University of Warwick institutional repository: http://go.warwick.ac.uk/wrap

A Thesis Submitted for the Degree of PhD at the University of Warwick

http://go.warwick.ac.uk/wrap/2372

This thesis is made available online and is protected by original copyright. Please scroll down to view the document itself. Please refer to the repository record for this item for information to help you to cite it. Our policy information is available from the repository home page.

James Jewkes
MEng(Warwick)

This report is submitted as partial fulfilment of the requirements for the PhD Programme of the School of Engineering
University of Warwick
April 2008
AUTHOR: James Jewkes       DEGREE: PhD


DATE OF DEPOSIT: ............................

I agree that this thesis shall be available in accordance with the regulations governing the University of Warwick theses.

I agree that the summary of this thesis may be submitted for publication.

I agree that the thesis may be photocopied (single copies for study purposes only).

Theses with no restriction on photocopying will also be made available to the British Library for microfilming. The British Library may supply copies to individuals or libraries, subject to a statement from them that the copy is supplied for non-publishing purposes. All copies supplied by the British Library will carry the following statement:

"Attention is drawn to the fact that the copyright of this thesis rests with its author. This copy of the thesis has been supplied on the condition that anyone who consults it is understood to recognise that its copyright rests with its author and that no quotation from the thesis and no information derived from it may be published without the author's written consent."

AUTHOR’S SIGNATURE: ........................................................

USER’S DECLARATION

1. I undertake not to quote or make use of any information from this thesis without making acknowledgement to the author.

2. I further undertake to allow no-one else to use this thesis while it is in my care.

DATE   SIGNATURE   ADDRESS

...........................................................................
...........................................................................
...........................................................................
...........................................................................
...........................................................................
...........................................................................
...........................................................................
...........................................................................
Abstract

The jet acting perpendicular to a cross-flow boundary layer is a commonly studied complex turbulent flow. Our research was motivated by their potential application in separation delay devices, where jets can be used to produce streamwise vortices in a similar manner to conventional solid vortex generating vanes.

This thesis addresses two problems; firstly the generation of inflow conditions for the simulation of a spatially developing turbulent boundary layer, and secondly the simulation of low velocity ratio jets interacting with the boundary layer. Our approach involved refining a popular turbulent inflow generation technique, validating the accuracy of our improved method against well established direct numerical simulation data. This turbulent boundary layer was used to simulate a low velocity ratio perpendicular jet test-case, which was validated against experimental data. Finally, a pitched and skewed jet model was investigated.

Our modifications to the turbulent boundary layer inflow generation method were successful, addressing problems described by various authors regarding the stability and accuracy of the technique. Secondly we have found excellent agreement in our perpendicular jet in cross flow test-case, and have produced what we believe to be the first documented unsteady numerical simulation of the flow field behind a low velocity ratio pitched and skewed jet.
Acknowledgements

This thesis is dedicated in memory of Professor Peter Carpenter. Peter’s patience, kindness, wisdom, and ability to explain the most difficult concepts clearly and simply were essential to the success of this research. Peter was a true gentleman, and it has been an honour and a pleasure to have had the opportunity to work so closely with such an accomplished scholar.

I would also like to thank Dr Yongmann Chung, my secondary supervisor. Yongmann’s LES code was the fundamental basis upon which this work has been built; without access to this code, none of the research presented herein would have been possible. Throughout the PhD, Yongmann has been a fantastic source of technical advice about CFD. I’m profoundly grateful that he stepped in for Peter during my final year, particularly with regards to the thorough feedback and criticism he has provided in proof-reading this thesis.

I am indebted to my many postgraduate colleagues in the Fluid Dynamics Research Centre at Warwick, for providing a fun, supportive and enthusiastic environment in which to learn and develop. Particular thanks to Adam Preece, who worked alongside Peter, Yongmann and myself on a similar project. Also Novak Elliott, who was an excellent source of technical advice and moral support, my thanks particularly for his help with \LaTeX. Thanks to Dave Hunter, Mark Brend, Jawad Nawasra, Amit Kiran, Sandy Gregorio, Dr Andrew Skeen, Dr Paul
Lastly, and most importantly, I wish to thank my family - my sister Katherine, and my parents, Mike and Jackie. They provided food and accommodation, endless patience, financial support, and love. It wouldn’t have been possible without them.
The research was funded as part of the European collaborative research project (AEROMEMS II) which focused on exploring the viability of applying Micro-fabricated Electro-Mechanical-Systems (MEMS) for active flow separation control. It was jointly funded by research establishments, industry, and the CEC (Commission of the European Communities) under the 5th framework, "Promoting Competitive and Sustainable Growth" programme, Key Action 4 - "New Perspectives in Aeronautics" (contract G4RD-CT-2002-00748).

The work was led by BAE Systems, key industrial collaborators included, Airbus, Auxitrol, CNRS, Dassault Aviation, DLR, EADS, Snecma, ONERA, and the Universities of Athens (NTUA), Berlin (TUB), Cranfield, Lille, Manchester and Warwick.
Abbreviations

- 2D - Two Dimensional
- 3D - Three Dimensional
- CFD - Computational Fluid Dynamics
- CFL - Courant-Friedrichs-Levy
- CVP - Counter-rotating Vortex Pair
- DNS - Direct Numerical Simulation
- JICF - Jet in Cross Flow
- LDA - Laser Doppler Anemometry
- LES - Large Eddy Simulation
- LSE - Linear Stochastic Estimation
- LWS - Lund Wu and Squires
- NS - Navier-Stokes
- PIV - Particle Image Velocimetry
- POD - Proper Orthogonal Decomposition
• RANS - Reynolds-Averaged Navier Stokes

• SGS - Sub-Grid Scale

• SIMPLE - Semi-Implicit Method for Pressure-Linked Equations

• TBL - Turbulent Boundary Layer

• TKE - Turbulent Kinetic Energy

• V/STOL - Vertical/Short Takeoff and Landing

• VGJ - Vortex Generating Jet

• VR - Velocity Ratio
# Contents

- Abstract iii
- Acknowledgements iv
- Abbreviations vii

## I  Problem Definition & Background 1

1 Introduction 2

2 Literature Review 4
  2.1 Introduction ........................................ 4  
  2.2 Review of existing inflow generation methods ........... 5  
    2.2.1 Using experimental data as an inflow condition ... data ...... 5  
    2.2.2 Numerical generation of turbulent inflow data ......... 7  
    2.2.3 Random fluctuations .............................. 8  
    2.2.4 Precursor simulations ............................ 11  
    2.2.5 Recycling ......................................... 12  
  2.3 Summary of the key advantages of using LWS-based methods . 14  
  2.4 Conclusions ........................................ 15

## II  Methods 16

3 LES Code Formulation 17
  3.1 Introduction .......................................... 17  
  3.2 Generalised LES formulation .......................... 18  
  3.3 Subgrid-scale model .................................. 20  
  3.4 Numerical method ..................................... 24  
  3.5 Summary of the key features of our LES code .......... 27  
  3.6 Conclusion .......................................... 28
# 4 Turbulent Inflow Generation

4.1 Introduction .................................. 29
4.2 Concept ..................................... 30
4.3 Initial conditions ........................... 31
4.4 Basic rescaling algorithm .................... 34
4.5 Iteration of $u_{r, init}$ to ensure $\delta_{init}^* = 1.0$ .... 39
4.6 Correction of durable spanwise mean flow variations . 39
4.7 Main simulation domain ..................... 42
4.8 Summary of our modified inflow generation technique .... 44
4.9 Rescaling flow-chart ....................... 45
4.10 Conclusions .............................. 46

## III Results & Discussion

5 Validation of the Inflow Condition 48

5.1 Introduction ............................... 48
5.2 Simulation parameters ...................... 49
5.3 Temporal flow development at the inlet ........ 50
5.4 Time-averaged validation data ............. 56
5.5 Grid refinement ............................ 64
5.6 Instantaneous contour plots ................ 66
5.7 Main simulation domain ................... 70
5.8 Summary of key validation plots .......... 75
5.9 Conclusions .............................. 77

6 Discussion of the Inflow-Generation Technique 78

6.1 Introduction .................................. 78
6.2 Existing observations ......................... 79
6.2.1 Issues regarding initial conditions ....... 79
6.2.2 Spurious periodicity - location of the recycling plane ... 80
6.2.3 Other observations ...................... 84
6.3 Modifications to the LWS technique .............. 84
6.3.1 Use of $\delta^*$ instead of $\delta$ ....... 85
6.3.2 Initial conditions ....................... 86
6.3.3 Upper boundary condition .......... 88
6.3.4 Alternative weighting function .......... 89
6.3.5 Use of $\gamma$ in $u'$ rescaling .......... 90
6.3.6 Placement of recycling plane, and correction of durable spanwise mean flow variations .......... 92
6.4 Summary ................................ 96
6.5 Conclusions .............................. 97
7 Simulation of Steady Jets in Cross-Flow

7.1 Introduction ........................................ 99

7.2 Jets in cross-flow .................................... 101
  7.2.1 Perpendicular jet in cross-flow ............... 102
  7.2.2 High velocity ratios - VR > 2.0 ............. 103
  7.2.3 Lower velocity ratios - VR < 2.0 ............ 106
  7.2.4 Pitched and skewed jets in cross-flow ...... 108
  7.2.5 Application of jets for flow control ....... 110

7.3 Simulation configuration ............................ 111
  7.3.1 Jet configuration ................................ 112

7.4 Jet test-cases - time-averaged velocity fields .... 114
  7.4.1 Perpendicular jet ................................ 114
  7.4.2 Pitched and skewed jet ....................... 120

7.5 Perpendicular jet validation ....................... 126

7.6 Pitched and Skewed jet ............................ 135

7.7 Discussion .......................................... 141

7.8 Conclusions ......................................... 144

8 Conclusion ............................................ 146

References .............................................. 150

IV Appendices ........................................... 160

A Summary of the Original LWS Formulation ........... 161

B Rescaling Subroutine ................................ 163

C Further JICF Results ................................ 170
  C.1 Steady Jet VR=0.5, perpendicular to flow ....... 170
  C.2 Steady Jet VR=0.5, pitched and skewed ......... 175

D Theoretical Model of a Laminar 2D Jet ................ 180
  D.1 Introduction ....................................... 180
  D.2 Theory ............................................ 180
    D.2.1 Variables ..................................... 180
    D.2.2 Approximate forms for velocity profiles .... 181
    D.2.3 Boundary conditions .......................... 182
    D.2.4 Conservation of mass ......................... 183
    D.2.5 Conservation of momentum ..................... 184
    D.2.6 Summary of the governing equations ......... 187
    D.2.7 Non-dimensional form of the governing equations .... 187
    D.2.8 Solution for small values of $\xi$ ............ 188
  D.3 Results & discussion ................................ 191
### List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1</td>
<td>Illustration of scale-separation in Large Eddy Simulations</td>
<td>18</td>
</tr>
<tr>
<td>3.2</td>
<td>Three dimensional illustration of a cell in a staggered grid</td>
<td>27</td>
</tr>
<tr>
<td>4.1</td>
<td>Illustration of the inflow generation technique</td>
<td>30</td>
</tr>
<tr>
<td>4.2</td>
<td>Example of u contour field at initialisation</td>
<td>32</td>
</tr>
<tr>
<td>4.3</td>
<td>Example of instantaneous u&amp;v velocity profiles at initialisation</td>
<td>33</td>
</tr>
<tr>
<td>4.4</td>
<td>Development of the characteristic timescale, $T$</td>
<td>34</td>
</tr>
<tr>
<td>4.5</td>
<td>Weighting function</td>
<td>38</td>
</tr>
<tr>
<td>4.6</td>
<td>Methods employed to disorganise durable spanwise fluctuations in the mean flow</td>
<td>40</td>
</tr>
<tr>
<td>4.7</td>
<td>Exaggerated illustration of time synchronisation between the inflow and main simulations</td>
<td>42</td>
</tr>
<tr>
<td>4.8</td>
<td>Flow-chart illustrating the rescaling process</td>
<td>45</td>
</tr>
<tr>
<td>5.1</td>
<td>Grid used for xy axes</td>
<td>49</td>
</tr>
<tr>
<td>5.2</td>
<td>Mass conservation</td>
<td>51</td>
</tr>
<tr>
<td>5.3</td>
<td>Development of $u_r$ at the inlet</td>
<td>52</td>
</tr>
<tr>
<td>5.4</td>
<td>Development of gamma at the inlet</td>
<td>53</td>
</tr>
<tr>
<td>5.5</td>
<td>Number of iterations required to achieve $\delta_{init}^* = 1.0.$</td>
<td>54</td>
</tr>
<tr>
<td>5.6</td>
<td>Development of $\delta_{init}^*$ and $\theta_{init}$ over time</td>
<td>55</td>
</tr>
<tr>
<td>5.7</td>
<td>Time-averaged boundary layer profile</td>
<td>56</td>
</tr>
</tbody>
</table>
5.8 Time-averaged logarithmic boundary layer profile. .......................... 57
5.9 Time-averaged turbulent fluctuations. ........................................ 58
5.10 Time-averaged $\delta^*$ and $\theta$. ........................................ 59
5.11 Time-averaged $\delta^*$ plotted against $Re_\theta$. ....................... 60
5.12 Time-averaged shape function. ................................................ 61
5.13 Time-averaged $C_f$ in a streamwise direction. ............................ 62
5.14 Time-averaged logarithmic boundary layer profile - finer grid res-
    olution. ...................................................................................... 64
5.15 Time-averaged turbulent fluctuations - finer grid resolution. ........... 65
5.16 Instantaneous $xz$ contour plot at $II = 100000$. ........................... 67
5.17 Instantaneous $xy$ contour plot at $II = 100000$. ........................... 68
5.18 Instantaneous $yz$ contour plot at $II = 100000$. ........................... 69
5.19 Inflow and main simulation grids. .............................................. 70
5.20 $\delta^*$ and $\theta$ for inflow and main simulations. ........................ 71
5.21 $\delta^*$ for inflow and main simulations, plotted against $Re_\theta$. .... 72
5.22 Time-averaged logarithmic boundary layer profile - main simulation. 73
5.23 Time-averaged turbulent fluctuations - main simulation. ............... 74
6.1 Example of relaminarisation caused by original LWS initial condi-
    tions. ......................................................................................... 87
6.2 Example of the alternative weighting function suggested by Liu &
    Pletcher. ...................................................................................... 89
6.3 Example of the poor control of turbulent fluctuations caused by
    the inclusion of $\gamma$ in $u'$ rescaling. ....................................... 91
7.1 Vane-style micro vortex generators on the flap of a Piper Malibu
    Meridian. ................................................................................... 100
7.2 Illustration of the effect of velocity ratio on the trajectory of the jet. 102
7.3 Illustration of a jet acting perpendicular to a boundary layer. ......... 104
7.4 Illustration of the mechanism behind the formation of the semi-
cylindrical vortical layer behind the jet at low VR. ............ 107
7.5 Orientation of a pitched and skewed jet relative to a boundary layer. 109
7.6 Velocity profile across jet. ......................................... 112
7.7 Perpendicular jet, time-averaged 3D surface plots of \( u = 0.4 \) . . 115
7.8 Perpendicular jet, \( u \) velocity \( xy \) contour fields. ..................... 116
7.9 Perpendicular jet centreline \( u \) velocity \( yz \) contour fields and velocity profiles. ........................................ 117
7.10 Perpendicular jet centreline \( v \) velocity \( yz \) contour fields and velocity profiles. ........................................ 118
7.11 Perpendicular jet centreline \( w \) velocity \( yz \) contour fields and velocity profiles. ........................................ 119
7.12 Pitched & skewed jet, time-averaged 3D surface plots of \( u = 0.4 \). 121
7.13 Pitched & skewed jet, \( u \) velocity \( xy \) contour fields. ..................... 122
7.14 Pitched & skewed jet, centreline \( u \) velocity \( yz \) contour fields and velocity profiles. ........................................ 123
7.15 Pitched & skewed jet, centreline \( v \) velocity \( yz \) contour fields and velocity profiles. ........................................ 124
7.16 Pitched & skewed jet, centreline \( w \) velocity \( yz \) contour fields and velocity profiles. ........................................ 125
7.17 Velocity contour plots & streamlines - mean \( \sqrt{u^2 + v^2} \) at the jet centreline. ........................................ 126
7.18 Vorticity contour plots - mean \( \omega_z \) at the jet centreline. ........ 128
7.19 Distributions of \( u^2 \) at the jet centreline. .......................... 129
7.20 Distributions of \( v^2 \) at the jet centreline. .......................... 130
7.21 Velocity contour plots & streamlines - mean \( \sqrt{u^2 + w^2} \) at \( y/D = 0.6 \). ........................................ 131
7.22 Vorticity contour plot - mean $\omega_y$ at $y/D = 0.6$ .... 132
7.23 Distributions of $u'^2$ and $w'^2$ at $y/D = 0.6$ .... 133
7.24 Vorticity contour plots - mean $\omega_z$ at offsets of (a) $z = 0.6D$, & (b)
        $z = 0.85D$. ... 134
7.25 Velocity contour plots & streamlines - mean $\sqrt{(u'^2 + v'^2)}$ at the jet
        centreline. .... 136
7.26 Vorticity contour plots - mean $\omega_z$ at the jet centreline. .... 137
7.27 Velocity contour plots & streamlines - mean $\sqrt{(u'^2 + w'^2)}$ at $y/D = 0.6$. .... 138
7.28 Vorticity contour plots - mean $\omega_z$ at offsets of (a)&(c) $z = 0.6D$,
        (b)&(d) $z = 0.85D$. .... 139
7.29 Velocity contour plots & streamlines for the pitched & skewed case
        - mean $\sqrt{(u'^2 + v'^2)}$ at various offsets. .... 140
C.1 VR = 0.5, perpendicular jet, 3D surface plot of $u = 0.4$ .... 170
C.2 VR = 0.5, perpendicular jet, $u$ contour fields for $xy$ plane. .... 171
C.3 VR = 0.5, perpendicular jet, $u$ contour fields and velocity profiles
        for $yz$ plane. .... 172
C.4 VR = 0.5, perpendicular jet, $v$ contour fields and velocity profiles
        for $yz$ plane. .... 173
C.5 VR = 0.5, perpendicular jet, $w$ contour fields and velocity profiles
        for $yz$ plane. .... 174
C.6 VR = 0.5, pitched & skewed jet, 3D surface plot of $u = 0.4$. .... 175
C.7 VR = 0.5, pitched & skewed jet, $u$ contour fields for $xy$ plane. .... 176
C.8 VR = 0.5, pitched & skewed jet, $u$ contour fields and velocity
        profiles for $yz$ plane. .... 177
C.9 VR = 0.5, pitched & skewed jet, $v$ contour fields and velocity
        profiles for $yz$ plane. .... 178
C.10 VR = 0.5, pitched & skewed jet, w contour fields and velocity profiles for yz plane.................. 179
D.1 Diagram of the 2D steady laminar jet configuration............. 181
D.2 Development of centreline velocity and integral thicknesses from the orifice exit......................... 192
D.3 Comparison between our theoretical model, and Schlichting's self-similar solution....................... 193
Part I

