Phys.*, Vol. 36 pp. 55-70.

Rumsey, C.L., Gatski, T.B., Morrison, J.H., 1999, "Turbulence Model Predictions of Extra-Strain Rate Effects in Strongly Curved Flows," *AIAA Paper 99-0157*.

Sandborn, V.A., 1989, "Measurement of Terms and Parameters in Turbulent Models," *NASA Contractor Report CR-183993*

Schneider, G.E., Zedan, M., 1981, "A Modified Strongly Implicit Procedure for the Numerical Solution of Field Problems," *J. Numerical Heat Transfer*, Vol. 4, pp. 1-19

Schubauer, G.B., Spangenberg, W.G., 1960, "Forced Mixing in Boundary Layers," J. Fluid Mechanics, Vol. 8, Part I.

Seddon, J., Goldsmith, E.L., 1989, *Intake Aerodynamics*, 2nd ed., AIAA Education Series.

Selby, G.V., Lin, J.C., Howard, F.G., 1992, "Control of Low-Speed Turbulent Separated Flow Using Jet Vortex Generators," Experiments in Fluids, Vol. 12, pp 394-400.

Senoo, Y., Nishi, M., 1974, "Improvement of the Performance of Conical Diffusers by Vortex Generators," ASME J. Fluids Engineering, pp. 4-10.

Senseney, M.B., Buter T.A., Bowersox, R.D.W., 1995, "Flow Structure and Performance Characterization of an Offset Diffuser With and Without Vortex Generator Jets," AIAA Paper 95-2884.

Shih, T-H., 1998, "Turbulence Modeling Developments at ICOMP," AIAA Paper 98-3243.

Shih, T-H., Liou, W.W., Shabbir, A., Yang, Z., Zhu, J., 1995, "A New k - ϵ Eddy-Viscosity Model for High Reynolds Number Turbulent Flows," Computers & Fluids, Vol. 24, pp. 227-238.

So, K.L., 1967, "Vortex Phenomena in a Conical Diffuser," AIAA Journal, Vol. 5, pp. 1072-1078.

So, R.M.C., Mellor, G., 1973, "Experiment on Convex Curvature Effects in Turbulent Boundary Layers," ASME J. Fluid Mech., Vol. 60, p. 43.

Sovran, G., Klomp, E.D., 1967, "Experimentally Determined Optimum Geometries for Rectilinear Diffusers with Rectangular, Conical or Annular Cross-Section," *Fluid Mechanics of Internal Flow*, Proc. Symposium on the Fluid Mechanics of Internal Flow, General Motors Research Labs, Elsevier Publishing, pp. 270-319.

Spalart, P.R., Allmaras, S.R., 1992, "A One Equation Turbulence Model for Aerodynamic Flows," AIAA Paper 92-0439.

Spalart, P.R., Allmaras, S.R., 1994, "A One Equation Turbulence Model for Aerodynamic Flows," *La Recherche Aerospatiale*, No. 1, pp. 5-21.

Stevens, S.J., Williams, G.J., 1980, "The Influence of Inlet Conditions on the Performance of Annular Diffusers," *ASME J. Fluids Engineering*, Vol. 102, pp. 1357-1363.

Stone, H.L., 1968, "Iterative Solution of Implicit Approximations of Multidimensional Partial Differential Equations," *SIAM J. Numer. Anal.*, Vol. 5, pp. 530-558.

Swanson, R.C., Radespiel, R., 1991, "Cell Centered and Cell Vertex Multigrid Schemes for the Navier-Stokes Equations," *AIAA Journal*, Vol. 29, pp. 697-703.

Tamrat, B.F., 1988, "Fighter Aircraft Agility Assessment Concepts and Their Implications on Future Agile Fighter Design," AIAA Paper 88-4400.

Taylor, H.D., 1948, "Design Criteria for and Applications of the Vortex Mixing Principle," United Aircraft Corp. Report No. M-15038-1.

Tobak, M., Peake, D.J., 1981, "Topology of Three-Dimensional Separated Flows," NASA Technical Memorandum TM-81294.

van Driest, E.R., 1956, "On Turbulent Flow Near a Wall," *J. Aeronautical Sciences*, Vol. 23, pp. 1007-1011.

Vinokur, M., 1989, "An Analysis of Finite-Difference and Finite Volume Formulations of Conservation Laws," *J. Comp. Phys.*, Vol. 81, pp.1-52.

Wallis, R.A., 1952 "The Use of Air Jets for Boundary Layer Control," A.R.L. Aero Note 110, Australia.

Wallis, R.A., 1960, "A Preliminary Note on a Modified Type of Air Jet for Boundary Layer Control," A.R.C. Technical Report C.P. 513.

Wendt, B.J., Reichert, B.A., 1996, "Vortex Ingestion in a Diffusing S-Duct Inlet," *AIAA J. Aircraft*, Vol. 33, No. 1, pp. 149-154.

Westwater, F.L., 1935, "Rolling Up of the Surface of Discontinuity Behind an Aerofoil of Finite Span," British ARC R and M Report No. 1692.

Wiegardt, K., Tillmann, 1969, "Wiegardt, Flat Plate Flow," *Computation of Turbulent Boundary Layers*, Proceedings, *AFOSR-IFP Stanford Conference 1968*, W., Coles, D.E., Hirst, E.A., eds., Stanford University, pp. 98-123.

White, F.M., 1991, "*Viscous Fluid Flow*" 2nd edition, McGraw-Hill.

Wilcox, D.C., 1988, "Reassessment of the Scale Determining Equation for Advanced Turbulence Models," *AIAA J.*, Vol. 26, pp. 1299-1310.

Wilcox, D.C., 1993, "Application of Low Reynolds Number Two-Equation Turbulence Models to High Reynolds Number Flows," in *Near-Wall Turbulent Flows*, Edited by So, R.M.C., Speziale, C.G., Launder, B.E., Elsevier Science, pp. 155-165.

Wilcox, D.C., 1994, *Turbulence Modeling for CFD*, DCW Industries.

Wilcox, D.C., 1998, *Turbulence Modeling for CFD*, 2nd ed. DCW Industries.

Williams, D.D., Surber, L.E., 1993, "Intake/Engine Compatibility," *Practical Intake Aerodynamic Design*, edited by E.L. Goldsmith and J. Seddon, AIAA Education Series, pp. 21-71.

Winternitz, F.A.L., Ramsay, W.J., 1957, "Effects of Inlet Boundary Layer on Pressure Recovery, Energy Conversion and Losses in Conical Diffusers," *J. Aeronautical Society*, Vol. 61, pp. 116-124.

Wolf, S., Johnston, J.P., 1969, "Effects of Nonuniform Inlet Velocity Profiles on Flow Regimes and Performance in Two-Dimensional Diffusers," *ASME J. of Basic Engineering*, Vol. 91, pp. 462-474.

Wolfshtein, M., 1969, "The Velocity and Temperature Distribution in One-Dimensional Flow with Turbulence Augmentation and Pressure Gradient," *Int. J. Heat Mass Transfer*, Vol. 12, pp. 301-318.

Yakhot, V., Orszag, S.A., 1986, "Renormalization Group Analysis of Turbulence. I. Basic Theory," *J. Scientific Computing*, Vol. 1.

Yaras, M.I., 1996, "Effects of Inlet Conditions on the Flow in a Fishtail Curved Diffuser With Strong Curvature," *ASME J. Fluids Engineering*, Vol. 118, pp. 772-778

Yaras, M.I., 1999, "Flow Measurements in a Fishtail Diffuser With Strong Curvature," *ASME J. Fluids Engineering*, Vol. 121, pp. 410-417.

Yoder, D.A., Georgiadis, N.J., 1999, "Implementation and Validation of the Chien k - ϵ Turbulence Model in the Wind Navier-Stokes Code," NASA TM-1999-209080 (also AIAA-99-0745).

Zhang, H.L., Zhang, X., Collins, M.W., 1996, "Near Field Flow Calculation of Jet Generated Vortices in a Turbulent Boundary Layer," *Engineering Turbulence Modelling and Experiments 3*, Edited by Rodi, W., and Bergeles, G., Elsevier Science, pp. 339-348.