Problem Definition &

Background
CHAPTER 1

Introduction

Numerical simulations of unsteady turbulent flows are often performed using periodic boundary conditions. Fully developed, time-evolving flows, (e.g. channel flows) naturally lend themselves to the use of simple, periodic conditions on the open flow boundaries, since downstream flow can be directly re-applied at the inlet. These boundary conditions, however, are not appropriate for spatially developing flows, such as turbulent flat-plate boundary layers.

Spatially developing flows are of considerable interest to aerodynamicists; for example, recent advances in boundary-layer control technology have fueled a desire to characterise and parameterise the interaction between various vortex generating devices and the turbulent boundary layer. An unsteady numerical method, (such as Large Eddy Simulation - LES), would be ideally suited for the detailed study of the mechanisms of drag reduction or separation delay enabled by these devices. Fundamentally however, the capacity to provide accurate inflow data for these simulations is a critical precursor to the subsequent investigation of more complex flows.

Among others, Spalart (1988) and Rai & Moin (1993)) have produced DNS (Direct Numerical Simulation) models of turbulent flat plate boundary layers. More recently, Lund et al. (1998) (herein referred to as LWS) outlined an auxiliary
simulation-based LES approach. The LWS method is popular and well-cited, however recent papers (eg. Liu & Pletcher (2006), Simens et al. (2007)) have noted various issues with the original formulation, particularly regarding stability.

The research presented in this thesis concerns modifications intended to improve the LWS inflow generation technique. Full validation results have been presented, and the differences between the original formulation and our method are explored in detail. Furthermore, a flow-control test-case has been investigated, to assess the suitability of the method for providing inflow conditions for the simulation of more complex flows (specifically, the well-documented case of steady vortex generating jets acting perpendicular to a flat-plate boundary layer).

We begin with a review of inflow generation techniques appropriate for unsteady turbulent boundary layer simulations (Chapter 2). This is followed by two 'methods' chapters; Chapter 3 provides a brief description of the formulation and discretisation of the underlying LES code, and Chapter 4 describes the specific formulation of the final, successful implementation of our LWS based inflow generation technique. The next set of chapters are 'results and discussion' oriented; Chapter 5 presenting detailed validation data regarding our inflow technique, and Chapter 6 exploring and discussing the justifications for - and the efficacy of, the various modifications to the original LWS formulation.

Chapter 7 is a self-contained treatment of the steady-jet test-case, including a brief review of established knowledge about steady jets interacting with a turbulent boundary layer, an explanation of the model configuration, presentation of the results, and a discussion of the extent of their agreement with established data concerning steady jets in cross-flow. The thesis is brought to a close with a wider discussion of the further changes that could be made to the existing LES model to enable the simulation of more complex flow control devices.
CHAPTER 2

Literature Review

This review chapter is focused on the various methods available to fluid dynamicists, capable of providing inflow boundary conditions for Direct Numerical Simulations (DNS) or Large Eddy Simulations (LES) of spatially developing turbulent boundary layer flows. For a wider introduction to turbulent boundary layers, DNS and LES, the author recommends Pope (2003). The chapter begins with a brief discussion of the use of experimental data as an inflow condition for numerical simulations, and then moves on to a more rigorous treatment of the various techniques available for generating a realistic inflow condition numerically. These can range from the simple application of random fluctuations upon a mean profile, to the use of precursor simulations, and finally the use of flow rescaling and recycling techniques.

2.1 Introduction

LES of a spatially developing turbulent boundary layer requires the solution of the filtered three-dimensional, unsteady Navier-Stokes (NS) and continuity equations with a prescribed initial flow-field and boundary conditions on the six faces of the rectilinear computational domain. The inlet boundary condition is
of particular interest; the flow downstream is typically very sensitive to the time-dependent turbulent inflow conditions set at the inlet, making it necessary to provide an accurate series of time-varying velocity components that satisfy the NS and continuity equations. This inflow data can be provided by a number of different techniques, each with their own advantages and disadvantages in terms of convenience, cost and numerical accuracy.

2.2 Review of existing inflow generation methods

The approaches for generating turbulent inflow data can arguably be grouped into two categories; inflow data taken directly from experimental measurements, and synthetic inflow data generated computationally. These (admittedly arbitrary) categories again break down into a wide range of different methods, varying in complexity and accuracy. We begin with a brief discussion of experimental inflow generation, after which we focus more carefully on numerical techniques.

2.2.1 Using experimental data as an inflow condition

An uninitiated observer might suggest that ‘real’ data, sampled directly from appropriate experimental apparatus, could provide by far the most realistic and accurate data for use as an inflow condition in a numerical simulation. Assuming good flow similarity between apparatus and simulation (Reynolds numbers, non-dimensional ratios between, say, displacement thickness and the domain size), there remain a number of practical hurdles regarding temporal and spatial synchronisation.

Hot-wire anemometry, particle image velocimetry (PIV), and laser Doppler anemometry (LDA), are among some of the more popular methods of experimental velocity measurement. Hot-wire is a relatively cheap and simple technique,
2. Literature Review

with a high temporal resolution; however it is an invasive method that uses probes to take point measurements. Laser-based techniques are generally less invasive, for example LDA is another accurate point-measurement used in the same spirit as hot-wire, however it is a seeded method. One might argue that the seeding process itself still represents some degree of flow disruption, however with well chosen seeding it can be less invasive than placing probes in the flow field. PIV can be used to capture 2D planes in great detail, providing information about instantaneous flow structures. Furthermore, modern techniques allow the sampling of a time-series of velocity planes, each plane comprising a great deal of data. For given storage limits, the experimentalist often has to choose between a high temporal resolution for a short time-period, or a lower resolution for a longer period (for further reference, Adrian (1991)).

Ideally, measurements could be coherently sampled at locations corresponding to the simulation’s inlet grid points, with a temporal resolution high enough to match each simulation time-step. Realistically, computational fluid dynamicists have had to work their inflow generation method around the limitations of the measurement techniques. Various methods have been developed to address the hurdles of time-synchronisation and spatial resolution, with some success.

Druault et al. (2005) provided a fairly recent description of a technique used to couple hot wire data with an LES simulation. In the paper Druault described using a ‘brush’ of 66 hot wire probes measuring the three orthogonal velocity components, simultaneously, at a number of carefully chosen locations. These provided a high temporal resolution, and to some extent preserved coherence with respect to the large-scale structures within the flow. However the lack of spatial resolution needed to be addressed. Linear stochastic estimation (LSE) was applied to reconstruct an instantaneous velocity field from the probe data for each of the grid points of the simulation. This was coupled with proper
orthogonal decomposition (POD - Berkooz et al. (1993)) to help preserve the spectral energy content of the flow, and address the LES code’s periodic span-wise boundary conditions. Druault found that the LES simulation demonstrated a short transient region near the inlet, after which the flow quickly recovered to a realistic 3D flow structure with realistic energy profiles, which was in reasonable agreement with reference DNS data. Perret et al. (2006) is a convenient converse example, where high spatially resolved PIV data was synchronised with a Large Eddy Simulation, drawing on POD techniques again to compensate for measurements under-resolved in time, by rebuilding time-step synchronised inflow data from spectral information.

In summary, a brief review of existing literature reveals a number of practical limitations to the use of experimental data as an inflow condition for a numerical simulation. Present measurement techniques appear to constrain either the temporal or spatial resolution of the data in some way, necessitating some form of approximation to ensure synchronisation with the numerical model. In the next section, examples of stand-alone numerical inflow generation techniques are described, some of which provide very accurate statistical agreement with established experimental data.

2.2.2 Numerical generation of turbulent inflow data

"Perhaps the most straightforward approach to simulate a spatially developing turbulent boundary layer is to start the calculation far upstream with a laminar profile plus some random disturbances and then allow a natural transition to turbulence to occur."

Lund et al. (1998)

Rai & Moin (1993) is a famous account of a DNS of transition in a boundary
layer. No fluctuations were specified at the inlet, however as LWS noted, the simulation of the transition process itself was computationally costly due to the sheer length of domain required for turbulent flow to fully develop, and as such would be prohibitively expensive as a turbulent inflow generation technique. Another key DNS by Spalart (1988) skipped the transition part of the flow altogether, and simulated a fully developed turbulent boundary layer using periodic stream-wise boundary conditions. Spalart essentially took advantage the self-similarity of the boundary layer profile in the inner 'law of the wall', and outer 'defect law' regions over a short domain length. He fitted his coordinate lines to the boundary layer to minimise inhomogeneity, and applied periodic stream-wise boundary conditions, modelling a fully developed turbulent boundary layer up to $Re_\theta = 1410$. His results demonstrated excellent statistical agreement with experimental data, indeed, Spalart's work has been used to study some of the more complex coherent structures and mechanisms in turbulent flows (Robinson (1991)).

Fundamentally, despite the proven accuracy of DNS, the sheer number of grid points required to correctly resolve all structures within the flow limits the maximum Reynolds number of the DNS, and typically demands long, computationally costly simulations. Given that in our case, the objective was to generate inflow conditions for an LES simulation, one would sensibly want any precursor simulation used to be at least as fast as, and no more accurate than, the main simulation.

2.2.3 Random fluctuations

A popular, computationally cheap alternative is to simply specify a mean turbulent profile directly at the inlet of the main simulation, upon which is superimposed some form of random fluctuation. The amplitude of the fluctuations can be constrained to correspond with desired fluctuation intensities across the profile,
however it is often difficult to specify coherent phase relationships between the randomly generated fluctuations themselves. This typically results in a transient region near the inlet where organised turbulent structures develop. Le & Moin (1994) is an early example of this technique, applied to a DNS of flow over a backward facing step. A mean velocity profile was applied at the inlet, upon which random fluctuations with given moments and spectra were applied. These random fluctuations were created using a method proposed by Lee et al. (1992), which ensured that the energy content at the smallest scales was insignificant, and that the peak in the spectrum corresponded to a well-resolved wavelength. Unfortunately they found that a fairly lengthy development section needed to be used in order for coherent turbulent structures to re-appear.

"The importance of coherent structures in the dynamics of turbulent flows was demonstrated in the 1970s by the flow visualisation experiments of Kline et al. (1967) and Brown & Roshko (1974). Since then, laboratory measurements and numerical simulations have identified a multiplicity of structural elements, which have directional preferences, shapes, generation and evolutions."

Piomelli et al. (2000)

In turbulence, large structures initiate the cascade of kinetic energy to smaller structures. This is not reflected in random fluctuations applied as synthetic turbulence, the energy in the high wave numbers dissipates very quickly without sustaining or initiating real turbulence. Essentially, Le's inflow fluctuations did not contain the organised phase information required for the accurate representation of real turbulent eddies. Indeed, despite a stream-wise development section of 10 boundary layer thicknesses being used for the backward facing step simulation, subsequent tests of the inflow technique on a channel flow revealed that
almost 20 thicknesses were required to recover the correct skin friction. Furthermore Akselvoll & Moin (1995), in repeating Le's simulation using LES instead of DNS, found that the coarser LES grid did not allow the random fluctuations to develop as rapidly, and instead he had to generate fluctuations in a precursor simulation with a much higher spanwise and wall-normal grid density.

Batten et al. (2004) and Smirnov et al. (2001) are recent examples of extensions to the technique, intended to improve the coherence of the fluctuations. For example, Batten et al. (2004) demonstrated a method of random fluctuation generation based on the superposition of random sinusoidal modes, with given moments and spectra. Their approach included a method of modifying wave-numbers to yield eddies more elongated in the direction of larger Reynolds stresses, producing more realistic coherent structures. Keating et al. (2004) noted that their method again produced a fairly long transient region downstream of the inlet, before the flow was correctly resolved.

LWS argued that even if one were to accept the costly development section associated with the random fluctuation method, "it has a second, perhaps more serious problem in that it is very hard to control the skin-friction and integral thicknesses at the end of the development section (where one would really like to specify them)." Essentially, it is very hard to configure the inlet profile and fluctuations such that desired, target values of, say, skin friction or momentum thickness, are accurately reached at the end of this non-physical, transient development section. Indeed this was another motivation for Akselvoll & Moin (1995) to split the inflow calculation into a separate, precursor simulation. This enabled them to retrospectively choose a plane with the target skin friction and integral thicknesses as an inflow condition for their main simulation.
2.2.4 Precursor simulations

Akselvoll & Moin (1995) touched on a more generalised method of devoting an entire auxiliary simulation to the generation of accurate inflow data for a main simulation. Among the features of this technique are the ability to choose a convenient sample plane, with desirable skin friction and integral thicknesses, in a well resolved region of the precursor simulation. Another feature is the 'one-way' information coupling between the two simulations. This coupling could be seen as problematic from a causality point of view - clearly information cannot feed back to the inflow simulation - something that could be seen as a problem if a signal, say an acoustic wave, was emitted in the main domain. However this feature lends itself particularly well to quasi-periodic recycling methods, where one would ideally like the inflow simulation to be 'insulated' from any unusual behaviour being modelled in the main domain, to ensure a stable recycling scheme.

With the precursor approach, the inflow calculation is temporally and spatially synchronised with the main simulation, such that grid points and time-steps correspond. While it might initially appear to be a costly and complicated method of generating inflow data, this is not necessarily the case. A precursor simulation will typically require fewer stream-wise grid-points than the main simulation (meaning a faster computation) and it can be run in parallel on a second processor. Indeed, one could choose to generate data some time before, with planes stored in a 'library' for use in multiple main simulations. LWS suggested that, "The use of actual simulation data for an inlet condition allows the development section to be either reduced or eliminated altogether. The cost savings due to a reduction in the development length of the main simulation will more than offset the cost of the auxiliary simulation in most cases." Therefore on the basis that the user has access to a machine with multiple processors, the use of
a precursor domain would significantly reduce the overall cost of any subsequent main simulations by enabling the removal of grid-points otherwise required for a lengthy development section.

Kaltenbach (1993) provided an early example of a simple application of the precursor method. He used a periodic channel flow simulation to generate inflow conditions for the LES of a plane diffuser. This parallel flow simulation was fine as an inflow condition for channel flow simulations, indeed the data was fully developed and contained coherent turbulent structures. However, when Lund & Moin (1996) tried applying periodic channel flow data as an inflow condition for a spatially developing boundary layer, a number of problems were encountered. Fundamentally the parallel flow simulation lacked mean advection - the inflow boundary layer was not spatially developing. This resulted in a transient adjustment region near the inlet of the main simulation, where the boundary layer recovered the correct spatial growth characteristics.

2.2.5 Recycling

The need to develop more accurate inflow conditions for a spatially developing boundary layer prompted Lund to develop a more sophisticated approach based partly on the technique used for the Spalart (1988) DNS simulations. As mentioned earlier, Spalart had applied periodic stream-wise boundary conditions to a spatially developing boundary layer, by defining a set of coordinate lines along which stream-wise inhomogeneity was minimised. Using this 'spatially-developing' coordinate system allowed the velocity field to be treated as approximately homogeneous, and thus amenable to true periodic boundary conditions. Unfortunately this came at the expense of a number of complicated 'growth-terms' that needed to be added to a modified version of the Navier-Stokes equations.
Lund's solution was to simply modify the boundary conditions (as opposed to the whole domain). Lund chose to leave the coordinate system unadulterated, and instead re-scale and recycle the developing boundary layer, for re-application at the inlet. Essentially a quasi-periodic method, it came at the expense of Spalart's strict stream-wise periodic boundary conditions, and the subsequent ability to use a highly efficient fourier representation. Lund argued that, "This is not a concern in the context of inflow generation, however, since the recipient spatially evolving simulation will invariably use discrete operators." The beauty of Lund's technique was that any conventional cartesian LES code could be adapted - with the addition of a simple subroutine - for the purposes of inflow generation; the spatial development of the boundary layer computed directly, requiring only a simple empirical wall shear stress calculation to relate the inlet boundary to the recycling plane. Essentially the procedure resulted in a straightforward spatially evolving simulation that generated its own inflow data.

As with many novel techniques, subsequent researchers (including the author) have encountered problems when implementing the specific formulation outlined in LWS's original paper. Issues regarding spurious periodicity, poor stability and durable spanwise variations have among others, been mentioned by Sagaut (2004), Keating et al. (2004), Liu & Pletcher (2006), Ferrante & Elghobashi (2004) and most recently, Simens et al. (2007). These hurdles, and how they were addressed are the basis of the research presented in the rest of this thesis, and will be dealt with in greater detail, in subsequent chapters.
2. Literature Review

2.3 Summary of the key advantages of using LWS-based methods

- The use of a precursor simulation;

  - A precursor simulation allows a well-conditioned plane, with target integral thicknesses and Reynolds number, to be chosen for use as an inflow condition for the main simulation.
  
  - With well-conditioned inflow data, there should be little or no spatial transient near the inlet of the main domain, ensuring Reynolds numbers and integral thicknesses evolve accurately from the inlet.

  - Despite overall computational cost increasing, the two simulations can be run in parallel. Without the costly spatial transient region near the inlet, the main simulation domain can be significantly shorter than it otherwise would have been. Assuming the simulations can be run in parallel, this would result in a lower overall time-cost.

  - Data from the precursor simulation can be generated independently, and stored for later use in consecutive main simulations. This would increase the time-cost advantage of the precursor simulation with every subsequent main simulation.

- Rescaling and recycling method;

  - Enables the self-generation of an accurate spatially developing turbulent boundary layer, near the target Reynolds number, without recourse to the simulation of a laminar or transition region.

  - Transients are temporal rather than spatial.
2. Literature Review

- The LWS rescaling method has the quasi-periodic benefits of the Spalart method, without recourse to tricky co-ordinate transformations - boundary conditions are transformed, rather than the entire domain.
- Conventional cartesian codes can be adapted with a simple subroutine.
- The Inflow data is coherent and very accurate, especially when compared to simpler random fluctuation inflow conditions.

2.4 Conclusions

This literature review chapter has presented a number of methods available to the computational fluid dynamicist, that are capable of providing inflow boundary conditions suitable for the LES or DNS of spatially developing turbulent boundary layer flows. We began with a discussion of various methods of using experimental data as an inflow condition, and concluded that present measurement techniques appear to constrain either the temporal or spatial resolution of the inflow data, necessitating some form of approximation to condition the data.

This was followed by a more in-depth discussion of the various numerical inflow methods, highlighting the long and costly spatial transients that are typical of less accurate inflow. The LWS rescaling and recycling technique was then introduced as a means to provide accurate and coherent inflow data, without having to sacrifice an upstream region of the main simulation to spatial transients. Problems were noted with the original LWS formulation, and these will be discussed in more depth in later chapters. The following section of this thesis is methods oriented, and describes the specific formulation of our LES code, and the final, successful formulation of our LWS-based inflow generation technique. A discussion of, and justification for the differences between our formulation, and the original, can be found in Chapter 6.
Part II

Methods
CHAPTER 3

LES Code Formulation

The following chapter describes the Large Eddy Simulation (LES) code used as a basis for the research described in later chapters. It was originally developed and validated by Dr Yongmann Chung (University of Warwick), and is based on the formulation described in the popular Yang & Ferziger (1993) paper. A notable feature of the code is the dynamic subgrid-scale model, which enables more accurate modelling of the eddy-viscosity term, particularly in near-wall regions. We begin with a brief introduction to LES, followed by a full description of the formulation of the Yang & Ferziger (1993) scheme, with particular focus on the dynamic sub-grid scale model. This is followed by a discussion of the numerical method, in particular the fractional-step time-advancement scheme.

3.1 Introduction

Kolmogorov (1941) introduced the idea of self-similarity in turbulent flow. He suggested that the smallest scales of turbulence are universal (similar for every turbulent flow), and that they depend only on $\nu$ (the kinematic viscosity of the fluid) and $\epsilon$ (the average rate of energy dissipation per unit mass). An implication of this self-similarity is that the smaller scales are more isotropic and
homogeneous, less affected by boundary conditions than the larger ones. Thus LES methods seek to explicitly resolve large scale turbulent eddies and structures (in a similar manner to Direct Numerical Simulation), whilst the smaller-scale behaviour is implicitly accounted for by using a subgrid-scale model.

Figure 3.1 (taken from Sagaut (2004), pg. 9) is a simplified illustration of the concept behind LES. Essentially, larger structures are resolved directly, and below a certain filter-width ($\Delta$), subgrid scale structures are modelled implicitly.

### 3.2 Generalised LES formulation

The LES filter defines the transition between the large-scale resolved structures, and the sub-grid scale model. In many practical cases, the filter width is simply defined by the grid itself, meaning that structures smaller than the grid are modelled rather than resolved. Equation 3.1 provides a general definition of the filtering operation:
3. LES Code Formulation

\[ \bar{f}(x) = \int_D f(x')G(x, x'; \bar{\Delta}) \, dx' \]  

(3.1)

A filtered (or large-scale) variable is denoted by an overbar, a subgrid scale variable by an apostrophe. \( D \) represents the entire domain, \( G \) is the filter function, and \( \bar{\Delta} \) is the filter width (the wavelength of the smallest scale retained by the filtering operation). Applying it to the governing equations, we obtain the filtered equations of motion. For incompressible flow of a Newtonian fluid, these are;

\[ \frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j}(\bar{u}_i \bar{u}_j) = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j \partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} \]  

(3.3)

On first inspection, Equation 3.3 appears to be similar to the incompressible Navier-Stokes equation, however there is an extra term, \( \frac{\partial \tau_{ij}}{\partial x_j} \), which is used as an abbreviation for the filtered subgrid scale stresses;

\[ \tau_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j \]  

(3.4)

These subgrid scale stresses can be further decomposed into three parts;
\[ \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j = L_{ij} + C_{ij} + R_{ij} \]  

\textit{where,} 

\[ L_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j \]  

\[ C_{ij} = \bar{u}_i \bar{u}_j' + \bar{u}_j \bar{u}_i' \]  

\[ R_{ij} = \bar{u}_i' \bar{u}_j' \]  

\( L_{ij} \) represents the 'Leonard' stresses, interactions between resolved scales that result in subgrid-scale contributions. When a second-order finite-difference numerical scheme is used, they are of the same order as the truncation error, and can be neglected (Shaanan et al. (1975)). \( C_{ij} \) represents the cross terms, which are interactions between resolved and unresolved scales. Finally, \( R_{ij} \) represents the subgrid-scale Reynolds stresses, the interactions between small, unresolved scales.

### 3.3 Subgrid-scale model

The primary role of the sub-grid scale model is to remove energy from the resolved scales, as implied by the Kolmogorov energy cascade. A commonly used model was developed by Smagorinsky (1963). The eddy viscosity is obtained by assuming energy production and dissipation are in balance, i.e. the small scales are in equilibrium, and dissipate instantaneously all the energy they receive from the resolved scales.

\[ \nu_T = (C_s \Delta)^2 |\bar{S}| \]  

(3.9)
In this case, $C_s$ is the Smagorinsky constant, $|\tilde{S}| = (2\tilde{S}_{ij}\tilde{S}_{ij})^{0.5}$ is the magnitude of the large-scale strain-rate tensor, and as before, $\Delta$ is the filter width (often implicitly tied to the grid itself - e.g. the cube root of the cell volume).

The standard Smagorinsky formulation involves the prescription of a specific Smagorinsky constant, $C_s$, usually in the range of $0.18 \leq C_s \leq 0.23$. Unfortunately, however, it has been found that this value needed to be decreased in the presence of shear, near solid boundaries or in transitional flows. This led to researchers making undesirable ad-hoc corrections to the constant, for example forcing sub-grid scale stresses to vanish at solid boundaries in the case of near-wall flows; and modifying the stresses to take into account the anisotropy of structures in the near-wall region.

Our code uses a dynamic subgrid-scale model, (Germano et al. (1991)) that overcomes difficulties with asymptotic behaviour near boundaries by locally calculating the eddy viscosity. Essentially, the smallest resolved flow-scales are sampled, and used to locally calculate the subgrid-scale model coefficient. It removes any requirement for ad-hoc corrections, and furthermore enables some degree of backscatter (energy transfer back from the modelled small scales to the resolved scales). The following description is based on the formulation detailed in Germano et al. (1991).

The flow field is filtered twice. Firstly using a volume-averaged box filter (of width $\bar{\Delta}$, corresponding to the grid), and secondly with a test filter (of width $\bar{\Delta}$) corresponding to a coarser mesh than the grid (in our case $\bar{\Delta}/\Delta = 2$). From Equation 3.1 for the box filter;

$$\tilde{f}(x) = \int_D f(x')\tilde{G}(x, x') \, dx'$$

As mentioned, applying this filter to the Navier-Stokes equations yields the LES
filtered N-S equations (3.3), and the corresponding sub-grid stress term (3.2).

\[ \tau_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j \]

Filtering again with the coarse test filter;

\[ \tilde{f}(x) = \int_D f(x') \tilde{G}(x, x') \, dx' \]  \hspace{1cm} (3.10)

yielding similarly filtered N-S equations, and a different sub-grid stress term,

\[ T_{ij} = \bar{\tilde{u}}_i \bar{\tilde{u}}_j - \tilde{\bar{u}}_i \tilde{\bar{u}}_j \]  \hspace{1cm} (3.11)

The box, and test filtered sub-grid scale stresses are related using the 'Germano' identity, \( \mathcal{L}_{ij} \);

\[ \mathcal{L}_{ij} = T_{ij} - \tilde{\tau}_{ij} = \bar{\tilde{u}}_i \bar{\tilde{u}}_j - \tilde{\bar{u}}_i \tilde{\bar{u}}_j \]  \hspace{1cm} (3.12)

The Germano identity essentially represents the resolved turbulent stresses (the contribution to the Reynolds stresses by length-scales between the test filter and grid filter), and relates the sub-grid scale stresses \( \tau_{ij} \) and the coarser, test filter sub-grid scale stresses \( T_{ij} \), to the filtered N-S equations (3.3). Since \( \mathcal{L}_{ij} \) is resolved, the identity can be used as a basis for computing \( C_s \) explicitly.