Zhang, X., 1993a, "Computational Analysis of Co- and Contrarotating Longitudinal Vortices in a Turbulent Boundary Layer," AIAA Paper 93-3035.

Zhang, X., 1993b, "Interaction Between a Turbulent Boundary Layer and Elliptic and Rectangular Jets," *Engineering Turbulence Modelling and Experiments 2*, Edited by Rodi, W., and Martelli, F., Elsevier Science, pp. 251-260.

Zhang, X., Collins, M.W., 1993, "Flow and Heat Transfer in a Turbulent Boundary Layer Through Skewed and Pitched Jets," AIAA J., Vol. 31, pp. 1590-1599.

Zhang, X., 1995, "Co- and Contrarotating Streamwise Vortices in a Turbulent Boundary Layer," AIAA J. Aircraft, Vol. 32, No. 5, pp. 1095-1101.

Zhang, X., 1998, "Turbulence Measurements of a Longitudinal Vortex Generated by an Inclined Jet in a Turbulent Boundary Layer," ASME J. Fluids Engineering, Vol. 120, pp. 765-771.

Zhang, X., Rona, A., 1999, "Measurements of Contra-Rotating Vortices in a Turbulent Boundary Layer," AIAA Paper 99-0555.

Zheng, X., Liao, C., Liu, C., Sung, C.H., Huang, T.T., 1997, "Multigrid Computations of Incompressible Flows Using Two-Equation Turbulence Models: Part I- Numerical Method," J. Fluids Eng., Vol. 119, pp. 893-899.

A.2 Pseudo Computer Codes for Turbulence Models

Implementations of the $k-\varepsilon$, $k-\omega$, SST and SA turbulence models are given in pseudo-code form in this section. Since the models have been coded into FS by Dr. M.I. Yaras, development of these pseudo-codes has served as part of the training of Mr. A.D. Grosvenor. The pseudo-codes have been included in the present report to serve as guidelines for future implementation of these turbulence models by other researchers.

In the interest of generality, no formal syntax specific to any programming language has been given. Rather, implementation has been provided in step-by-step structured English, in a form that can be readily translated into computer code. The pseudo-code only describes turbulence model implementation and not the mean-flow algorithm. However, since the mean flow variables are required in the turbulence calculations, reference is made to mean-flow computations where appropriate. To be consistent with the FS code, the pseudo-code is based on the solution of the nondimensional form of the governing equations.

Pseudo-code Nomenclature

Note: The following variable definitions correspond to normalised quantities.

A_mu	A_μ
A_tdr	A_τ
A_x	x-projection of $A_{c.v.}$
A_y	y-projection of $A_{c.v.}$
A_z	z-projection of $A_{c.v.}$
beta_wall	β_{wall}
C_b1	C_{b1}
C_b2	C_{b2}
C_eps1	$C_{\epsilon1}$
C_eps2	$C_{\epsilon2}$
C_mu	C_μ
C_omega1	$C_{\omega1}$
C_omega2	$C_{\omega2}$
C_omega3	$C_{\omega3}$
C_v1	C_{v1}
C_w1	C_{w1}
C_w2	C_{w2}
C_w3	C_{w3}
d_nu_sa_x	$\partial \tilde{\nu} / \partial x$
d_tdr_n	gradient of turbulence dissipation rate normal to surface at the wall
d_tdr_x	gradient of turbulence dissipation rate in the x-direction
d_tdr_y	gradient of turbulence dissipation rate in the y-direction
d_tdr_z	gradient of turbulence dissipation rate in the z-direction
d_tke_x	gradient of turbulence kinetic energy in the x-direction
d_tke_y	gradient of turbulence kinetic energy in the y-direction
d_tke_z	gradient of turbulence kinetic energy in the z-direction

delta_ij	δ_{ij}
D_a	D_a
Dest	destruction term in the model equation
E	E
E_I	E (convective)
E_V	E (diffusive)
f_2	f_2
f_mu	f_μ
f_v1	f_{v1}
f_v2	f_{v2}
f_w	f_w
F	F
Fd2	second diffusion term in the <i>SA</i> model.
F_I	F (convective)
F_V	F (diffusive)
g	g
G	G
G_I	G (convective)
G_V	G (diffusive)
H	H
H_Q	$\partial H / \partial Q$
I	identity matrix
kappa	κ
L	length
mu	μ
mu_t	μ_t
nu	ν
nu_sa	$\tilde{\nu}$
Prod	production term

Q	Q
r	r (component of SA function f_w)
R	R (residual)
Re_ref	Re_{ref}
Re_t	Re_t
Re_y	Re_y
rho	ρ
sigma	σ
sigma_eps	σ_ϵ
sigma_omega	σ_ω
sigma_tke	σ_k
t	t
tau_ij	τ_{ij}
tau_wall	τ_w
tdr	ϵ or ω depending on model used.
tke	k
U	U
U_tau	U_τ
V	V
Vol_c	$V_{c.v.}$
vort	Ω
vort_sa	$\tilde{\Omega}$
W	W
x	x
y	y
y_l	y_l
y_lplus	y_l^+
y_wall	y_{wall}
z	z

A.2.1 The k - ϵ Turbulence Model

The following pseudo-code applies to two different variants of Launder and Spalding's (1974) standard k - ϵ model. Calculation of turbulence quantities near the wall is made using either Chien's (1982) low- Re model, or one of Chen and Patel's (1988) and Rodi et al.'s (1988) two-layer models that employ algebraic relations for ϵ near the wall.

STEP 1 Initialisation of Variables

- Initialise turbulence quantities based on user-specified values.
All variables are specified at time t_j , either as initial values or from a previous solution.
- Control-volume area and volume calculations are assumed to have been already completed.

- Calculate reference Reynolds number from user-specified values:

$$Re_{ref} = \rho_{ref} * U_{ref} * L_{ref} / \mu_{ref}$$

- Set the following variables to initial values:

```
tke = tke_init
tdr = tdr_init
U = U_init
V = V_init
W = W_init
```

- Set lower cutoff values for the turbulence quantities :

```
tke = tke_min
tdr = tdr_min
```

STEP 2 Runge-Kutta Integration Stage 1

The application of preconditioning and multigrid convergence acceleration have been excluded for brevity. The eddy-viscosity value used in the present iteration is obtained from the previous integration cycle or from initial values.

● Evaluate turbulent fluxes:

Volume integrals containing derivatives are converted to surface integrals by applying the Gauss divergence theorem: $\oint_V (\nabla \cdot \phi) dV = \oint_S \phi \cdot dS$

(1) Convective fluxes: $E^{i'} = \begin{Bmatrix} \rho \dot{U} k \\ \rho \dot{U} \varepsilon \end{Bmatrix}, F^{i'} = \begin{Bmatrix} \rho \dot{V} k \\ \rho \dot{V} \varepsilon \end{Bmatrix}, G^{i'} = \begin{Bmatrix} \rho \dot{W} k \\ \rho \dot{W} \varepsilon \end{Bmatrix}$

$$E_I_tke = \rho \cdot U \cdot tke$$

$$E_I_tdr = \rho \cdot U \cdot tdr$$

$$F_I_tke = \rho \cdot V \cdot tke$$

$$F_I_tdr = \rho \cdot V \cdot tdr$$

$$G_I_tke = \rho \cdot W \cdot tke$$

$$G_I_tdr = \rho \cdot W \cdot tdr$$

(2) Diffusive fluxes:

$$E^{v'} = \begin{Bmatrix} \frac{1}{Re_{\tau}} \left(\mu + \frac{\mu_t}{\sigma_t} \right) \frac{\partial k}{\partial x} \\ \frac{1}{Re_{\tau}} \left(\mu + \frac{\mu_t}{\sigma_t} \right) \frac{\partial \varepsilon}{\partial x} \end{Bmatrix}, F^{v'} = \begin{Bmatrix} \frac{1}{Re_{\tau}} \left(\mu + \frac{\mu_t}{\sigma_t} \right) \frac{\partial k}{\partial y} \\ \frac{1}{Re_{\tau}} \left(\mu + \frac{\mu_t}{\sigma_t} \right) \frac{\partial \varepsilon}{\partial y} \end{Bmatrix}, G^{v'} = \begin{Bmatrix} \frac{1}{Re_{\tau}} \left(\mu + \frac{\mu_t}{\sigma_t} \right) \frac{\partial k}{\partial z} \\ \frac{1}{Re_{\tau}} \left(\mu + \frac{\mu_t}{\sigma_t} \right) \frac{\partial \varepsilon}{\partial z} \end{Bmatrix}$$