Let \( M_{ij} \) and \( m_{ij} \) be the models for the anisotropic parts of \( T_{ij} \) and \( \tau_{ij} \);

\[ \tau_{ij} - \frac{\delta_{ij}}{3} \tau_{kk} \approx m_{ij} = -2C_s \bar{\Delta}^2 \bar{S} \bar{S}_{ij} \]  \hspace{1cm} (3.13)
\[ T_{ij} - \frac{\delta_{ij}}{3} T_{kk} \approx M_{ij} = -2C_s \tilde{\bar{\Delta}}^2 \tilde{\bar{S}} \tilde{\bar{S}}_{ij} \]  \hspace{1cm} (3.14)
where $\delta_{ij}$ is the Kronecker symbol, and,

$$\tilde{S}_{ij} = \frac{1}{2} \left( \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right), \quad (3.15)$$

$$|\tilde{S}| = \sqrt{2\tilde{S}_{ij}\tilde{S}_{ij}} \quad (3.16)$$

($\tilde{S}_{ij}$ and $|\tilde{S}|$ being of similar form)

Substituting Equations 3.13 and 3.14 into the Germano identity Equation 3.12, and rearranging, yields,

$$\mathcal{L}_{ij} - \frac{\delta_{ij}}{3} \mathcal{L}_{kk} = 2C_S (\tilde{\Delta}^2|\tilde{S}|\tilde{S}_{ij} - \tilde{\Delta}^2|\tilde{S}|\tilde{S}_{ij}) = 2C_S P_{ij}, \quad \text{where,} \quad (3.17)$$

$$P_{ij} = \tilde{\Delta}^2|\tilde{S}|\tilde{S}_{ij} - \tilde{\Delta}^2|\tilde{S}|\tilde{S}_{ij} \quad (3.18)$$

There are a number of different approaches for calculating $C_s$ from Equation 3.17, indeed the original method proposed by Germano has been superseded by an improved, least-squares approach proposed by Lilly (1992);

$$C_s = \frac{1}{2} \frac{\mathcal{L}_{ij} P_{ij}}{P_{ij} P_{ij}} \quad (3.19)$$

Equation 3.19 avoids computational singularities better than the original Germano equation. Furthermore the numerator (and therefore the sign of $C_s$) can become negative, leading to 'backscatter', or the transfer of energy upscale. This is consistent with the ability of real sub-grid-scale eddies to transfer randomness to the explicit scales.

Since $C_s$ is an instantaneous, local value, some degree of temporal and spatial
averaging is required to avoid wild variations, and numerical instability. To this end, $C_s$ is averaged across the $z$ axis, and additional weighted average is performed over nine neighbouring $xy$ grid points, (three wall normal points at boundaries).

In summary, despite the apparently complex formulation, the dynamic sub-grid scale eddy viscosity model can be simply described as the explicit computation of the Smagorinsky constant from the smallest resolved scales in the flow. This enables the Smagorinsky constant to 'auto-correct' in near-wall regions of turbulent flows without the use of an ad-hoc damping function, and is therefore a significant improvement over conventional sub-grid scale models.

### 3.4 Numerical method

In CFD, the key challenge in producing a time-accurate solution for an incompressible flow is provided by the lack of a time-derivative term in the continuity equation. This issue is usually addressed by exploiting the incompressibility assumption, and using the pressure term in the momentum equations for time-advancement. In incompressible problems, pressure does not have its usual thermodynamic meaning; theoretically, the value of the speed of sound becomes infinite. Indeed, Ferziger & Peric (2002) stated that, 'the absolute pressure is of no significance in an incompressible flow; only the gradient of the pressure (pressure difference) affects the flow'. Thus mass-conservation between time-steps is enabled by an implicit coupling between the continuity equation, and the pressure term in the momentum equations.

In a simple case, one could use an explicit time-advancement scheme that would use the pressure at the current time-step to satisfy the continuity equation at the next step. More sophisticated methods, (for example the 'Semi-Implicit Method for Pressure Linked Equations' (SIMPLE)) use an iterative scheme that
begins by guessing the velocity field at the next time-step based on the current pressure field. This estimated velocity field is then used to calculate a pressure correction, which is used as the basis for the next velocity field iteration. This process is repeated until velocity and pressure fields converge.

The Yang & Ferziger (1993) formulation used as the basis for our code applied the Kim & Moin (1985) fractional-step scheme for time-advancement - essentially a variant of the methods introduced in the late 1960s by Chorin (1969) and Temam (1969). The two-step scheme for time-advancement of the governing equations can be written as;

\[
\frac{u_i^* - u_i^n}{\Delta t} = \frac{1}{2}(3H^n_i - H^{n-1}_i) + \frac{1}{Re} \left( \frac{\delta^2}{\delta x_1^2} + \frac{\delta^2}{\delta x_2^2} + \frac{\delta^2}{\delta x_3^2} \right) (u_i^* + u_i^n) \tag{3.20}
\]

\[
\frac{u_i^{n+1} - u_i^*}{\Delta t} = -G(\phi^{n+1}) \tag{3.21}
\]

\[
D(u_i^{n+1}) = 0 \tag{3.22}
\]

where the convective terms are;

\[
H_i = \left(-\frac{\delta}{\delta x_j}\right)u_i u_j \tag{3.23}
\]

\(u^*\) represents the velocity field at the intermediate step, \(\delta/\delta x_i\) represent discrete finite-difference operators, \(\phi\) is a scalar 'pseudo-pressure' to be determined, and G and D are discrete gradient and divergence operators. Equation 3.20 uses the implicit second-order Crank-Nicolson method for the viscous terms; and the explicit second-order Adams-Bashforth method for the nonlinear, convective term. The value of \(\phi\) is computed by solving Poisson’s equation, ensuring mass conservation at the end of a complete step.

Solving 3.21 for \(u^*\) and substituting back into 3.20 would demonstrate that

\[
p = \phi + \frac{1}{2}(\frac{\Delta t}{Re})\nabla^2 \phi \tag{3.24}
\]

plays the role of pressure, making a clear distinction between
3. LES Code Formulation

\( \phi \) and \( p \). Setting \( \phi + \frac{1}{2}(\frac{\Delta t}{2}Re) \nabla^2 \phi \) at the \( n+1 \) time-level to be equal to the pressure at the \( n^* \) level suggests that Equation 3.21 is second-order accurate in \( \Delta t \).

To carry out the convection-diffusion step, Equation 3.20 can be re-written as follows,

\[
(1 - A_1 - A_2 - A_3)(u^{*}_{i} - u^{n}_{i}) = \frac{\Delta t}{2}(3h^{n}_{i} - h^{n-1}_{i}) + 2(A_1 + A_2 + A_3)u^{n}_{i}
\]

where,

\[A_n = (\Delta t/2 Re)(\delta^2/\delta x^2_n); \quad n = 1, 3\]  \hspace{1cm} (3.25)

Solution of Equation 3.24 would necessitate the computationally costly inversion of a large, sparse matrix. As such, the terms on the left hand side are approximated via an \( O(\Delta t^3) \) factorisation, enabling the inversion of far less costly tri-diagonal matrices. This results in the following equation;

\[
(1 - A_1)(1 - A_2)(1 - A_3)(u^{*}_{i} - u^{n}_{i}) = \frac{\Delta t}{2}(3h^{n}_{i} - h^{n-1}_{i}) + 2(A_1 + A_2 + A_3)u^{n}_{i}
\]

In the code, convective terms were treated using an explicit 3rd order Runge-Kutta method. Poisson's equation is solved on a staggered grid (Harlow & Welch (1965)) to avoid the explicit implementation of boundary conditions for \( \phi \). (In staggered grids, velocities are defined at the centers of cell faces, the pressure is defined at the cell center, this helps overcome difficulties with pressure-velocity coupling that would otherwise exist in a non-staggered mesh - see Figure 3.2.)

The most obvious difference between and pressure-correction methods (eg. SIMPLE) and fractional-step methods, is that whilst the former involves a num-
Figure 3.2: Three dimensional illustration of a cell in a staggered grid.

ber of pressure-correction iterations within each time-step, the latter solves the pressure equation only once - a stability enhancement in unsteady simulations.

3.5 Summary of the key features of our LES code

- Finite volume approach
- Second-order central differencing for spatial discretisation, staggered grid
- Fractional-step time-advancement
3. LES Code Formulation

- Explicit 3rd order Runge-Kutta for convective terms
- Implicit Crank-Nicolson for viscous terms
- Dynamic sub-grid scale eddy-viscosity model

The original code has been successfully used for numerous investigations, including, (among others) transitional flows (Chung et al. (1997)), complex turbulent channel flows (Chung & Sung (2001)), and simulations of unsteady, impinging jets (Chung et al. (2002)). Given the extensive list of prior publications in which this LES code has been applied, validation studies presented in this thesis have been focused on the results produced by the inflow generation technique itself. Comparisons between well established turbulent boundary layer data, and our boundary layer model can be found in Chapter 5.

3.6 Conclusion

This chapter has detailed the formulation, and numerical discretisation of the LES code used as a basis for the work presented in later chapters. The dynamic subgrid-scale model has been discussed, in particular, its suitability for modelling the near-wall region of a boundary layer due to the automatic adjustment of the Smagorinsky constant. This was followed by a description of the fractional-step time-advancement scheme. Finally, a summary of notable code features has been presented.

The next chapter explicitly details the final, successful formulation of our LWS-based inflow generation technique.
CHAPTER 4

Turbulent Inflow Generation

4.1 Introduction

The following chapter describes the final, fully-validated formulation of our modified, Lund et al. (1998) based inflow generation technique. We begin with a description of the conditions used to initialise the flow field at the start of the simulation, followed by a detailed description of the rescaling algorithm itself. There is a discussion of the need to iterate the calculation of the friction velocity, $u_r$, at the inlet to ensure a stable inlet boundary-layer thickness, and a description of a successful technique for correcting spurious durable spanwise flow variations. A section detailing the synchronisation of the inlet simulation with the main simulation domain is included, and the chapter is concluded with a summary of the key features of the inflow generation scheme.

A Fortran 77 copy of this subroutine has been included in Appendix B, Chapter 5 compares data produced by the subroutine to well established results, and Chapter 6 discusses and justifies the major differences between our formulation, and the original LWS scheme.
4. Turbulent Inflow Generation

4.2 Concept

"The heart of our method is a means of estimating the velocity at the inlet plane, based on the solution downstream."

Lund et al. (1998)

Figure 4.1 is a simple 2D illustration of the basic concept behind the new rescaling method. Essentially, a $yz$ plane of the 3D velocity field was sampled at a position 40% along the $x$-axis. The $u$, $v$, and $w$ velocities within the plane were rescaled in terms of the law of the wall for the inner boundary layer, and the defect law for the outer region. These rescaled profiles were then combined using a weighting function, and reintroduced as 'rescaled inflow' at the inlet (represented by 'A' in the figure). Once the flow was fully developed, a sample plane ('B')
was used to provide inflow data for the main simulation.

The rescaling subroutine was called at the end of the final iteration of the fractional-step solver in the code, in order to provide fresh inflow boundary conditions for the next time-step.

4.3 Initial conditions

At initialisation, an approximate u velocity flow field was built around a simple mean profile provided by the Spalding (1961) law (Equation 4.1), growing in a stream-wise direction, according to the increase in $Re_x$.

$$y^+ = u^+ + 0.1108 \left[ e^{0.4u^+} - 1 - 0.4u^+ - \frac{(0.4u^+)^2}{2!} - \frac{(0.4u^+)^3}{3!} - \frac{(0.4u^+)^4}{4!} \right] \quad (4.1)$$

where,

$$y^+ = y \sqrt{\frac{\tau}{\mu}} \quad (4.2)$$

$$u^+ = u \sqrt{\frac{\rho}{\tau}} \quad (4.3)$$

The Spalding law was applied in LWS and provided a reasonable near-wall approximation to a time-averaged turbulent boundary layer profile. At $y$ values greater than $\delta^{99\%}$ the free-stream velocity was imposed.

The initial random fluctuation intensities were stepped to roughly match the intensity profiles demonstrated by Spalart (1988) and Klebanoff (1955); such that the turbulent intensity within the boundary layer peaked at $y/\delta = 0.05$, then progressively died away towards the outer boundary layer (see Figure 5.9).
Equations 4.4, 4.5 and 4.6, (fluctuation approximations determined through a process of trial and error), were used to provide reasonably realistic $u$ fluctuations about the mean Spalding profile, and about zero for $v$&$w$. Figure 4.2 shows an example of the resulting $u$ velocity contour along the stream-wise centreline at initialisation (of note is the realistic growth in the stream-wise direction). Figure 4.3 is an example of instantaneous velocity profiles at initialisation for $u$&$v$ at

![Figure 4.2: Example of u contour field at initialisation.](image)
a randomly chosen \( xz \) location. It will be demonstrated in Section 6.3.2 that this modified method of flow field initialisation results in a significantly faster convergence on the desired time-averaged skin friction, velocity and fluctuation profiles, than the more simplistic approach suggested by LWS.
4.4 Basic rescaling algorithm

Firstly, a $yz$ velocity plane from the 3D velocity field was taken 40% along the $x$-axis. This velocity field, $u^{n+1}$ was averaged in a spanwise direction, $(u^{n+1})_z$, and then combined with a running time-average, $U^n$ (Equation 4.7).

$$U^{n+1} = \frac{\Delta t}{T}(u^{n+1})_z + (1 - \frac{\Delta t}{T})U^n$$  \hspace{1cm} (4.7)

(Where $(\cdot)_z$ denotes spanwise averaging, $t$ is time, $\Delta t$ is the time-step, and $T$ is a characteristic timescale for the averaging interval.)

The characteristic timescale, $(T$, - time-averaging interval) varied throughout the simulation (See Figure 4.4), with a fixed timescale for a period near the
start of the simulation, intended to allow initial starting transients to settle. The characteristic timescale subsequently increased, thus 'fixing' the fully-developed time-averaged velocity profile.

\[
T = t \quad \text{for} \quad t < 80\delta^*/u_\infty \tag{4.8}
\]

\[
T = 80\delta^*/u_\infty \quad \text{for} \quad 80\delta^*/u_\infty \leq t < 800\delta^*/u_\infty \tag{4.9}
\]

\[
T = 80\delta^*/u_\infty + t - 800\delta^*/u_\infty \quad \text{for} \quad t > 800\delta^*/u_\infty \tag{4.10}
\]

This spanwise and time-averaged velocity profile was then subtracted from the actual velocity field, yielding a planar array of velocity fluctuations \(u'_i(y, z, t)\).

\[
u'_i(x, y, z, t) = u_i(x, y, z, t) - UU(y) \tag{4.11}
\]

\(u_r, \delta^*\) and \(\theta\) were required for the rescaling operation, both at the recycle point and at the inlet. These were calculated from \(U^{n+1}\) at the recycle point, where;

\[
u_i, \text{resc} = \sqrt{\nu(\partial U/\partial y)_{wall}} \tag{4.12}
\]

\[
\delta^*_\text{resc} = \int_0^\infty \left(1 - \frac{U}{U_\infty}\right) dy \tag{4.13}
\]

\[
\theta\text{resc} = \int_0^\infty \frac{U}{U_\infty} \left(1 - \frac{U}{U_\infty}\right) dy \tag{4.14}
\]

The problem would be over-determined if we were to independently fix all of the above parameters at the inlet, thus \(\delta^*_{\text{init}}\) was arbitrarily set to 1.0, and the remaining values were calculated (\(\delta^* = 1.0\) was a convenient choice since the simulation length-scales were non-dimensionalised with respect to the inlet displacement thickness).

It was assumed that the shape function \((H = \delta^*/\theta)\) would remain constant across the rescaling region (since \(Re_{\delta^*}\) would be expected to change by a negligible
amount over such a short distance), and as such the following equations were used to calculate the remaining inlet parameters;

\[
\begin{align*}
\frac{\theta_{\text{resc}}}{\theta_{\text{init}}} &= \frac{\theta_{r, \text{init}}}{\theta_{r, \text{resc}}} , \quad \text{and} \\
ur_{r, \text{init}} &= ur_{r, \text{resc}} \left( \frac{\theta_{\text{resc}}}{\theta_{\text{init}}} \right)^{1/[2(n-1)]} , \quad n = 5
\end{align*}
\]  

Equation 4.16 can be derived from standard power-law approximations, and is similar to the Ludwig & Tillmann (1949) correlation.