$$E_V_tke = (\mu + \mu_t / \sigma_{tke}) \cdot d_tke_x / Re_ref$$

$$E_V_tdr = (\mu + \mu_t / \sigma_{eps}) \cdot d_tdr_x / Re_ref$$

$$F_V_tke = (\mu + \mu_t / \sigma_{tke}) \cdot d_tke_y / Re_ref$$

$$F_V_tdr = (\mu + \mu_t / \sigma_{eps}) \cdot d_tdr_y / Re_ref$$

$$G_V_tke = (\mu + \mu_t / \sigma_{tke}) \cdot d_tke_z / Re_ref$$

$$G_V_tdr = (\mu + \mu_t / \sigma_{eps}) \cdot d_tdr_z / Re_ref$$

(3) Total fluxes:

$$E_tke = E_I_tke - E_V_tke,$$

$$F_tke = F_I_tke - F_V_tke,$$

$$G_tke = G_I_tke - G_V_tke,$$

$$E_tdr = E_I_tdr - E_V_tdr$$

$$F_tdr = F_I_tdr - F_V_tdr$$

$$G_tdr = G_I_tdr - G_V_tdr$$

● **Source Term Calculation**

$$H_k^* = \frac{1}{Re_{ref}} \tau_{ij}^* \frac{\partial U_i^*}{\partial x_j} - Re_{ref} \rho^* \varepsilon^*, \quad H_\varepsilon^* = C_{\varepsilon 1} \frac{1}{Re_{ref}} \frac{\varepsilon^*}{k^*} \tau_{ij}^* \frac{\partial U_i^*}{\partial x_j} - C_{\varepsilon 2} Re_{ref} \rho^* \frac{\varepsilon^{*2}}{k^*}$$

(1) **Production Terms:**

- **Reynolds Stress Tensor Calculation:**

$$\tau_{ij}^* = \mu_t^* \left[\left(\frac{\partial U_i^*}{\partial x_j} + \frac{\partial U_j^*}{\partial x_i} \right) - \frac{2}{3} \frac{\partial U_k^*}{\partial x_k} \delta_{ij} \right] - \frac{2}{3} \rho^* k^* \delta_{ij}$$

The contribution of each stress tensor component to the total production of turbulence kinetic energy is calculated.

e.g.,

$$Prod_xy_tke = \tau_{xy}^* \cdot d_U_y / Re_ref$$

$$Prod_xy_tdr = \tau_{xy}^* \cdot d_U_y \cdot (tdr / tke) \cdot C_eps1 / Re_ref$$

(2) **Destruction Terms:**

$$Dest_tke = \rho^* \cdot tdr \cdot Re_ref$$

$$Dest_tdr = \rho^* \cdot (tdr^2 / tke) \cdot C_eps2 \cdot Re_ref$$

(3) **Source Terms:**

$$H_tke = Prod_tke - Dest_tke$$

$$H_tdr = Prod_tdr - Dest_tdr$$

(4) **Source Term Linearisation Matrix:**

$$\frac{\partial H^*}{\partial Q^*} = \begin{bmatrix} -2 Re_{ref} \left\{ \frac{C_\mu \rho^*}{\mu_t^*} \right\} k^* & 0 \\ 0 & -\frac{3}{2} C_{\varepsilon 2} Re_{ref} \left\{ \frac{C_\mu \rho^*}{\mu_t^*} \right\}^{\frac{1}{2}} \varepsilon^{*\frac{1}{2}} \end{bmatrix}$$

$$d_H_Q_tke = -2 \cdot Re_ref \cdot C_mu \cdot \rho^* \cdot tke / \mu_t$$

$$d_H_Q_tdr = -(3/2) \cdot C_eps2 \cdot Re_ref \cdot (C_mu \cdot \rho^* / \mu_t)^{\frac{1}{2}} \cdot tdr^{\frac{1}{2}}$$

Near-wall Treatment with Algebraic Relation for ε (Chen and Patel, 1988):

Repeat flux and source term evaluation with the following changes:

$$\varepsilon^* = \frac{1}{Re_{ref}} \left(\frac{k^*}{\ell_t^*} \right), \quad \ell_t^* = \kappa C_\mu^{-0.75} y^* \left[1 - e^{-Re_y / A_\varepsilon} \right]$$

$$Re_y = Re_{ref} * \rho * \nu^{1/2} * y_{wall} / \mu$$

if: $Re_y < 200$

$$L_{tdr} = \kappa * C_\mu^{-0.75} * (y_{wall}) * (1 - e^{-Re_y / A_{tdr}})$$

$$L_\mu = \kappa * C_\mu^{-0.75} * (y_{wall}) * (1 - e^{-Re_y / A_\mu})$$

Near-wall treatment proposed by Rodi et al. is the same as the method of Chen and Patel except the ℓ_t function.

if: $\kappa * C_\mu^{-0.75} * y / (1 + 5.3 / Re_y) < 0.95$

$$L_{tdr} = \kappa * C_\mu^{-0.75} * y_{wall} / (L_{ref} * (1 + 5.3 / Re_y))$$

Near-wall treatment of Chien (1982)

Repeat flux and source term evaluation with the following changes:

- Redefine ε to include finite wall value:
 $tdr = tdr + 2 * \mu * \nu / (\rho * y_{wall}^2 * Re_{ref}^2)$

- Source term additions:

$$H_k^* = \frac{1}{Re_{ref}} \tau_{ij}^* \frac{\partial U_i^*}{\partial x_j} - Re_{ref} \rho^* \varepsilon^*,$$

$$H_\varepsilon^* = C_{\varepsilon 1} \frac{1}{Re_{ref}} \frac{\tilde{\varepsilon}^*}{k^*} \tau_{ij}^* \frac{\partial U_i^*}{\partial x_j} - C_{\varepsilon 2} f_2 Re_{ref} \rho^* \frac{\tilde{\varepsilon}^{*2}}{k^*} + \frac{1}{Re_{ref}} \rho^* E^*$$

$$Re_t = (1 / C_\mu) * (\mu_t / \mu)$$

$$y^* = \rho * U_{tau} * y_{wall} / \mu$$

$$E_C = -2 * (\mu / \rho) * tdr * e^{(-y^* / 2)} / y_{wall}^2$$

$$f_2 = 1 - 0.22 * e^{-(Re_t / 6)} * (Re_t / 6)$$

$$f_\mu = 1 - e^{-0.0115 y^*}$$

- **Boundary Conditions and Lower Limits:**

- Entire domain:

if: turbulence quantity < minimum setting:

$$\text{tke} = \text{tke_min}$$

$$\text{tdr} = \text{tdr_min}$$

Specified BC's for k^ and ε^* are enforced at inflow, outflow, wall boundaries, etc.*

- Inflow Boundary:

$$\text{tke} = \text{tke_inflow}$$

$$\text{tdr} = \text{tdr_inflow}$$

- Outflow Boundary:

Values at the outlet are obtained by a first-order extrapolation from the interior of the domain.

$$\text{tke} = \text{tke_upstream}$$

$$\text{tdr} = \text{tdr_upstream}$$

- Wall Boundary:

$$\text{tke} = 0$$

$$\text{tdr} = 0 \text{ (if Chien's near-wall treatment is used)}$$

$$\text{d_tdr_d_n} = 0 \text{ (if algebraic relation for } \varepsilon \text{ is used near walls)}$$

- **Evaluate Residuals**

- Update Solution Vectors:
$$\bar{Q}^{(1)} = \bar{Q}^{(0)} - \frac{\alpha_l R^{(0)}}{V \cdot \left[\frac{I}{\Delta t} - \frac{\partial H}{\partial Q} \right]^*}$$

$$R_tke = E_tke * A_x + F_tke * A_y + G_tke * A_z - H_tke * Vol_c - D * Vol_c$$

$$R_tdr = E_tdr * A_x + F_tdr * A_y + G_tdr * A_z - H_tdr * Vol_c - D * Vol_c$$

$$Q_tke = Q_tke - \alpha_l * R_tke / ((I / \Delta t - H_tke_Q) * Vol_c)$$

$$Q_tdr = Q_tdr - \alpha_l * R_tdr / ((I / \Delta t - H_tdr_Q) * Vol_c)$$

$$\text{tke} = Q_tke / \rho$$

$$\text{tdr} = Q_tdr / \rho$$

Calculate eddy-viscosity

If Chien's model is used:

$$\mu_t = f_{\mu} * C_{\mu} * \rho * k\epsilon^2 / \tau_{dr}$$

If the two-layer approach is used:

$$\mu_t = Re_{ref} * \rho * C_{\mu} * k\epsilon^{1/2} * L_{\mu}$$

STEP 3 Runge-Kutta Integration Stage 2

Same as stage 1 in STEP 2

STEP 4 Smooth the residuals implicitly using ADI or MSIP.

A.2.2 The k - ω Turbulence Model

The following pseudo-code applies to Wilcox's (1988) k - ω model.

STEP 1 Initialisation of Variables

- Initialise turbulence quantities based on user-specified values.
All variables are specified at time t , either as initial values or previous solution.
- Control volume area and volume calculations are assumed completed.

- Calculate reference Reynolds number from user-specified values:

$$Re_ref = \rho_ref * U_ref * L_ref / \mu_ref$$

- Set following variables to initial values:

$$\begin{aligned} tke &= tke_init \\ tdr &= tdr_init \\ U &= U_init \\ V &= V_init \\ W &= W_init \end{aligned}$$

- Set lower cutoff values for the turbulence quantities:

$$\begin{aligned} tke &= tke_min \\ tdr &= tdr_min \end{aligned}$$

STEP 2 Runge-Kutta Integration Stage 1

The application of preconditioning and multigrid convergence acceleration have been excluded for brevity. The eddy-viscosity value used in the present iteration is obtained from the previous integration cycle or from initial values.

- Evaluate turbulent fluxes:

Volume integrals containing derivatives are converted to surface integrals by applying the Gauss divergence theorem: $\oint_V (\nabla \cdot \phi) dV = \oint_S \phi \cdot dS$

(1) Convective fluxes: $E' = \begin{Bmatrix} \rho U k \\ \rho U \omega \end{Bmatrix}, F' = \begin{Bmatrix} \rho V k \\ \rho V \omega \end{Bmatrix}, G' = \begin{Bmatrix} \rho W k \\ \rho W \omega \end{Bmatrix}$

$$E_I_tke = \rho * U * tke$$

$$E_I_tdr = \rho * U * tdr$$

$$F_I_tke = \rho * V * tke$$

$$F_I_tdr = \rho * V * tdr$$

$$G_I_tke = \rho * W * tke$$

$$G_I_tdr = \rho * W * tdr$$

(2) Diffusive fluxes:

$$E^{v*} = \begin{Bmatrix} \frac{1}{Re_{ref}} \left(\mu + \mu_t / \sigma_k \right) \frac{\partial k}{\partial x} \\ \frac{1}{Re_{ref}} \left(\mu + \mu_t / \sigma_\omega \right) \frac{\partial \omega}{\partial x} \end{Bmatrix}, F^{v*} = \begin{Bmatrix} \frac{1}{Re_{ref}} \left(\mu + \mu_t / \sigma_k \right) \frac{\partial k}{\partial y} \\ \frac{1}{Re_{ref}} \left(\mu + \mu_t / \sigma_\omega \right) \frac{\partial \omega}{\partial y} \end{Bmatrix}, G^{v*} = \begin{Bmatrix} \frac{1}{Re_{ref}} \left(\mu + \mu_t / \sigma_k \right) \frac{\partial k}{\partial z} \\ \frac{1}{Re_{ref}} \left(\mu + \mu_t / \sigma_\omega \right) \frac{\partial \omega}{\partial z} \end{Bmatrix}$$

$$E_V_tke = (\mu + \mu_t / \sigma_k) * d_tke_x / Re_ref$$

$$E_V_tdr = (\mu + \mu_t / \sigma_\omega) * d_tdr_x / Re_ref$$

$$F_V_tke = (\mu + \mu_t / \sigma_k) * d_tke_y / Re_ref$$

$$F_V_tdr = (\mu + \mu_t / \sigma_\omega) * d_tdr_y / Re_ref$$

$$G_V_tke = (\mu + \mu_t / \sigma_k) * d_tke_z / Re_ref$$

$$G_V_tdr = (\mu + \mu_t / \sigma_\omega) * d_tdr_z / Re_ref$$

(3) Total fluxes:

$$E_tke = E_I_tke - E_V_tke,$$

$$E_tdr = E_I_tdr - E_V_tdr$$

$$F_tke = F_I_tke - F_V_tke,$$

$$F_tdr = F_I_tdr - F_V_tdr$$

$$G_tke = G_I_tke - G_V_tke,$$

$$G_tdr = G_I_tdr - G_V_tdr$$

● Source Term Calculation

$$H_k = \frac{1}{Re_{ref}} \tau_y \frac{\partial U_i}{\partial x_j} - Re_{ref} C_\mu \rho k \omega, H_\omega = \frac{C_{\omega 1}}{Re_{ref}} \frac{\omega}{k} \tau_y \frac{\partial U_i}{\partial x_j} - Re_{ref} C_{\omega 2} \rho \omega^2$$

(1) **Production Terms:**

- **Reynolds Stress Tensor Calculation:**

$$\tau_{ij} = \mu_t \left[\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \frac{\partial U_k}{\partial x_k} \delta_{ij} \right] - \frac{2}{3} \rho \dot{k} \delta_{ij}$$

The contribution of each stress tensor component to the total production of turbulence kinetic energy is calculated.

e.g.,

$$\text{Prod_xy_tke} = \tau_{xy} * d_U_y / \text{Re_ref}$$

$$\text{Prod_xy_tdr} = \tau_{xy} * d_U_y * (\text{tdr} / \text{tke}) * C_{\omega 1} / \text{Re_ref}$$

(2) **Destruction Terms:**

$$\text{Dest_tke} = \rho * \text{tke} * \text{tdr} * C_{\mu} * \text{Re_ref}$$

$$\text{Dest_tdr} = \rho * \text{tdr}^2 * C_{\omega 2} * \text{Re_ref}$$

(3) **Source Terms:**

$$H_{\text{tke}} = \text{Prod_tke} - \text{Dest_tke}$$

$$H_{\text{tdr}} = \text{Prod_tdr} - \text{Dest_tdr}$$

(4) **Source Term Linearisation Matrix:** $\frac{\partial H}{\partial Q} = \begin{bmatrix} -2 \text{Re}_{ref} C_{\mu} \omega & 0 \\ 0 & -2 \text{Re}_{ref} C_{\omega 2} \omega \end{bmatrix}$

$$d_H_Q_tke = -2 * \text{tdr} * C_{\mu} * \text{Re_ref}$$

$$d_H_Q_tdr = -2 * \text{tdr} * C_{\omega 2} * \text{Re_ref}$$

- **Boundary Conditions and Lower Limits:**

- **Entire domain:**

if: turbulence quantity < minimum setting:

$$\text{tke} = \text{tke_min}$$

$$\text{tdr} = \text{tdr_min}$$

Specified BC's for k^ and ω^* are enforced at inflow, outflow, wall boundaries, etc.*

- Inflow Boundary:

$$\begin{aligned} tke &= tke_inflow \\ tdr &= tdr_inflow \end{aligned}$$

- Outflow Boundary:

Values at the outlet are obtained by a first-order extrapolation from the interior of the domain.

$$\begin{aligned} tke &= tke_upstream \\ tdr &= tdr_upstream \end{aligned}$$

- Wall Boundary:

$$\omega_{wall}^* = \frac{U_\tau^2 S_R}{\nu}, S_R = \begin{cases} (50 / k_R^*)^2 & k_R^* < 25 \\ 100 / k_R^* & k_R^* > 25 \end{cases}, k_R^* = 2.4 y_1^{+0.85}$$

The k - ω model can be integrated all the way to the wall. At the wall, turbulence kinetic energy is equal to zero whereas ω has a finite value.