The mean flow was then rescaled according to the defect law in the outer region of the boundary layer, and the law of the wall in the inner region. These dictate that the inner and outer regions at the recycle point and the inlet are related via

\[
\begin{align*}
U_{\text{init}}^{\text{inner}} &= \gamma U_{\text{resc}}(y_{\text{init}}^+) \\
U_{\text{init}}^{\text{outer}} &= \gamma U_{\text{resc}}(\eta_{\text{init}}) + (1 - \gamma) U_\infty
\end{align*}
\]

where (from Equation 4.16);

\[
\gamma = \left( \frac{\theta_{\text{resc}}}{\theta_{\text{init}}} \right)^{1/[2(5-1)]} = \left( \frac{ur_{r, \text{init}}}{ur_{r, \text{resc}}} \right) \eta = y/\delta^*
\]  

Thus the inner region Equation (4.17) described the mean velocity at the recycle point expressed as a function of \(y_{\text{resc}}^+\) and mapped via a linear interpolation to the corresponding \(y_{\text{init}}^+\) grid points at the inlet. For the outer region (Equation 4.18), the mean velocity was expressed as a function of \(\eta_{\text{init}}\) and interpolated to fit the corresponding \(\eta_{\text{init}}\) points at the inlet.
4. Turbulent Inflow Generation

The mean vertical velocity was assumed to scale in a similar way, such that:

\[ V_{\text{inner}}^{\text{init}} = V_{\text{resc}}(y_{\text{init}}^+) \]  \hfill (4.21)

\[ V_{\text{outer}}^{\text{init}} = V_{\text{resc}}(\eta_{\text{init}}) \]  \hfill (4.22)

Scaling relations were considered unnecessary for the spanwise velocity, (it should theoretically be zero in the mean); however problems were encountered regarding durable spanwise error, the solution to this issue is described in Section 4.6.

The \( u \) velocity fluctuations at the recycle point were related to those imposed at the inlet via;

\[ (u'_i)_{\text{inner}}^{\text{init}} = (u'_i)_{\text{resc}}(y_{\text{init}}^+, z, t) \]  \hfill (4.23)

\[ (u'_i)_{\text{outer}}^{\text{init}} = (u'_i)_{\text{resc}}(\eta_{\text{init}}, z, t) \]  \hfill (4.24)

Note that when compared to the original LWS formulation, \( \gamma \) has been omitted from our calculation of \( u' \). Liu & Pletcher (2006) noted the tendency of the original LWS formulation to overproduce RMS turbulent fluctuations. Indeed it was found that in our case, a fixed \( \gamma = 1.0 \) provided a more stable and consistent time-averaged fluctuation intensity profile. This is explored in more detail in Section 6.3.5.

Finally the various inner and outer \( u \& v \) profiles were brought together in a composite velocity profile that was approximately valid across the entire boundary layer, by means of a weighted average of the inner and outer profiles (See Figure 4.5). Liu & Pletcher (2006) suggested a more accurate weighting function, (described in 6.3.4). However given that no real improvement was gained from its implementation, the author chose to keep the original formulation.
4. Turbulent Inflow Generation

Figure 4.5: Weighting function.

\[
(u_i)_{init} = \left[ (U_i)_{init}^{\text{inner}} + (u_i^{\prime})_{init}^{\text{inner}} \right] (1 - W(\eta_{init})) \\
+ \left[ (U_i)_{init}^{\text{outer}} + (u_i^{\prime})_{init}^{\text{outer}} \right] W(\eta_{init})
\]

(4.25)

Where;

\[
W(\eta/8) = \frac{1}{2} \left\{ 1 + \tanh \left[ \frac{\alpha(\eta/8 - b)}{(1 - 2b)\eta/8 + b} \right] / \tanh(\alpha) \right\}
\]

(4.26)

[\alpha = 4 and \( b = 0.2 \). Thus \( W(\eta/8) \) is 0.0 at \( \eta/8 = 0 \), 0.5 at \( \eta/8 = b \), and 1.0 at \( \eta/8 \geq 1.0 \).]
4. Turbulent Inflow Generation

4.5 Iteration of $u_{r, init}$ to ensure $\delta_{init}^* = 1.0$

Equation 4.16 provided a reasonable initial estimate for $u_{r, init}$ at each time-step. However it was found that the resultant $\delta_{init}^*$ was rarely perfectly rescaled to the target value of 1.0 by the end of Equation 4.25. Therefore the whole rescaling process was iterated a number of times with improved values of $u_{r, init}$ substituted for Equation 4.16 until the desired $\delta_{init}^*$ was achieved. Essentially this method involved stretching the composite average velocity profile (from a version of Equation 4.25 that excluded $u'$) in the $y$ axis until $\delta_{init}^* = 1.0$, then measuring the resulting, improved $u_{r, (init)}$, and using that value for the next iteration. It was found that $\delta_{init}^*$ usually converged on the target displacement thickness within 3 or 4 iterations (see Figure 5.5 in the next chapter). Note that LWS suggested a similar technique of iterating $\delta$ until their target $\theta$ was achieved.

4.6 Correction of durable spanwise mean flow variations

Spalart et al. (2005), among others, noted a tendency of the LWS method to generate spurious durable spanwise variations in the mean flow. Any tendency of the flow to shift sideways at the recycling plane would automatically be reintroduced at the inlet, compounding the effect. Over a large number of time-steps this error would build until it exceeded the LWS formulation's ability to damp natural spanwise diffusion, and the spurious spanwise flow would become the dominant behaviour. Our simulations appeared to be particularly sensitive to this effect - essentially invalidating LWS's assumption that spanwise velocity would remain zero in the mean, and crippling the stability and accuracy of the simulation. Figure 4.6 illustrates a number of approaches that attempted to disrupt this durable spanwise behaviour, whilst (critically) maintaining the spatial coherence of the recycling scheme.
Figure 4.6: Methods employed to disorganise durable spanwise fluctuations in the mean flow.
4. Turbulent Inflow Generation

The novel 'inflow mirroring' technique was found by the author to be by far the most effective approach for correcting these spurious variations, leading to very stable time-averaged velocity and rms fluctuation profiles. Essentially it involved taking the planar flow field at the recycle point, rescaling, then spanwise mirroring point for point the velocity field to be introduced at the inlet. A coherent clockwise rotating flow structure at the recycle plane would produce a perfectly mirrored coherent anticlockwise rotating structure at the inlet, self-correcting any spurious spanwise behaviour.

From the rescaled inlet velocity field calculated at the final iteration of Equation 4.25;

\[
\begin{align*}
    u(y, z, t)_{\text{mirror, init}} &= u(y, (W - z), t)_{\text{inlt}} \\
    v(y, z, t)_{\text{mirror, init}} &= v(y, (W - z), t)_{\text{inlt}} \\
    w(y, z, t)_{\text{mirror, init}} &= -w(y, (W - z), t)_{\text{inlt}}
\end{align*}
\]

Where \( W \) is the domain width. The new mirrored values for \( u, v \& w \) were applied at the inlet instead of the originally calculated velocity field. Note that the scheme is consistent and compatible with the spanwise \( w \) offset caused by the staggered grid, and also that \( w \) has to be negative to ensure spatial coherence once mirrored.

Spalart’s inflow shifting technique (taking advantage of the periodic spanwise boundary conditions) is discussed in more detail in Section 6.3.6; along with some of the theories regarding the causes of durable spanwise flow variation.
Figure 4.7: Exaggerated illustration of time synchronisation between the inflow and main simulations.

4.7 Main simulation domain

The main simulation was initialised with random fluctuations about a mean Spalding profile, in a similar manner to the inflow generation subroutine (see Section 4.3). Boundary layer thicknesses were adjusted at the inlet to match the chosen sample plane from the inflow simulation, growing in a streamwise direction according to $Re_x$. At every time-step of the inflow simulation, complete velocity fields from the chosen sample plane were saved to disk. This data was subsequently read in to the main simulation for use as the inflow boundary condition.
Spatial synchronisation was ensured by enforcing identical $xz$ grid points for both simulations, however temporal synchronisation required more careful treatment.

The underlying LES code used for these simulations featured adaptive time-stepping. In most cases it was found that time-steps remained constant throughout the duration of the simulation (indeed the inflow generation simulations were seen to maintain a constant time-step). However a number of the steady jet in cross flow test-cases (see Chapter 7) occasionally provoked a slight reduction in time-step. The simulation was configured with a fairly high $y$ axis grid compression, to ensure sufficient resolution in the near-wall region of the boundary layer. Theoretically, a perpendicular jet issuing across these heavily compressed cells could require a lower time-step to prevent violation of the CFL stability criterion.

Figure 4.7 is an illustration of the technique (exaggerated for clarity) that was used to ensure temporal synchronisation. Essentially, an instantaneous inflow plane would be 'held over' for a time-step when out of synchronisation with the main simulation domain. An admittedly clumsy and unrefined approach, a more complex interpolation-based method was deemed unnecessary given the low occurrence of adjustments.
4.8 Summary of our modified inflow generation technique

Differences between this and the original formulation are highlighted in bold. An equivalent summary of the original LWS formulation can be found in Appendix A for comparison.

1. Inflow simulation flow-field is initialised with a spatially developing Spalding profile, with graduated random fluctuations (Section 4.3).

2. Simulation begins, rescaling subroutine called at the end of each time-step.

3. Temporally, and spatially averaged boundary layer profile is sampled at the recycle plane (Equation 4.7), and instantaneous turbulent fluctuations are obtained (Equation 4.11).

4. $\delta_{resc}^*, \theta_{resc} & u_{r, resc}$ calculated from the mean profile (Equation 4.12).

5. $\delta_{inlt}^* = 1.0$ fixed, $u_{r, inlt}, \theta_{inlt}$ calculated for the inlet (Equation 4.16).

6. $\gamma = \frac{u_{r, inlt}}{u_{r, resc}}$, $\eta = \frac{\theta_{inlt}}{\theta_{resc}}$: are produced (Equation 4.19) for use in step 7.

7. $U$, $u'$ and $v$ velocities at the recycle plane are mapped to those at the inlet via a linear interpolation. Law of the wall for the inner boundary layer (Equation 4.17), and defect law for the outer region (Equation 4.18).

8. Composite velocity profile produced (Equation 4.25).

9. Resultant $\delta_{inlt}^*$ is measured from the composite profile, and stretched such that $\delta_{inlt}^* = 1.0$, $u_{r, inlt}$ is adjusted accordingly. This new $u_{r, inlt}$ is substituted into step 5, and the process is iterated until resultant composite profile produces $\delta_{inlt}^* = 1.0$ at step 8 (Section 4.5).

10. The final inlet profile is mirrored, point for point, to correct spurious spanwise flow variations (Section 4.6).
4. Turbulent Inflow Generation

4.9 Rescaling flow-chart

1. Initialisation.

2. Call rescaling subroutine at the end of each time-step.

3. Sample velocity profile at recycle plane,
   - obtain temporal and spatial average, $U$.
   - obtain instantaneous fluctuations, $u'$. 

4. Calculate $\delta^*$, $\theta$, $u_r$ from the mean profile for the recycling plane.

5. For a fixed $\delta^*=1.0$ at the inlet, calculate $\theta$, $u_r$.

6. Use these values to calculate $\gamma$ and $\eta$.

7. Map $U$, $u'$ and $v$ velocities from recycle plane to inlet plane, via linear interpolation,
   - law of wall for inner.
   - defect law for outer.

8. Produce a composite velocity profile.

9. Measure resultant $\delta^*$, adjust such that $\delta^*=1.0$, and calculate adjusted $u_r$ at the inlet.
   Iterate until measured $\delta^*=1.0$.

10. Spanwise mirror of inlet profile, apply as inflow condition for next timestep.

Figure 4.8: Flow-chart illustrating the rescaling process.
4.10 Conclusions

This chapter has detailed the fully-validated formulation of our modified, LWS-based inflow generation technique. We began with a description of the initial conditions, moving on to a discussion of the algorithm itself, the need to iterate the calculation of the composite inflow profile to ensure a target inlet displacement thickness. A description of our novel inflow-mirroring technique was included, used to suppress spurious durable spanwise flow variations. Finally a brief treatment of the method used to ensure spatial and temporal synchronisation between the inflow simulation and the main simulation was provided. The chapter was closed with a step-by-step summary of the inflow generation process.

Chapter 6 justifies and discusses in detail, the various differences between the original formulation, and our method. Particular attention will be given to our substitution of $\delta^*$ for $\delta$ in the rescaling formulation, our improved initial conditions, our modifications to the rescaling of $u'$, and most importantly, the issues regarding the location of the recycling plane, and it's effect on spurious durable spanwise variations.

The next part of this thesis is 'Results & Discussion' oriented, beginning with a full set of validation results demonstrating the efficacy of our modified turbulent inflow technique (Chapter 5). As mentioned, this is followed by Chapter 6 discussing the differences between formulations, and finally we present a steady jet in cross-flow test-case (Chapter 7).
Part III

Results & Discussion
CHAPTER 5

Validation of the Inflow Condition

5.1 Introduction

The aim of this chapter is to demonstrate the efficacy and accuracy of our modified inflow generation technique, described in Chapter 4. We begin by specifying the configuration of the inflow simulation, (Reynolds number, dimensions, grid, CFL number), then results are presented showing its temporal development. Time-averaged validation data for the inflow generation simulation is presented, typically compared to the well-cited Spalart (1988) DNS simulation, or LWS' results. A grid refinement case is presented, in addition to some instantaneous contour plots intended to demonstrate the generation of realistic near-wall turbulent structures. Finally, fully developed main simulation data is presented for a flat plate boundary layer, making use of the data produced by the inflow simulation. The chapter is brought to a close with a summary of some of the key plots.
5. Validation of the Inflow Condition

5.2 Simulation parameters

Length-scales were non-dimensionalised with respect to $\delta_{inl}$, velocities with respect to $U_{\infty}$; $Re_\delta = \frac{U_{\infty}\delta}{\nu} = 2000$ at the inlet, where $\nu$ is the kinematic viscosity. The inflow simulation domain had dimensions $64\delta_{inl} \times 24\delta_{inl} \times 4\pi\delta_{inl}$, (roughly equivalent to $8\delta_{inl} \times 3\delta_{inl} \times 1.6\delta_{inl}$) with a corresponding grid density of $100 \times 45 \times 64$ in streamwise ($x$), wall normal ($y$) and spanwise ($z$) directions respectively. The mesh was uniform in $x$&$z$, with hyperbolic tangent stretching in $y$ to ensure sufficient grid resolution at the wall. Given a resultant $u_{r,inl} \approx 0.046$ this yielded a mesh resolution, $\Delta x^+ \approx 59$, $\Delta y_{wall}^+ \approx 1.2$, and $\Delta z^+ \approx 18$, values entirely consistent with Piomelli & Chasnov (1996);

"To represent accurately the structures in the near-wall region, the first grid point must be located at $y^+ < 2...$ and the grid spacing must be of order $\Delta x^+ \approx 50 - 150$, $\Delta z^+ \approx 15 - 40."$

Piomelli & Chasnov (1996)
5. Validation of the Inflow Condition

The boundary conditions at the top surface of the domain were;

\begin{align}
  u &= U_\infty \\
  \frac{\partial v}{\partial y} &= 0 \\
  w &= 0
\end{align}

(5.1) (5.2) (5.3)

The spanwise domain boundary was periodic, and the exit plane used a standard convective boundary condition augmented with a streamwise $u$ velocity correction to ensure global mass conservation.

The CFL number ($= \frac{u\Delta t}{\Delta x}$ - a condition that ensures sufficiently small time-steps to maintain cell continuity) was set at $0.9\sqrt{3.0}$, and time-averaged data was sampled over 5000 time-steps at a plane located at the spanwise centreline for streamwise data, and $x = L/2$ (where $L$ is the length of the domain in the $x$ direction) for wall-normal velocity profile data. Simulations were typically run for up to 100000 time-steps.

5.3 Temporal flow development at the inlet

Figure 5.2 shows the temporal development of the mass flow at the convective boundaries of the domain. From initialisation, the simulation took roughly 10000 time-steps ($\approx \Delta T = 400$) to settle to a fairly stable mass flux. Given the growth of the boundary layer in a streamwise direction, one would expect mass flux at the inlet to exceed that at the exit plane, with the sum of the exit plane and upper boundary fluxes exactly equal to that at the inlet, (thus demonstrating a conservative simulation). This can be seen to be the case. The stability of the inlet mass flow is due to the rescaling scheme accurately fixing $\delta^* = 1.0$. Anderson (1991) described $\delta^*$ as being an 'index proportional to the "missing
mass flow" due to the presence of the boundary layer' (pg. 715). Essentially, iterating the calculation of the inlet flow field to accurately converge on $\delta^* = 1.0$ ensured that the mass flow across the inlet boundary remained constant.

Figure 5.3 shows the development of $u_T$ at the inlet plane. Again, one can see that approximately 10000 time-steps ($\approx \Delta T = 400$) were required for $u_T$ to settle on a value of $\approx 0.0465$. This is consistent with LWS, however as a secondary check;

Using the 7th power law an estimation of $u_T$ (from Houghton & Carpenter (2003) pg. 418),

$$C_f = \frac{\tau_w}{\frac{1}{2} \rho U^2 \infty} = \frac{0.0468}{Re_\delta^{-\frac{1}{4}}}$$  \hspace{1cm} (5.4)
Figure 5.3: Development of \( u_r \) at the inlet.

(for a flat-plate turbulent boundary layer), also,

\[
 u_r = \sqrt{\frac{\tau_w}{\rho}} 
\]  \hspace{1cm} (5.5)

substituting and rearranging;

\[
\frac{u_r^2}{\frac{1}{2}U_\infty^2} = \frac{0.0468}{Re_\delta^{\frac{1}{4}}} 
\]  \hspace{1cm} (5.6)

\[
\Rightarrow u_r = \sqrt{\frac{0.0234U_\infty^2}{Re_\delta^{\frac{1}{4}}}} 
\]  \hspace{1cm} (5.7)

or,

\[
\frac{u_r}{U_\infty} = \sqrt{\frac{0.0234}{Re_\delta^{\frac{1}{4}}}} 
\]  \hspace{1cm} (5.8)
5. Validation of the Inflow Condition

We know, from the 7th power law $\delta^* \approx 0.125\delta$, thus $Re_\delta^* \approx 0.125Re_\delta$;

$$\Rightarrow u_\tau = \sqrt{\frac{0.0234U_\infty^2}{(8Re_\delta^*)^4}}$$  \hspace{1cm} (5.9)

In our case $Re_\delta^* = 2000$, thus the 7th power law estimates $\frac{u_\tau}{U_\infty} \approx 0.046$, a value consistent with our results.

Figure 5.4 shows the development of $\gamma$ over time. Again reflecting a short period of development at the start of the simulation, $\gamma$ settled to a value of around 1.017. Relating this back to Equation 4.19, and performing a 7th power
law estimation again (Equation 5.9), this yields;

$$\gamma = \frac{0.0234 U_\infty^2}{(8 Re_{\delta^{*},\text{init}})^{\frac{1}{4}}}$$  (5.10)

simplifying;

$$\gamma = \left[\frac{Re_{\delta^{*},\text{rec}}}{Re_{\delta^{*},\text{init}}}\right]^{\frac{1}{6}} = \left[\frac{\delta^{*}_{\text{rec}}}{\delta^{*}_{\text{init}}}\right]^{\frac{1}{6}}$$  (5.11)

Given a recycle point placed 40% along the length of the x axis, the distance from the inlet to the recycle point would be $25.6 \delta^{*}_{\text{init}}$.

![Figure 5.5: Number of iterations required to achieve $\delta^{*}_{\text{init}} = 1.0$.](image)
5. Validation of the Inflow Condition

From Houghton & Carpenter (2003) pg. 419;

\[
\delta^* = \frac{0.0479x}{(Re_x)^{\frac{1}{5}}} = \frac{0.0479x}{(\frac{x U_{\infty}}{\nu})^{\frac{1}{5}}} = \frac{0.0479x^{\frac{4}{5}} \nu^{\frac{1}{5}}}{U_{\infty}^{\frac{1}{5}}} \tag{5.12}
\]

Applying our non-dimensional parameters and solving yields \(x_{\text{init}} = 298.4 \delta_{\text{init}}^*\).
Thus \(x_{\text{recy}} = 298.4 \delta_{\text{init}}^* + 25.6 \delta_{\text{init}}^* = 324 \delta_{\text{init}}^*\), as such \(\delta_{\text{recy}}^* = 1.07\). Applying to Equation 5.11 yields \(\gamma = 1.013\), reasonable agreement with our measured \(\gamma\).

Figure 5.5 refers directly to the number of iterations required to ensure \(\delta_{\text{init}}^* = 1.0\), (mentioned in Section 4.5). It can be seen that for at each time-step no more than 4 iterations were required to converge on \(\delta_{\text{init}}^* = 1.0\).

Figure 5.6 shows the development of \(\delta^*\) and \(\theta\) at the inlet over time. Again one can see a developmental period of approximately 10000 time-steps, after which

![Figure 5.6: Development of \(\delta_{\text{init}}^*\) and \(\theta_{\text{init}}\) over time.](image)
momentum thickness is correctly resolved. $\theta$ and $\delta^*$ are instantaneous spanwise-averaged values, hence the unsteady distribution of $\theta$ about a mean of $\approx 0.7$. This is reasonably consistent with a 7th power law estimation (Houghton & Carpenter (2003)) pg. 419) where $\delta^* = 0.125\delta$ and $\theta = 0.0973\delta$, implying $\theta = 0.78\delta^*$.

5.4 Time-averaged validation data

Figure 5.7 shows the mean boundary layer velocity profile at $x = L/2$, spanwise and time-averaged over 5000 time-steps. Other than demonstrating that our boundary layer data has a realistic 'textbook' turbulent velocity profile, it can be seen that the Spalart validation profiles for $Re_\theta = 670$ and $Re_\theta = 1410$ are fairly similar. Of greater interest is the following log-log plot of $y^+$ vs $u^+$, which gives

![Figure 5.7: Time-averaged boundary layer profile.](image)
5. Validation of the Inflow Condition

Figure 5.8: Time-averaged logarithmic boundary layer profile.

A much better impression of the accuracy of the various wall regions. Please note that lines represent validation data, and points represent the author's own data.

Figure 5.8 shows good agreement with Spalart's DNS data, slightly deviating towards the outer layer, but otherwise providing agreement in the viscous wall region \( y^+ < 5 \) and log-law regions \( y^+ > 30, y/\delta < 0.3 \). It is interesting to note that the LWS method over-predicts the log-law region, they argued that,

"This defect is a common feature of simulations using finite-difference methods on relatively coarse meshes and is not related to the rescaling approach used in the inflow generation process."

The author suspects that the coarse mesh argument accounts for the slight deviation that can be seen at higher values of \( y^+ \). Essentially the fairly high hyperbolic
grid compression factor used to ensure high grid density in the near-wall region would reduce the number of grid points available for the outer layer and free-stream.

Figure 5.9 shows time-averaged velocity fluctuation profiles at the same streamwise location. It shows good agreement with Spalart's DNS data, however a slight discrepancy can be seen in $u'$ at the fluctuation peak in the near-wall region. Note that LWS demonstrated a similar discrepancy, arguing that, 'it is related to the numerical method and not the rescaling procedure'. The use of LES instead of DNS involves some degree of approximation for the smaller, sub-grid scale flow structures, which enables the modelling of higher Reynolds number flows. Some loss of accuracy therefore would be expected in the near-wall region, where small-scale structures are prevalent. One might argue that this could be improved by
increasing the grid density at the wall, however it could also be argued that an LES simulation with a fine enough grid resolution is essentially a DNS. It can be seen from Figure 5.8 that the grid density at the wall is sufficient for good time-averaged velocity profile agreement in the viscous sub-layer, perhaps at the expense of perfect agreement toward the free stream. It was decided that any further increase in grid density at the wall was surplus to requirements, given the corresponding impact on free-stream grid density.

Figure 5.10 shows the development of $\delta^*$ and $\theta$ over the length of the domain. We know from earlier that $x_{init} = 298.4\delta_{init}^*$, and the length of the domain is $\Delta x = 64\delta_{init}^*$. Thus at the end of the domain $x_{end} = 362.4\delta_{init}^*$, applying Equation 5.12 yields $\delta_{end}^* = 1.17$, consistent with our results. One can note the non-linearity towards the end of the domain due to the outflow conditions of the simulation.
5. Validation of the Inflow Condition

Of much greater interest is the next Figure (5.11), showing $\delta^*$ plotted against $Re_\theta$, compared with LWS's data, and a momentum integral estimate.

Clearly the use of LWS' data for validation purposes required some degree of manipulation in this case; the LWS data having been non-dimensionalised with respect to $\delta$ instead of our $\delta^*$. Their displacement thickness data was re-non-dimensionalised with respect to their $\delta^*$ at our given inlet $Re_\theta = 1400$, and plotted against our data. It can be seen that all three $\delta^*$ lines grow at exactly the same rate, demonstrating agreement, especially given that $Re_\theta$ was calculated from our time-averaged $\theta$ data given in the previous figure.

Figure 5.12 shows the development of the shape function ($H$) over the length of the domain. The LWS and momentum integral estimate data was taken directly from the LWS paper. It can be seen that whilst the shape function decays at
5. Validation of the Inflow Condition

much the same rate as the validation data, it's a little bit higher, decaying from a peak value of 1.43 as opposed to 1.4. The axes of the graph deliver a fairly fine resolution, and create the illusion of poor agreement, however a shape function of approximately 1.4 for this range of Reynolds numbers is acceptable. Indeed the shape function measured by Spalart in his $Re_\theta = 1410$ DNS simulation was 1.43.

The time-averaged skin friction coefficient across the domain (Figure 5.13) was a difficult value to measure, due to the stiffness of the underlying calculations. It was essentially based on a calculation of the friction velocity at the wall;

Figure 5.12: Time-averaged shape function.
5. Validation of the Inflow Condition

\[ u_r = \sqrt{\nu (\partial U/\partial y)_{\text{wall}}} \]  
(5.13) 
\[ C_f = 2 \left( \frac{u_r}{U_{\infty}} \right)^2 \]  
(5.14)

In practical terms this involved setting \( \partial y \) as the distance from the wall to the first wall-normal grid point, and setting \( \partial U \) as the corresponding velocity value (exploiting the no-slip condition at the wall), taking into account the staggered grid. From Figures 5.7 and 5.8 it can be seen that despite the first grid point lying within the viscous sub-layer, the value of \( u \) increases rapidly in this region, amplifying any minute variations in the measured velocity. Furthermore as \( x \) increases, the boundary layer thickness increases, changing the value of \( y^+ \) for
the fixed $y$ grid point in a streamwise direction, and thus the location of the measurement within the viscous sub-layer.

Therefore it was found that a very long time-average of 90000 time-steps was required to produce a reasonable figure showing the streamwise development of $C_f$. It is worth noting at this point that the value of $u_r$ used for the rescaling process itself was derived from the weighted time-averaged rescaling profile detailed in Equation 4.7, and was as such a far more stable calculation (see Figure 5.3).

With reference to Figure 5.13 itself, it shows a period of spatial development of approximately 1/3 of the length of the domain. The author believes this is due to the location of the recycling plane, and is an unfortunate consequence of the inflow mirroring. Essentially, any large-scale structure rotating in one direction at the inlet would, for obvious reasons have to be rotating in the other direction at the recycling plane, and therefore one would expect an area of unrealistic behaviour at the start of the domain. $C_f$ appears to recover by $x = 20$, and decays with good agreement with a 7th power law estimation until $x=55$, whereupon the exit conditions cause the skin friction to drop. This is compatible with the overall inflow generation scheme, since the inflow plane used for the main simulation domain is sampled at $x = 32$, an area of good agreement with the 7th power law estimation.

An in-depth discussion of the reasons behind, and advantages of using inflow mirroring can be found in Section 6.3.6, along with a discussion of the physical implications of recycle plane placement.
5. Validation of the Inflow Condition

5.5 Grid refinement

One might argue that given the agreement already demonstrated with existing validation data, conventional convergence testing through grid refinement would be unnecessary. However given the slight discrepancies demonstrated in Figures 5.9, and 5.8, a higher resolution simulation was run, with the grid densities increased by a factor of 1.5 in each axis, yielding \( nx = 150, ny = 68 \) and \( nz = 96 \) (where \( nx, ny \) and \( nz \) are the number of grid points in the \( x, y \) and \( z \) directions respectively). These convergence simulations were computationally costly, and as such the original grid resolution was chosen for further inflow simulations.

It is interesting to note that despite the time-averaged logarithmic boundary layer profile remaining largely the same, the turbulent fluctuation agreement was
improved towards the free-stream. A quick comparison of Figure 5.15 with figure 5.9 shows that agreement for $u'$, $v'$ and $w'$ is improved from $y > 0.3$ outwards. This is consistent with the argument that any discrepancy noted in those regions was largely due to the high mesh compression at the wall, however the slight overestimation of the peak $u'$ fluctuations in the near-wall region was still present.
5. Validation of the Inflow Condition

5.6 Instantaneous contour plots

The following instantaneous plots are provided to give a snapshot of some of the quasi-coherent structures generated as part of the rescaling process.

"The idea is that they are regions of space and time (significantly larger than the smallest flow or turbulence scales) within which the flow field has a characteristic coherent pattern. Different instances of the structure occur at different positions and times, and their flow fields certainly differ in detail: but they possess a common characteristic coherent pattern."

Pope (2003) pg. 322

Figure 5.16 is an instantaneous $u$ velocity field taken in the $xz$ plane at $y^+ \approx 12$. The blue areas of the plot give a good idea of the low-speed streaks being formed in the near-wall region, and can be seen (by arbitrary choice of streak pairs) to be roughly $1.1\delta^*$ apart in a spanwise direction. Since $\delta^*$ at the inlet is equivalent to $y^+ = 90$ this yields a streak spacing of approximately 100 wall units, consistent with established experimental observations (eg. Smith & Metzler (1983)). Pope (2003) argued that streak lifting typically begins to occur around $y^+ \approx 10$ which may account for the lack of coherence of a few of the structures.