$$\begin{aligned} \text{if: } k_R < 25 \\ & S_R = (50 / k_R)^2 \\ \text{if: } k_R > 25 \\ & S_R = 100 / k_R \end{aligned}$$

$$tdr_wall = \rho * U_tau^2 * S_R / \mu$$

- Evaluate Residuals

- Update Solution Vectors: $Q^{(1)} = Q^{(0)} - \frac{\alpha_i R^{(0)}}{V \cdot \left[\frac{I}{\Delta t} - \frac{\partial H}{\partial Q} \right]^n}$

$$R_tke = E_tke * A_x + F_tke * A_y + G_tke * A_z - H_tke * Vol_c - D * Vol_c$$

$$R_tdr = E_tdr * A_x + F_tdr * A_y + G_tdr * A_z - H_tdr * Vol_c - D * Vol_c$$

$$Q_tke = Q_tke - \alpha_1 * R_tke / ((I / \Delta t - H_tke_Q) * Vol_c)$$

$$Q_tdr = Q_tdr - \alpha_1 * R_tdr / ((I / \Delta t - H_tdr_Q) * Vol_c)$$

$$tke = Q_tke / \rho$$

$$tdr = Q_tdr / \rho$$

Calculate eddy-viscosity

$$\mu_t = \rho * tke / tdr$$

STEP 3 **Runge-Kutta Integration Stage 2**

Same as stage 1 in STEP 2

STEP 4 **Smooth the residuals implicitly using ADI or MSIP.**

A.2.3 Menter's SST Turbulence Model

The following pseudo-code applies to Menter's (1994) Shear Stress Transport (SST) model.

STEP 1 Initialisation of Variables

- Initialise turbulence quantities based on user-specified values.
All variables are specified at time t_i , either as initial values or previous solution.
- Control volume area and volume calculations are assumed completed.

- Calculate reference Reynolds number from user-specified values:

$$Re_{ref} = \rho_{ref} * U_{ref} * L_{ref} / \mu_{ref}$$

- Set following variables to initial values:

$$tke = tke_{init}$$

$$tdr = tdr_{init}$$

$$U = U_{init}$$

$$V = V_{init}$$

$$W = W_{init}$$

- Set lower cutoff values for the turbulence quantities:

$$tke = tke_{min}$$

$$tdr = tdr_{min}$$

STEP 2 Runge-Kutta Integration Stage 1

The application of preconditioning and multigrid convergence acceleration have been excluded for brevity. The eddy-viscosity value used in the present iteration is obtained from the previous integration cycle or from initial values.

- Evaluate turbulent fluxes:

Volume integrals containing derivatives are converted to surface integrals by applying Gauss' divergence theorem: $\oint_V (\nabla \cdot \phi) dV \approx \oint_S \phi \cdot dS$

(1) Convective fluxes: $E' = \begin{Bmatrix} \rho' U' k' \\ \rho' U' \omega' \end{Bmatrix}$, $F' = \begin{Bmatrix} \rho' V' k' \\ \rho' V' \omega' \end{Bmatrix}$, $G' = \begin{Bmatrix} \rho' W' k' \\ \rho' W' \omega' \end{Bmatrix}$

$$E_I_tke = \rho \cdot U \cdot tke$$

$$E_I_tdr = \rho \cdot U \cdot tdr$$

$$F_I_tke = \rho \cdot V \cdot tke$$

$$F_I_tdr = \rho \cdot V \cdot tdr$$

$$G_I_tke = \rho \cdot W \cdot tke$$

$$G_I_tdr = \rho \cdot W \cdot tdr$$

(2) Diffusive fluxes:

$$E^{v*} = \begin{Bmatrix} \frac{1}{Re_{ref}} \left(\mu' + \mu_t' / \sigma_k \right) \frac{\partial k'}{\partial x'} \\ \frac{1}{Re_{ref}} \left(\mu' + \mu_t' / \sigma_\omega \right) \frac{\partial \omega'}{\partial x'} \end{Bmatrix}, F^{v*} = \begin{Bmatrix} \frac{1}{Re_{ref}} \left(\mu' + \mu_t' / \sigma_k \right) \frac{\partial k'}{\partial y'} \\ \frac{1}{Re_{ref}} \left(\mu' + \mu_t' / \sigma_\omega \right) \frac{\partial \omega'}{\partial y'} \end{Bmatrix}, G^{v*} = \begin{Bmatrix} \frac{1}{Re_{ref}} \left(\mu' + \mu_t' / \sigma_k \right) \frac{\partial k'}{\partial z'} \\ \frac{1}{Re_{ref}} \left(\mu' + \mu_t' / \sigma_\omega \right) \frac{\partial \omega'}{\partial z'} \end{Bmatrix}$$

$$E_V_tke = (\mu + \mu_t / \sigma_k) \cdot d_tke_x / Re_ref$$

$$E_V_tdr = (\mu + \mu_t / \sigma_\omega) \cdot d_tdr_x / Re_ref$$

$$F_V_tke = (\mu + \mu_t / \sigma_k) \cdot d_tke_y / Re_ref$$

$$F_V_tdr = (\mu + \mu_t / \sigma_\omega) \cdot d_tdr_y / Re_ref$$

$$G_V_tke = (\mu + \mu_t / \sigma_k) \cdot d_tke_z / Re_ref$$

$$G_V_tdr = (\mu + \mu_t / \sigma_\omega) \cdot d_tdr_z / Re_ref$$

(3) Total fluxes:

$$E_tke = E_I_tke - E_V_tke, \quad E_tdr = E_I_tdr - E_V_tdr$$

$$F_tke = F_I_tke - F_V_tke, \quad F_tdr = F_I_tdr - F_V_tdr$$

$$G_tke = G_I_tke - G_V_tke, \quad G_tdr = G_I_tdr - G_V_tdr$$

● Source Term Calculation

$$H_k' = \frac{1}{Re_{ref}} \tau_y' \frac{\partial U_i'}{\partial x_j'} - Re_{ref} C_\mu \rho' k' \omega',$$

$$H_s^* = \frac{C_{s1}}{Re_{ref}} \frac{\omega^*}{k^*} \tau_y^* \frac{\partial U_i^*}{\partial x_j^*} - Re_{ref} C_{s2} \rho^* \omega^{*2} + \frac{2}{Re_{ref}} \rho^* \left(\frac{1 - F_1}{\sigma_s \omega^*} \frac{\partial k^*}{\partial x_j^*} \frac{\partial \omega^*}{\partial x_j^*} \right)$$

(1) **Blending Function F_1 Calculation:**

$$F_1 = \tanh \left\{ \left[\min \left(\max \left(\frac{1}{Re_{ref}} \frac{\sqrt{k^*}}{C_\mu \omega^* y_{wall}^*}; \frac{1}{Re_{ref}^2} \frac{500 \mu^*}{\rho^* \omega^* y_{wall}^{*2}} \right); \frac{4 \rho^* \sigma_{s2} k^*}{CD_{kw} y_{wall}^{*2}} \right) \right]^4 \right\}$$

$$\text{Cross Diffusion: } CD_{kw}^* = \max \left[\frac{\rho_{ref} U_{ref}^2}{L_{ref}^2} \left(\frac{2 \rho^*}{\sigma_{s2} \omega^*} \frac{\partial k^*}{\partial x_j^*} \frac{\partial \omega^*}{\partial x_j^*} \right); CD_{kw}^{min} \right]$$

$$CD_a = \rho_{ref} * U_{ref}^2 * (2 * \rho * tke * tdr * A_y) / \\ (L_{ref}^2 * \sigma_{w2} * tdr * Vol_c) \\ CD = \max(CD_a; 10^{-20})$$

$$F1_a = tke^{1/2} / (tdr * y_{wall} * C_mu * Re_{ref}) \\ F1_b = 500 * \mu / (tdr * y_{wall}^2 * \rho * Re_{ref}^2) \\ F1_c = 4 * \rho * \sigma_{w2} * tke / (CD * y_{wall}^2) \\ F1 = \tanh \{ [\min (\max (F1_a; F1_b); F1_c)]^4 \}$$

$$(2) \quad \text{F2 Calculation: } F_2 = \tanh \left\{ \left[\max \left(\frac{1}{Re_{ref}} \frac{2 \sqrt{k^*}}{C_\mu \omega^* y_{wall}^*}; \frac{1}{Re_{ref}^2} \frac{500 \mu^*}{\rho^* \omega^* y_{wall}^{*2}} \right) \right]^2 \right\}$$

$$F2 = \tanh \{ [\max (2 * F1_a; F1_b)]^2 \}$$

$$(3) \quad \text{Vorticity Calculation: } \Omega_y^* = \left(\frac{\partial U_i^*}{\partial x_j^*} - \frac{\partial U_j^*}{\partial x_i^*} \right)$$

$$vort = ((U * A_y - V * A_x) + (U * A_z - W * A_x) + (V * A_z - W * A_y)) / (Vol_c)$$