Figure 5.17 is an instantaneous $u$ velocity field, taken in the $xy$ plane, at the domain centreline. It gives a good impression of some of the large-scale motions within the boundary layer. Indeed, evidence of the bulges and valleys associated with the structure of the super-layer (eg. Falco (1977)) can be seen in the outer profile. Finally, Figure 5.18 is provided for completeness, and shows the inlet $u$ velocity in the $yz$ plane.
Figure 5.16: Instantaneous $xz$ contour plot at $II = 100000$. 
Figure 5.17: Instantaneous $xy$ contour plot at $II = 100000$. 
Figure 5.18: Instantaneous $yz$ contour plot at $II = 100000$. 
5. Validation of the Inflow Condition

5.7 Main simulation domain

Having demonstrated the accuracy of the inflow simulation, this section intends to show that effective and accurate synchronisation between the inflow and main simulations was achieved. Figure 5.19 shows the relationship between the main simulation and inflow simulation grids; for the purposes of later test-case simulations, the domain height of the inflow simulation was increased from $24\delta_{inlet}^*$ to $32\delta_{inlet}^*$, with a corresponding increase in $y$ grid points from 45 to 60, other parameters (including $\Delta y_{wall}^+ \approx 1.2$) remaining the same. The main simulation
domain was essentially double the length of the inflow simulation, with identical YZ grid-points, and identical $\Delta x^+$, $\Delta y_{wall}^+$ and $\Delta z^+$ values. Sample inflow planes were taken from the streamwise midpoint of the inflow simulation domain, approximately $Re_\theta = 1500$.

Figure 5.20 is a time-averaged plot of $\delta^*$ and $\theta$, demonstrating smooth transition between the two simulations. It’s worth noting the non-linear growth towards the downstream limits of both simulations due to the outflow boundary conditions. One can clearly see that placement of the inflow sample plane further downstream than $x \approx 32\delta_{inlt}^*$ could result in poorly conditioned inflow data, and indeed this was an aspect of the reasoning behind the author’s decision to place the inflow sample plane at $x = 32\delta_{inlt}^*$. 

Figure 5.20: $\delta^*$ and $\theta$ for inflow and main simulations.
Figure 5.21 compares \( \delta^* \), plotted against \( Re_\theta \) for both simulations, and against established displacement thickness growth data. One can see agreement with both LWS and a momentum integral estimate. We know from earlier calculations that \( x_{\text{init}} = 298.4 \delta^*_{\text{init}} \), and the composite length of the two domains (accounting for the location of the inflow sample plane at the streamwise midpoint of the inflow simulation) is \( \Delta x = (128+32)\delta^*_{\text{init}} \). Thus at the end of the main simulation domain \( x_{\text{end}} = 458.4 \delta^*_{\text{init}} \), applying Equation 5.12 yields \( \delta^*_{\text{end}} \approx 1.4 \), consistent with our results.
5. Validation of the Inflow Condition

Figure 5.22: Time-averaged logarithmic boundary layer profile - main simulation.

Figure 5.22 is a time-averaged logarithmic boundary layer profile taken from the streamwise centreline, $Re_\theta = 1730$, demonstrating reasonable agreement, with Spalart’s DNS, $Re_\theta = 1410$, noting the slightly higher peak value of $u^+$, that one would reasonably expect from a higher Reynolds number profile.

Finally, Figure 5.23 shows agreement with established turbulent fluctuation data, slightly higher than Spalart’s $Re_\theta = 1410$ data, however as before, this seems entirely reasonable given our sample profile $Re_\theta = 1730$. 
Figure 5.23: Time-averaged turbulent fluctuations - main simulation.
5. Validation of the Inflow Condition

5.8 Summary of key validation plots

- Temporal development of the inflow simulation

  - Approximately 10000 time-steps from initialisation are required for transients to settle.

  - Mass-flux at convective boundaries accurately conserved (Figure 5.2).

  - \( u_r \) at the inlet converges on a realistic mean value of approximately 0.046 (Figure 5.3).

  - Instantaneous values of \( \gamma \) for the recycling subroutine distributed about a mean, consistent with a 7th power law estimation (Figure 5.4).

  - The recycling subroutine accurately converges on an inlet \( \delta^* = 1.0 \) (Figure 5.6), within 4 or less iterations (Figure 5.5).

- Time-averaged data for the inflow simulation

  - The logarithmic boundary layer profile for the inflow simulation (at the streamwise centreline - \( x = 32\delta^{*}_{inlet}, \, Re_\theta = 1490 \)) compares well with Spalart’s DNS data (Figure 5.14).

  - Despite a slight over-estimation of the peak \( u' \) in the viscous sub-layer, turbulent fluctuations at \( x = 32\delta^{*}_{inlet}, \, Re_\theta = 1490 \) again compare well with Spalart’s data (Figure 5.15).

  - \( \delta^* \) & \( \theta \) growth, consistent with 7th power law estimation (Figure 5.10).

  - Growth of \( \delta^* \), when plotted against \( Re_\theta \), matches both LWS’ results and a momentum integral estimate (Figure 5.11).

  - The inflow simulation shape function is slightly higher than both LWS’ data and a momentum integral estimate, however it decays in a consis-
5. Validation of the Inflow Condition

tent manner, and compares well to Spalart’s shape function measured at $Re_\theta = 1410$ (Figure 5.12).

- $C_f$ initially decays unrealistically in the recycling region of the simulation, possibly due to the inflow-mirroring technique, however it recovers well further downstream and shows good agreement with a 7th power law estimation (Figure 5.13). This is discussed in greater detail in Chapter 6.

• Instantaneous contour plots - inflow simulation

  - Instantaneous low-speed streaks are spaced by approximately 100 wall units, consistent with well established experimental observations (Figure 5.16).

  - Evidence of large-scale bulges and valleys associated with the instantaneous structure of the super-layer (Figure 5.17).

• Main simulation data

  - Smooth transition between inflow and main simulation integral thicknesses, comparing well with LWS’ data, and momentum integral estimates (Figures 5.20 & 5.21).

  - Agreement for time-averaged velocity profiles and turbulent intensities, at the streamwise centreline of the main simulation domain -
    $x = 96\delta_{inlet}^*, Re_\theta = 1730$ (Figures 5.22 & 5.23).
5.9 Conclusions

This chapter has demonstrated the successful implementation of our modified inflow generation technique. Temporal development of the inflow simulation has been investigated, establishing the time-cost of starting transients, demonstrating mass conservation, and verifying realistic behaviour for a number of the key rescaling-procedure parameters. Time-averaged integral thickness data was presented for the inflow simulation, in addition to boundary layer profile data and turbulent fluctuation profiles; in all cases, agreement with well cited DNS data was established. Some minor discrepancies were noted with other figures; the shape function was higher than the values quoted by LWS and higher than the momentum integral estimate. More importantly the time-averaged skin friction showed an initial upstream deviation from the expected momentum integral estimation, which the author believes is related to the inflow-mirroring technique (discussed in Chapter 6).

Instantaneous contour plots indicated the formation of realistic turbulent structures, particularly with regards to low-speed streak spacing. Finally we established that inflow data was being successfully passed from the inflow simulation to the main simulation domain, with a smooth transition between inflow and main simulation domains for the integral thickness plots, and time-averaged profile agreement.

Chapter 6 discusses the various differences between the original LWS formulation, and our improved method in detail. This includes our substitution of $\delta^*$ for $\delta$ in the rescaling formulation, our improved initial conditions, our modifications to the rescaling of $u'$, and most importantly, the issues regarding the location of the recycling plane, and it’s effect on spurious durable spanwise variations.
CHAPTER 6

Discussion of the Inflow-Generation Technique

6.1 Introduction

In this chapter, the major differences between the original LWS formulation and our modified technique are outlined, explored and justified. We begin with a brief review of some of the more recent publications that have implemented the LWS method, highlighting their key areas of difficulty, and draw together the various approaches taken to overcome them. We then discuss the specific changes that were made in our case. This begins with our reformulation of LWS’s approach to use $\delta^*$ instead of the poorly-conditioned $\delta$ measurement, followed by our improvements to the initial conditions. We briefly discuss LWS’ original upper boundary condition, comparing it to our approach, and we explore an alternative weighting function suggested by Liu & Pletcher (2006). Our reasons for modifying the calculation of $u'$ (Equation 4.23) are explained, and finally we move on to a detailed discussion of the location of the recycling plane, and the correction of durable spanwise mean flow variations. The chapter is closed with a summary of the key differences between the original LWS formulation, and our technique.
6. Discussion of the Inflow-Generation Technique

The author is of the opinion that the LWS inflow generation method is a simple, elegant and accurate method of producing an inflow condition for a spatially developing simulation of a turbulent flat plate boundary layer. This chapter is not intended as a criticism of the method in any way! Indeed, the modifications detailed herein are merely progressive improvements to the technique, that were necessary for success in our case.

6.2 Existing observations

A number of recent spatially-developing TBL simulation papers have cited the LWS inflow generation method; a smaller number have adapted the formulation for their own use. Within this group, the two most common observations concern, A - the importance of using more accurate initial conditions than were suggested by LWS, and B - the sensitivity of the method to the position of the recycling plane.

6.2.1 Issues regarding initial conditions

A common issue regarded the original LWS' formulation's tendency to relaminarise after initialisation;

"The initial conditions for such a [LWS-based] simulation are of some importance. For instance, random perturbations that are too weak or too inadequate in length-scales can very-well 'die out'. If so, the simulation will become laminar... it could be helpful to start with the recycling station further downstream, and then move it closer to the inflow."

Spalart et al. (2005)
6. Discussion of the Inflow-Generation Technique

Spalart initialised his DNS simulation with a RANS solution, upon which he superimposed perturbations obtained from a homogeneous turbulence code, correcting their intensity with a shape function such that they matched the expected turbulent fluctuation intensities from the wall to the free-stream. Transients were observed to settle fairly quickly - within a few flow-through times.

Liu & Pletcher (2006) addressed the issue differently, with a paper focused on preventing relaminarisation by manipulating the starting transients, rather than improving the initial flow-field. They opted to continue to initialise the flow field with random fluctuations, and to implement a novel dynamic recycling plane method. Essentially, this technique ensured the recycling plane moved dynamically downstream from the start of the simulation, such that it was held in within the turbulent region produced by inflow conditions. Ferrante & Elghobashi (2004) found that their DNS LWS implementation, again initialised with random fluctuations, required the imposition of an appropriate turbulent kinetic energy spectrum at the inlet, to prevent relaminarisation.

Simens et al. (2007) had no such difficulties, having access to an accurate DNS flow-field for initialisation. They suggested that the issues encountered by Liu and Ferrante were associated with the use of an initial field with inappropriate Reynolds stresses. This suggestion is consistent with Spalart's decision to generate coherent, accurately profiled fluctuations for the initial flow field.

6.2.2 Spurious periodicity - location of the recycling plane

Various authors have hinted at problems associated with spurious periodic feedback between the recycling and inflow planes. In their inflow-condition review, Keating et al. (2004) suggested that recycling methods might introduce periodic errors into the time-series, and referred the reader to a paper by Spille-Kohoff & Kaltenbach (2001). This paper provided only a limited discussion of the mecha-
6. Discussion of the Inflow-Generation Technique

nism behind spurious periodicity;

"Several methods for generating turbulent inflow data for wall-bounded flows have been proposed which introduce a temporal periodicity (eg. LWS) on a time-scale of order $10^5/U_\infty$ that can interfere with low-frequency flow dynamics."

Spille-Kohoff & Kaltenbach (2001)

Klein et al. (2003) provided a link to a more detailed discussion by Lygren & Andersson (1999), which, despite being a plane couette flow oriented paper, discussed the extent to which periodic boundary conditions disturb large-scale structures. They suggested that the inflow-outflow coupling was associated with elongated coherent structure linking, and noted the resultant secondary formation of spanwise ‘roll-cells’, non-physical structures unsupported by the experimental work of Bech et al. (1995) and Tillmark & Alfredsson (1998).

"The counter-rotating streamwise vortices observed in numerically generated plane Couette flow are a spurious flow phenomenon. The origin of these roll cells is the localised elongated vortical structures found in the perturbed laminar couette flow, which are believed to be present also in the turbulent flow regime. The use of periodic boundary conditions in the streamwise direction provides a mechanism for self-amplification of these large-scale vortices, which eventually develop into a roll-cell pattern which extends throughout the entire computational domain"

Andersson et al. (1998)

Lygren & Andersson (1999) explored a number of different methods for disrupting this inflow-outflow coupling, including a spanwise inflow translation (or
shift), and an inflow mirroring technique (see Figure 4.6). They found that inflow mirroring continued to elongate structures at the spanwise midplane of the simulation, but prevented the formation of spurious roll cells. They had more success with the spanwise shift, finding that it suppressed any tendency for structures to elongate, in addition to preventing the spurious secondary vortices.

Their explanation for the elongation of these structures was based on the position of the recycle plane within the computational domain;

"The computational domain used in the present simulations was too short to capture the extremely large streamwise scales observed in the experiments by Bech et al. (1995) and Tillmark & Alfredsson (1998). Like ordinary periodic boundary conditions, the two new sets of boundary conditions therefore intervene in a way that would not happen if the computational domain were much larger than the largest eddies in the flow."

Andersson et al. (1998)

Essentially, they hypothesized that large streamwise structures were being sampled before natural dissipation occurred, and reintroduced at the inlet. This, they argued resulted in inlet and recycle planes linking, and the structures (non-physically) elongating, eventually causing the spurious 'roll-cell' formation.

With respect to a spatially developing boundary layer simulation, Liu & Pletcher (2006) hinted at a similar argument. They had found a paper by Guarini et al. (2000) regarding the DNS of a supersonic boundary layer, who performed a two-point correlation analysis that demonstrated that if the recycling station is far enough downstream of the inlet, the recycle plane fluctuations become fully independent of the inlet fluctuations. Furthermore, they cited a paper by Smith & Metzler (1983), who observed that streaky structures extend over a streamwise
6. Discussion of the Inflow-Generation Technique

distance of $\Delta L_x > 1000$ - implying that a sufficiently large streamwise spacing between inlet and recycle planes is necessary to avoid feedback.

Simens et al. (2007) re-iterated the point;

"There is, [in the inflow simulation], a noticeable streamwise periodicity in the flow-field, with a wavelength of the order of the distance between the [recycling] and inflow planes. This is probably the result of a weak feedback instability caused by the inflow condition, as it travels to the [recycling] plane... It could only be weakened by moving the reference plane far enough from the inflow to allow for the phase of the velocity to de-cohere. In fact the observed periodicity is much weaker in the case [of a longer recycling length], the data suggests that it would essentially be absent if length had been chosen beyond roughly $1000\theta_{inlet}$.”

Simens et al. (2007)

In conclusion, there appear to be two different methods of dealing with spurious behaviour caused by feedback between the recycling plane and the inlet. The first is to disrupt the spatial synchronisation of structures re-applied at the inlet by shifting or mirroring the inlet flow. The second (possibly costly) method is simply to ensure the inflow simulation domain is long enough to allow the full development and natural dissipation of the longest streaky structures in the flow, before sampling the recycle plane.

It is interesting to note that Spalart et al. (2005) had chosen the first technique of inflow shifting to disrupt feedback, and furthermore moved the recycle plane to a position very close to the inlet in order to facilitate the use of simplified LWS-based recycling scheme. This method took advantage of two favourable facts; firstly that near-wall turbulence regenerates itself much quicker than the outer
region, and little damage is done by simply applying the outer region rescaling throughout. Secondly that when the recycling station is taken quite close to the inflow, (which is desirable in terms of cost), the conflict between inner-region and outer-region scaling essentially vanishes.

6.2.3 Other observations

Some final, formulation-based suggestions were made by Liu & Pletcher (2006); firstly with regards to under and overproduction of turbulent fluctuations in the rescaling procedure, and secondly the suggestion of an improved weighting function, for blending the inner and outer rescaled velocity profiles. These will be dealt with in more detail later in the chapter.