(4) **Production Terms:**

$$\bullet \quad \text{Stress Tensor Calculation: } \tau_{ij}^* = \mu_i^* \left[\left(\frac{\partial U_i^*}{\partial x_j^*} + \frac{\partial U_j^*}{\partial x_i^*} \right) - \frac{2}{3} \frac{\partial U_k^*}{\partial x_k^*} \delta_{ij} \right] - \frac{2}{3} \rho^* k^* \delta_{ij}$$

The contribution of each stress tensor component to the total production of turbulence kinetic energy is calculated.

e.g.,

e.g.,

$$\text{Prod_xy_tke} = \tau_{xy} * d_U_y / \text{Re_ref}$$

$$\text{Prod_xy_tdr} = \tau_{xy} * d_U_y * (\text{tdr} / \text{tke}) * C_{\omega 1} / \text{Re_ref}$$

(5) Destruction Terms:

$$\text{Dest_tke} = \rho * \text{tke} * \text{tdr} * C_{\mu} * \text{Re_ref}$$

$$\text{Dest_tdr} = \rho * \text{tdr}^2 * C_{\omega 2} * \text{Re_ref}$$

(6) Source Terms:

$$H_{\text{tke}} = \text{Prod_tke} - \text{Dest_tke}$$

$$H_{\text{tdr}} = \text{Prod_tdr} - \text{Dest_tdr} + 2 * \rho * (1 - F1) * (\text{tke} * \text{tdr} * A_y^2) / (\text{tdr} * \sigma_w * \text{Vol}_c * \text{Re_ref})$$

(7) Source Term Linearisation Matrix: $\frac{\partial H}{\partial Q} = \begin{bmatrix} -2 \text{Re}_{ref} C_{\mu} \omega^* & 0 \\ 0 & -2 \text{Re}_{ref} C_{\omega 2} \omega^* \end{bmatrix}$

$$d_H_Q_{\text{tke}} = -2 * \text{tdr} * C_{\mu} * \text{Re_ref}$$

$$d_H_Q_{\text{tdr}} = -2 * \text{tdr} * C_{\omega 2} * \text{Re_ref}$$

● Boundary Conditions and Lower Limits:

- Entire domain:

if: turbulence quantity < minimum setting:

$$\text{tke} = \text{tke_min}$$

$$\text{tdr} = \text{tdr_min}$$

Specified BC's for k^ and ω^* are enforced at inflow, outflow, wall boundaries, etc.*

- Inflow Boundary:

$$\text{tke} = \text{tke_inflow}$$

$$\text{tdr} = \text{tdr_inflow}$$

- **Outflow Boundary:**
Values at the outlet are obtained by a first-order extrapolation from the interior of the domain.

$$tke = tke_upstream$$

$$tdr = tdr_upstream$$

- **Wall Boundary:** $\omega_{wall} = \frac{\beta_{wall}}{Re_{ref}^2} \frac{C_{\omega 3} \mu}{C_{\omega 2} \rho y_1^2}$

At the wall, turbulence kinetic energy is equal to zero whereas ω has a finite value.

$$tdr_wall = \beta_{wall} * C_{\omega 3} * \mu / (C_{\omega 2} * \rho * y_1^2 * Re_{ref}^2)$$

• Evaluate Residuals

- **Update Solution Vectors:** $Q^{(1)} = Q^{(0)} - \frac{\alpha_l R^{(0)}}{V \left[\frac{I}{\Delta t} - \frac{\partial H}{\partial Q} \right]}$

$$R_tke = E_tke * A_x + F_tke * A_y + G_tke * A_z - H_tke * Vol_c - D * Vol_c$$

$$R_tdr = E_tdr * A_x + F_tdr * A_y + G_tdr * A_z - H_tdr * Vol_c - D * Vol_c$$

$$Q_tke = Q_tke - \alpha_l * R_tke / ((I / \Delta t - H_tke_Q) * Vol_c)$$

$$Q_tdr = Q_tdr - \alpha_l * R_tdr / ((I / \Delta t - H_tdr_Q) * Vol_c)$$

$$tke = Q_tke / \rho$$

$$tdr = Q_tdr / \rho$$

Calculate eddy-viscosity

$$\mu_t = \rho * tke / tdr$$

STEP 3 Runge-Kutta Integration Stage 2

Same as stage 1 in STEP 2

STEP 4 Smooth the residuals implicitly using ADI or MSIP.

A.2.4 The SA Turbulence Model

The following pseudo-code applies to the one-equation model of Spalart and Allmaras (1992).

STEP 1 Initialisation of Variables

- Initialise turbulence quantities based on user-specified values.
All variables are specified at time t_j , either as initial values or previous solution.
- Control volume area and volume calculations are assumed completed.

- Calculate reference Reynolds number from user-specified values:

$$Re_ref = \rho_ref * U_ref * L_ref / \mu_ref$$

- Set the following variables to initial values:

$$\nu_sa = \nu_sa_init$$

$$U = U_init$$

$$V = V_init$$

$$W = W_init$$

- Set lower cutoff values for the turbulence quantities:
 $\nu_sa = \nu_sa_min$

STEP 2 Runge-Kutta Integration Stage 1

The application of preconditioning and multigrid convergence acceleration have been excluded for brevity. The eddy-viscosity value used in the present iteration is obtained from the previous integration cycle or from initial values.

- Evaluate turbulent fluxes:

Volume integrals containing derivatives are converted to surface integrals by applying the Gauss divergence theorem: $\oint_V (\nabla \cdot \phi) dV = \oint_S \phi \cdot dS$

- (1) Convective fluxes: $E^{i*} = \{\rho^* U^* \tilde{v}^*\}$, $F^{i*} = \{\rho^* V^* \tilde{v}^*\}$, $G^{i*} = \{\rho^* W^* \tilde{v}^*\}$

$$E_I_nu_sa = \rho * U * \nu_sa$$

$$F_I_nu_sa = \rho \cdot V \cdot nu_sa$$

$$G_I_nu_sa = \rho \cdot W \cdot nu_sa$$

(2) Diffusive fluxes:

$$E^{v^*} = \left\{ \frac{1}{\sigma Re_{ref}} (\mu^* + \rho^* \tilde{v}^*) \frac{\partial \tilde{v}^*}{\partial x^*} \right\}, F^{v^*} = \left\{ \frac{1}{\sigma Re_{ref}} (\mu^* + \rho^* \tilde{v}^*) \frac{\partial \tilde{v}^*}{\partial y^*} \right\},$$

$$G^{v^*} = \left\{ \frac{1}{\sigma Re_{ref}} (\mu^* + \rho^* \tilde{v}^*) \frac{\partial \tilde{v}^*}{\partial z^*} \right\}$$

$$E_V_nu_sa = (\mu + \rho \cdot nu_sa) \cdot d_nu_sa_x / (Re_ref \cdot sigma_sa)$$

$$F_V_nu_sa = (\mu + \rho \cdot nu_sa) \cdot d_nu_sa_y / (Re_ref \cdot sigma_sa)$$

$$G_V_nu_sa = (\mu + \rho \cdot nu_sa) \cdot d_nu_sa_z / (Re_ref \cdot sigma_sa)$$

(3) Total fluxes:

$$E_nu_sa = E_I_nu_sa - E_V_nu_sa$$

$$F_nu_sa = F_I_nu_sa - F_V_nu_sa$$

$$G_nu_sa = G_I_nu_sa - G_V_nu_sa$$

- Source Term Calculation**

$$H_{\tau}^* = C_{b1} \tilde{\Omega}^* \rho^* \tilde{v}^* - \frac{C_{w1} f_w}{Re_{ref}} \rho^* \left(\frac{\tilde{v}^*}{y_{wall}^*} \right)^2 + \frac{C_{b2}}{\sigma Re_{ref}} \rho^* \frac{\partial \tilde{v}^*}{\partial x_j^*} \frac{\partial \tilde{v}^*}{\partial x_j^*}$$

The SA source term contains a production term, a destruction term and a second diffusion term.

(1) Production Term

- Vorticity Calculation:** $\Omega_y^* = \left(\frac{\partial U_i^*}{\partial x_j^*} - \frac{\partial U_j^*}{\partial x_i^*} \right)$

$$vort = ((U \cdot A_y - V \cdot A_x) + (U \cdot A_z - W \cdot A_x) + (V \cdot A_z - W \cdot A_y)) / (Vol_c)$$

- Modified Vorticity Calculation:**

$$\tilde{\Omega}^* = \left| \Omega_y^* \right| + \frac{\tilde{v}^*}{\kappa^2 y_{wall}^2} f_{v2}, f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}}, \chi = \frac{\tilde{v}^*}{v^*}, f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}}$$

$$\begin{aligned}
\chi &= \nu_{sa} / (\mu / \rho) \\
f_{v1} &= \chi^3 / (\chi^3 + C_{v1}^3) \\
f_{v2} &= 1 - \chi / (1 + \chi * f_{v1}) \\
vort_{sa} &= (vort^2)^{1/2} + \nu_{sa} / (\kappa^2 + y_{wall}^2) \\
Prod &= C_{b1} * vort_{sa} * \rho * \nu_{sa}
\end{aligned}$$

(2) Destruction Term

- Damping Function Calculation:

$$f_w = g \left[\frac{1 + C_{w3}^6}{g^6 + C_{w3}^6} \right]^{1/6}, g = r + C_{w2} (r^6 - r), r = \frac{1}{Re_{ref}} \frac{\tilde{v}^2}{\tilde{\Omega}^2 \kappa^2 y_{wall}^2}$$

$$\begin{aligned}
r &= \nu_{sa} / (vort_{sa} * \kappa^2 * y_{wall}^2 * Re_{ref}) \\
g &= r + C_{w2} * (r^6 - r) \\
f_w &= g * ((1 + C_{w3}^6) / (g^6 + C_{w3}^6))^{1/6}
\end{aligned}$$

$$Dest = C_{w1} * f_w * \rho * (\nu_{sa} / y_{wall})^2 / Re_{ref}$$

(3) Second Diffusion Term $F_{d2} = \frac{C_{b2}}{\sigma Re_{ref}} \rho \frac{\partial \tilde{v}^2}{\partial x_j} \frac{\partial \tilde{v}^2}{\partial x_j}$

$$Fd2 = (\nu_{sa}^2 * A_y^2 * \rho * C_{b2}) / (\sigma_{sa} * Vol_c * Re_{ref})$$

(4) Source Term:

$$H = Prod - Dest + Fd2$$

(5) Source Term Linearisation: $\frac{\partial H_{\tilde{v}}}{\partial \tilde{Q}} = -2 \frac{C_{w1} f_w}{Re_{ref}} \frac{\tilde{v}^2}{y_{wall}^2}$

$$d_H_Q = -2 * f_w * \nu_{sa} * C_{w1} / (y_{wall}^2 * Re_{ref})$$

- Boundary Conditions and Lower Limits:

- Entire domain:

$$\begin{aligned}
&\text{if: turbulence quantity} < \text{minimum setting:} \\
&\quad \nu_{sa} = \nu_{sa_min}
\end{aligned}$$

- **Inflow Boundary:**
 $\text{nu_sa} = \text{nu_sa_inflow}$
- **Outflow Boundary:**
Values at the outlet are obtained by a first-order extrapolation from the interior of the domain.
 $\text{nu_sa} = \text{nu_sa_upstream}$
- **Wall Boundary:**
 $\text{nu_sa} = 0.0$
- **Evaluate Residuals**
- **Update Solution Vectors:** $Q^{*(1)} = Q^{*(0)} - \frac{\alpha_1 R^{(0)}}{V \cdot \left[\frac{I}{\Delta t} - \frac{\partial H}{\partial Q} \right]}$
- $R = E * A_x + F * A_y + G * A_z - H * \text{Vol_c} - D * \text{Vol_c}$
- $Q_{\text{nu_sa}} = Q_{\text{nu_sa_n}} - \alpha_1 * R / ((I / \Delta t) - H_Q) * \text{Vol_c}$
- $\text{nu_sa} = Q_{\text{nu_sa}} / \rho$
- **Calculate eddy-viscosity**
- $\mu_t = \rho * \text{nu_sa} * f_{v1}$

STEP 3 Runge-Kutta Integration Stage 2

Same as stage 1 in STEP 2

STEP 4 Smooth the residuals implicitly using ADI or MSIP.

A.3 Grid Details

The grids for all the test cases of this study were generated using the *I-DEAS*TM computer-aided engineering software. The grid-generation process in *I-DEAS* involves development of a solid model of the desired shape of the computational domain, followed by mapping of a “structured mesh” (i.e. one that contains uniquely defined grid directions of i and j in 2-D, and i , j and k in 3-D) onto the edges of this domain. Once the mapping process is completed, *I-DEAS* establishes the locations of the nodes that are off the edges of the computational domain through interpolation. Subsequent smoothing of the interpolated grid is not feasible within the *I-DEAS* environment. The clustering of the grid nodes in certain regions of the domain is realised by specifying end- or centred-biasing of the nodes that are mapped along the edges. Such bias is specified through a biasing factor, β_{I-DEAS} , which can be assigned values within a limited range. Specification of this factor yields a nonuniform variation of the rate of change of node spacing along the edges, which herein is referred to as the cell stretch:

$$Stretch = \frac{L_{cell_2} - L_{cell_1}}{L_{cell_1}} \quad (A.3.1)$$

where L_{cell_1} and L_{cell_2} are the sizes of two adjacent grid-cells in a selected grid direction.

Often such nonuniformity of cell *stretch* is not optimum for the flow problem at hand.

Although *I-DEAS* accommodates user-specified biasing functions to replace the biasing factors, it has been found that these functions are manipulated by the *I-DEAS* software

before being used for distributing the edge nodes. Communications with *SDRC*, the developer of the *I-DEAS* software, failed to identify the exact nature of this manipulation, and thus specification of biasing functions could not be used to distribute these nodes in a predictable fashion. Therefore, to establish sufficient control over the amount of *stretch* within the domain while using the biasing factors of *I-DEAS*, the computational domain for each of the test cases of the present study had to be divided into subdomains, for each of which the grid was generated independently. This can be a somewhat tedious process at times with the *VGJ* test case requiring 18 subdomains as an example. Use of such subdomains is effectively equivalent to the multiblock grid concept, except that at the end of the grid generation process these blocks of grid are *merged* into a single grid through a preprocessor for *FS*, so that the subdomains that exist during the grid generation process are transparent to *FS*. Details of the grids generated through this procedure for all the test cases of the present study are summarised in Figures A.3.1 to A.3.8¹. In the flat-plate and the axisymmetric diffuser cases, where extensive parametric studies were performed with respect to optimisation of the node distribution in the cross-stream direction, the domain subdivision process did not provide sufficiently independent control over the location of the first node off the wall and the number of nodes residing in the boundary layer. In these instances, the grid generated through *I-DEAS* had to be further processed by redistribution of the nodes in certain subdomains according to a user-specified function outside the *I-DEAS* environment.

¹The skew factor given for the M2129 grid in Figure A.3.8b is defined as the ratio of the shorter and longer diagonals of a cell subtracted from 1.

Figure A.3.1a Leading edge section of flat-plate grid A

i)	Nodes from wall			ii)	Nodes in BL / Nodes in BL at $x/L=0.6$	
		$y_{wall}(m.)$	$y^*(x/L=0.6)$		x/L	
	1	1.5×10^{-5}	1.3		9%	48%
	3	5.4×10^{-5}	4.7		38%	83%
	10	3.6×10^{-4}	31		96%	115%
	18	1.2×10^{-3}	104		100%	123%
	81	4.0×10^{-2}	3422			

Table A.3.1 Node locations in j direction of flat-plate grid A
i) distribution at $x/L=0.6$, ii) variation of BL resolution Figure A.3.1b Grid distribution of flat-plate grid A,
cell stretch distribution in i) j direction, ii) i direction,
iii) y_1^* distribution

$y = 0.06 - 0.15m.$,
33 nodes, $\beta_{1-DEAS}=2$

$y = 0.00 - 0.06m.$,
97 nodes, $\beta_{1-DEAS}=100$

$x = 0.0 - 0.1m.$,
9 nodes, $\beta_{1-DEAS}=10$

$x = 0.1 - 2.1m.$,
265 nodes, $\beta_{1-DEAS}=4$

Figure A.3.1c Layout of flat-plate grid A

Figure A.3.2a Leading edge section of flat-plate grid B

i)	Nodes from wall			ii)	Nodes in BL / Nodes in BL at $x/L=0.6$	
		$y_{wall}(m.)$	$y^* (x/L=0.6)$		x/L	
	1	3.3×10^{-5}	2.8		9%	48%
	2	7.2×10^{-5}	6.2		38%	83%
	6	3.7×10^{-4}	32		96%	115%
	11	1.2×10^{-3}	107		100%	123%
	30	4.0×10^{-2}	3448			

Table A.3.2 Node locations in j direction of flat-plate grid B
i) distribution at $x/L=0.6$, ii) variation of BL resolution Figure A.3.2b Grid distribution of flat-plate grid B,
cell stretch distribution in i) j direction, ii) i direction,
iii) y_1^* distribution

$y = 0.04 - 0.15m.$,
15 nodes, $\beta_{1-DEAS}=2$

$y = 0.001 - 0.04m.$,
21 nodes, $\beta_{1-DEAS}=19$

$y = 0.00 - 0.001m.$,
11 nodes, $\beta_{1-DEAS}=6$

$x = 0.0 - 0.1m.$,
9 nodes, $\beta_{1-DEAS}=5.2$

$x = 0.1 - 2.1m.$,
89 nodes, $\beta_{1-DEAS}=12$

Figure A.3.2c Layout of flat-plate grid B

Figure A.3.3a Leading edge section of flat-plate grid C

i)	Nodes from wall	$y_{wall}(m.)$	$y^+(x/L=0.6)$	ii)	x/L	Nodes in BL / Nodes in BL at $x/L=0.6$
	1	5.9×10^{-5}	5.1		9%	48%
	5	3.9×10^{-4}	33		38%	83%
	10	1.1×10^{-3}	98		96%	115%
	35	4.1×10^{-3}	3503		100%	123%

Table A.3.3 Node locations in j direction of flat-plate grid C
i) distribution at $x/L=0.6$, ii) variation of BL resolution Figure A.3.3b Grid distribution of flat-plate grid C,
cell stretch distribution in i) j direction, ii) i direction,
iii) y_1^+ distribution

Figure A.3.3c Layout of flat-plate grid C

Figure A.3.4a Leading edge section of flat-plate grid D

i)	Nodes from wall			ii)	Nodes in BL / Nodes in BL at x/L=0.6	
		$y_{wall}(m.)$	$y^+(x/L=0.6)$		x/L	
	1	5.9×10^{-5}	5.1		9%	48%
	3	2.8×10^{-4}	24		38%	83%
	6	1.3×10^{-3}	110		96%	115%
	14	4.0×10^{-2}	3448		100%	123%

Table A.3.4 Node locations in j direction of flat-plate grid D
i) distribution at $x/L=0.6$, ii) variation of BL resolution Figure A.3.4b Grid distribution of flat-plate grid D,
cell stretch distribution in i) j direction, ii) i direction,
iii) y_1^+ distribution

$y = 0.04 - 0.15m.$
9 nodes, Uniform

$y = 0.00 - 0.04m.$
15 nodes, %Stretch=52

$x = 0.0 - 0.1m.$
9 nodes, $\beta_{I-DEAS}=5.2$

$x = 0.1 - 2.1m.$
89 nodes, $\beta_{I-DEAS}=12$

Figure A.3.4c Layout of flat-plate grid D

Figure A.3.5a Leading edge section of flat-plate grid E

i)	Nodes from wall	$y_{wall}(m.)$	$y^+ (x/L=0.6)$	ii)	x/L	Nodes in BL / Nodes in BL at $x/L=0.6$
	1	3.5×10^{-5}	3.0		9%	48%
	5	3.9×10^{-4}	33		38%	83%
	8	1.2×10^{-3}	106		96%	115%
	18	3.9×10^{-2}	3387		100%	123%

Table A.3.5 Node locations in j direction of flat-plate grid E
i) distribution at $x/L=0.6$, ii) variation of BL resolution Figure A.3.5b Grid distribution of flat-plate grid E,
cell stretch distribution in i) j direction, ii) i direction,
iii) y_1^+ distribution

$y = 0.00 - 0.15m.$,
23 nodes,
%Stretch=40

$x = 0.0 - 0.1m.$,
9 nodes, $\beta_{1-DEAS}=5.2$

$x = 0.1 - 2.1m.$,
89 nodes, $\beta_{1-DEAS}=12$

Figure A.3.5c Layout of flat-plate grid E

Figure A.3.6a A streamwise segment of the C.S0 grid

i)	Nodes from wall ($x/R_0 = -6.53$)	$y_{wall}(m.)$	y^*
	1	1.6×10^{-5}	1.3
	3	5.9×10^{-5}	4.8
	9	3.9×10^{-4}	31.3
	16	1.4×10^{-3}	111.3
	40	13.1×10^{-3}	1063.7

ii)	Nodes from wall ($x/R_0 = 1.09$)	$y_{wall}(m.)$	U/U_{edge}
	14	1.6×10^{-3}	-2.0×10^{-2}
	18	2.9×10^{-3}	-9.8×10^{-3}
	24	6.0×10^{-3}	1.5×10^{-2}
	56	5.7×10^{-2}	6.8×10^{-2}
	61	6.8×10^{-2}	6.8×10^{-2}

Table A.3.6 Node locations in the C.S0 grid,
i) inflow boundary, ii) region of separation Figure A.3.6b Cell stretch distribution of C.S0 grid,
i) j direction, ii) i direction,
iii) y_1^* distribution

Figure A.3.6c Layout of C.S0 grid

Figure A.3.7a VGJ Grid A

Nodes from wall	$y_{wall}(m.)$	y^*
1	1.8×10^{-4}	0.6
4	7.0×10^{-4}	4.8
10	4.5×10^{-3}	31.5
16	1.4×10^{-2}	97.2
20	2.5×10^{-2}	170.0

Table A.3.7 Node locations in j-direction of VGJ Grid A at the inflow boundary

Figure A.3.7b Grid distribution of VGJ Grid A, cell stretch in i) k-direction, ii) i-direction (at $y/D=0$), iii) y_i^* along $y/D=0$

13 nodes, $\beta_{kmax}=8$	13 nodes, $\beta_{kmax}=0.2$	21 nodes, $\beta_{kmax}=8$
21 nodes, $\beta_{kmax}=0.7$	21 nodes, $\beta_{kmax}=0.2$	
9 nodes, $\beta_{kmax}=2$		

Figure A.3.7c Layout of VGJ Grid A

Figure A.3.8a M2129 Grid A

Nodes from wall	$y_{wall}(m.)$	y^*
1	2.2×10^{-5}	1.2
3	9.6×10^{-5}	5.2
7	5.8×10^{-4}	31.3
11	1.9×10^{-3}	104.3
19	9.6×10^{-3}	510.0

Table A.3.8 Node locations in M2129 Grid A at the inflow boundary ($z = -0.5D_1$, $y = 0$) Figure A.3.8b Details of M2129 Grid A, i) j-direction cell stretch distribution, ii) k-direction cell stretch distribution, iii) skew, iv) i-direction cell stretch distribution, v) y_1^+ distribution

Figure A.3.8c Layout of M2129 Grid A