6.3 Modifications to the LWS technique

The following section provides a detailed description and discussion of the specific differences between the original formulation described by LWS and our final implementation, highlighting any improvements from the individual changes. This begins with the reformulation of the recycling scheme to use $\delta^*$ instead of $\delta$, followed by our approach to improving the flow-field used to initialise the simulation. Our changes to the upper boundary condition are discussed, followed by our investigation into some of the suggestions proposed by Liu & Pletcher (2006) to improve the weighting function, and the rescaling of $u'$. Finally we move on to a discussion of, and justification for our location of the recycling plane, and the correction of spurious spanwise flow variations.
6.3.1 Use of $\delta^*$ instead of $\delta$

Perhaps the most obvious difference between LWS and our formulation was our substitution of $\delta^*$ for $\delta$ throughout the rescaling process.

"The boundary-layer thickness $\delta$ is generally defined as the value of $y$ at which $u$ equals 99% of the freestream velocity $u_\infty$. This is a poorly conditioned quantity, since it depends on the measurement of a small velocity difference. More reliable are integral measures such as the displacement thickness."

Pope (2003) pg. 299

Indeed, despite time-averaging $u$ at the recycle point, our initial use of $\delta$ proved unsuccessful. It yielded poor control of inlet mass flux, wildly fluctuating values for $\gamma$, poor control of outer velocity profile rescaling (resulting in corrupted composite inlet profiles), and unstable time-averaged velocity and turbulent fluctuation profiles. As such, the use of $\delta$ was abandoned in favour of $\delta^*$ at an early stage.

LWS had initially suggested that;

$$\eta_{\text{resc}} = \frac{y}{\delta_{\text{resc}}}$$  \hspace{1cm} (6.1)

One can immediately see that the poorly conditioned $\delta$ measurement would have an impact on the calculation of $\eta$ and thus the rescaling of the outer profile. Over time it was found that the composite profile of the outer and inner boundary layer rescaling would fail to blend smoothly, resulting in distended inlet profiles with spurious kinks in the blending region of the weighting function. Furthermore, over time, this would corrupt other critical rescaling parameters,
such as the resultant skin-friction at the rescaling point, $\gamma$ and the displacement thickness.

An added advantage of basing $\eta$ on $\delta^*$ was mentioned in Section 5.3, in that a very stable inlet mass flux was produced. Anderson (1991) described $\delta^*$ as being an 'index proportional to the missing mass flow due to the presence of the boundary layer' (pg. 715). Essentially, iterating the calculation of the inlet flow field to accurately converge on $\delta^* = 1.0$ ensured that the mass flow across the inlet boundary remained stable.

Indeed, LWS's suggestion that, "In many cases it is more advantageous to control the inlet momentum thickness than the inlet boundary layer thickness. This can be done with a little extra effort by iteratively adjusting the inlet boundary layer thickness until the target inlet momentum thickness is achieved." hinted at the benefits of using integral thickness calculations. Given the poor conditioning of the boundary layer thickness, we found it much easier to adjust the skin friction at the inlet directly, until a target displacement thickness was achieved.

6.3.2 Initial conditions

Another modification made to the LWS rescaling technique was to improve the choice of flow-field used to initialise the simulation. LWS had suggested a velocity profile initialised with a mean profile given by the Spalding (1961) law, upon which random fluctuations with a maximum amplitude of $0.1U_\infty$ were superimposed. It can be clearly seen from Figure 6.1 that in our case this led to a rapid re-laminarisation of the flow, followed by eventual turbulent transition. This problem has been noted by a number of other authors, particularly Ferrante & Elghobashi (2004) and Liu & Pletcher (2006) who found that the prescription of more realistic initial velocity fluctuations was required.
6. Discussion of the Inflow-Generation Technique

Figure 6.1: Example of relaminarisation caused by original LWS initial conditions.

"The rescaling method has a weak point: it is difficult to rapidly generate correct downstream turbulence to use for recycling if the initial inflow conditions are not accurate. Similarly it is difficult to improve the inflow conditions by recycling a profile that is far from correct. If the initial conditions are not well posed, the interior Reynolds stress may continuously decay."

Liu & Pletcher (2006)

They chose to implement a dynamically located recycling plane during the startup period, that shifted from the inlet to the final recycling position, such that the recycling plane was kept within the turbulent region produced by the inflow conditions during the early part of the simulation. Alternatively, Simens et al.
6. Discussion of the Inflow-Generation Technique

(2007) simply initialised their simulation with a pre-stored DNS flow field. Clearly our experience was far from unusual, however our simpler, (but less elegant) initial conditions, described in Section 4.3 were found to be sufficient in quickly developing a realistic turbulent boundary layer profile. Clearly Simens' DNS initialisation technique would provide the greatest accuracy, however producing this flow-field would be more expensive than the LES simulation itself.

6.3.3 Upper boundary condition

From LWS;

\[
\frac{\partial \bar{u}}{\partial y} = 0 \quad (6.2)
\]

\[
\bar{v} = U_\infty \frac{d\delta^*}{dx} \quad (6.3)
\]

\[
\frac{\partial \bar{w}}{\partial y} = 0 \quad (6.4)
\]

Essentially the calculation of \( \bar{v} \) involved computing the spanwise averaged \( \delta^* \) in a streamwise direction, then performing a linear regression of the resulting distribution to determine \( \frac{d\delta^*}{dx} \). LWS themselves noted that local values of vertical velocity must be used when applied to flows with a non-zero pressure gradient. Since our method was intended for the eventual simulation of the interaction between a jet and a boundary layer, it was felt more prudent to use an upper boundary condition amenable to localised changes in pressure gradient. Indeed, the LWS method was investigated, and found to be no more accurate than our upper boundary condition.
6.3.4 Alternative weighting function

Liu & Pletcher (2006) suggested that the weighting function (used to 'blend' the inner and outer parts of the rescaled inflow) could be improved. They argued that since the law of the wall is only valid in the inner part of the boundary layer, and the defect law in the outer, blending should only occur in the log-law region. They proposed a modified weighting function which would ensure a composite profile only in this region;

Figure 6.2: Example of the alternative weighting function suggested by Liu & Pletcher.
6. Discussion of the Inflow-Generation Technique

\[ W(y^+) = 1.0, \quad \text{when } y^+ < 50 \quad (6.5) \]

\[ W(y^+) = \frac{1}{2} \left[ 1.0 - \tanh \left( \frac{a(q - b)}{(1.0 - 2b)q + b} \right) / \tanh(a) \right], \quad \text{otherwise, } (6.6) \]

\[ W(y^+) = 0.0, \quad \text{when } y^+ > 300 \quad (6.7) \]

where \( a = 0.5, \ b = 0.4 \) and \( q = (y^+ - 50)/(250) \).

This function was designed to be applied to a reverse-weighted version of Equation 4.25, (and as such has been mirrored for easy comparison with LWS' function in Figure 6.2). It is immediately clear that despite it being a more rigorous implementation of the theory, there is not a great deal of difference between the two. Indeed no discernable improvement was noted when it was applied to a full simulation. It would be interesting to investigate if it lends early stability to a simulation initialised with poor starting conditions, however given the lack of improvement in our case, we chose to proceed with the LWS weighting function.

6.3.5 Use of \( \gamma \) in \( u' \) rescaling

Figure 6.3 shows the time-averaged turbulent fluctuation profile from an early simulation that used \( \gamma \) in the rescaling of \( u' \). Essentially, LWS had suggested that the velocity fluctuations at the inlet should be related to those at the recycling plane via;

\[ (u')_{\text{inner}}^{\text{init}} = \gamma (u')_{\text{recy}}^{+}(y_{\text{init}}, z, t) \quad (6.8) \]

\[ (u')_{\text{outer}}^{\text{init}} = \gamma (u')_{\text{recy}}(\eta_{\text{init}}, z, t) \quad (6.9) \]
Liu & Pletcher (2006) argued that, "after many recycling and rescaling operations, it is possible to underproduce or overproduce the rms fluctuations." This seems to explain the poor profile shown in our figure. Essentially the varying $\gamma$ term in the equation causes the fluctuations to be amplified or diminished according to its instantaneous value. They suggested that a well chosen, fixed value for $\gamma$ for rescaling velocity fluctuations would result in a more stable rms profile. Applying $\gamma = 1.0$ (essentially leading to Equations 4.23 and 4.24) resulted in much better control of turbulent fluctuations (Figure 5.9).
6. Discussion of the Inflow-Generation Technique

6.3.6 Placement of recycling plane, and correction of durable spanwise mean flow variations

"The recycle plane was located $8.25\delta_0$ downstream of the inlet... The domain exit was not chosen as the recycle station in order to avoid transferring errors associated with the outflow boundary condition to the inlet via the rescaling operation. The station $8.25\delta$ was determined to be as far downstream as possible without being affected by outflow boundary condition errors."

Lund et al. (1998)

LWS highlighted one of the wider, fundamental issues concerning recycling techniques - the transferral of errors associated with the outflow boundary condition to the inlet via the rescaling operation. Keating et al. (2004) called it 'spurious periodicity' in their review of inflow generation methods. Simply put, a recycle plane placed too close to the inlet can cause a, "noticeable streamwise periodicity in the flow field, with a wavelength of the order of the distance between the reference and inflow planes." (Simens et al. (2007)). Indeed their recent work suggested that the recycle plane should be placed at least $100\theta \approx 10\delta$ away from the inlet to allow for the phase of the velocity to de-cohere.

In our case this spurious periodicity manifested itself in the undamped accumulation of spanwise error - any spanwise convection of the flow-field would be recycled and re-applied at the inlet. This error would accumulate, and eventually lead to the spurious spanwise flow becoming the dominant behaviour. This was a critical problem that had a direct impact on the long-term stability and accuracy of our inflow generation method.

In the absence of the recent work by Simens et al. (2007) and Liu & Pletcher (2006), we began a series of trial and error simulations to try to correct this
durable spanwise error. The major clue that proved key to the eventual solution of this problem was found in Spalart et al. (2005). Spalart essentially moved the recycling plane to a position very close to the inlet. He solved the resulting problems with spurious periodicity by taking advantage of the periodic spanwise boundary conditions.

"The inflow condition for the velocity vector $U$ is:

$$U(0, y, z, t) = U(x_r, y_0, z + \Delta z)$$

(6.10)

where the spanwise shift, $\Delta z$, is introduced in order to keep turbulence at the inlet and recycling sections out of phase (in the simulations... the value of $\Delta z$ was set equal to half the spanwise period). This is done to disorganise any durable spanwise variations of the mean flow, which would otherwise be recycled and possibly take much time to be damped by spanwise diffusion."

Spalart et al. (2005)

This is perhaps best illustrated by Figure 4.6. A simulation that included a spanwise shift was attempted, which immediately improved the stability of the inflow generation method, allowing much longer simulations to be performed before the durable spanwise error became dominant. Fundamentally however, disorganising the durable spanwise variations by shifting the inflow plane sideways merely delayed the accumulation of spanwise error. Eventually we tried inflow mirroring; this basically involved mirroring the flow-field across the centreline of the $z$ axis, point for point, taking into account the staggered grid, and the coherence of each velocity component (in particular the $w$ velocity), such that the flow at the inlet was an exact, rescaled 'mirror' of the flow at the recycle
point. The idea being that any spurious tendency for the flow to move sideways at the recycle point would almost immediately be corrected by flow moving in the other direction at the inlet. This approach was very successful, and solved all of our problems concerning spurious spanwise convection. Once applied, the simulation was stable, and could be run for with no problems whatsoever.

This successful simulation led to further investigation of the positioning of the recycle plane, length of computational domain etc. We found the best results with the recycle plane placed much closer to the inlet than LWS initially advocated, a short distance upstream of the plane to be extracted as the inlet boundary condition for the main simulation domain. Reference to Figure 5.13 (a time-averaged plot of $C_f$ in a streamwise direction) makes things a little clearer. Clearly the Mirroring technique is not without its disadvantages - one can see the value of $C_f$ initially diverge slightly from the expected trend between the inlet and the recycling plane, possibly due to the weak non-physical effects of the inflow mirroring on the spanwise development of coherent structures in this region, however we considered this to be a worthy sacrifice given the excellent agreement shown for the rest of the domain.

In retrospect, it may have been useful to investigate lengthening the domain beyond LWS' initial suggestion, to try determine whether or not a stable simulation could be achieved without any mirroring or shifting once the recycle plane had been placed sufficiently far downstream of the inlet. Our results certainly raised questions regarding the physical flow behaviour that underpins the accumulation of spanwise error. Indeed we are inclined to agree with Simens' postulation that spurious periodicity is related to the phase coherence of the turbulent structures being modelled, and that ideal placement allows structures to decohere before flow is recycled and reapplied at the inlet. An interesting paper by Lygren & Andersson (1999) concerning inflow-outflow coupling associated with
periodic streamwise boundary conditions in the simulation of turbulent plane Couette flow, inspired a possible explanation. They suggested that the periodic conditions provided a mechanism for self-amplification of large-scale streamwise structures, eventually leading to non-physical, 'infinite' structures extending entirely across the domain. Indeed they investigated a form of inflow shifting and mirroring not unlike our own, to disrupt the spurious coupling. Perhaps this was an issue affecting our low-speed streaky structures, recycled before they had had an opportunity to burst and reform, eventually leading to non-physical structures extending from inlet to recycle plane.

Nevertheless, we had achieved our initial objective of generating a realistic turbulent inflow condition for the main simulation domain. In some respects the application of the mirroring method to decouple and correct spurious periodicity is of greater benefit than merely extending the domain. We have shown that a statistically accurate boundary layer can be generated with a short domain, a short recycling period, and a low number of grid points. Indeed we would suggest that with a little further investigation, the precursor simulation could now be re-integrated as part of the main simulation at relatively little computational cost, in the spirit of Spalart's 2005 paper.
6. Discussion of the Inflow-Generation Technique

6.4 Summary

- Recently published research has indicated that there are two key problematic areas with LWS-based inflow generation techniques;
  
  - The sensitivity of the technique to the use of accurate initial conditions.
  - Spurious feedback between recycling and inflow planes, that can lead to secondary errors.

- Modifications were made to the LWS technique to address these issues;
  
  - Improved inflow conditions were developed, which prevented re-laminarisation, and hastened the settling of initial transients.
  - An inflow mirroring technique was successfully applied which disrupted the spurious feedback of streamwise structures, and facilitated a stable and accurate inflow generation simulation.

- Further changes were made to the formulation;
  
  - $\delta^*$ was substituted for $\delta$, (a poorly conditioned value) in the formulation. This enabled a more stable recycling scheme, with better control of inlet mass flux.
  - A different upper boundary condition was applied, better suited to unusual pressure gradients in the main simulation domain.
  - An alternative weighting function was investigated.
  - $\gamma = 1.0$ in the turbulent fluctuation rescaling ($u'$), which facilitated more realistic development of turbulent fluctuations.
6.5 Conclusions

Lund, Wu and Squires broke new ground with their inflow generation technique, indeed the author feels that it is a fundamentally simple, accurate and elegant means of generating stand-alone turbulent inflow data for a boundary layer simulation, superior in many ways to alternative random fluctuation, and experimentally based methods. Our modifications, however, have improved the method, and with further work to fully explore and characterise the issues concerning recycle plane placement, spurious periodicity and durable spanwise variations, it could become a mature, 'textbook' approach for computational fluid dynamicists to call upon when generating accurate turbulent inflow data for their boundary layer simulations.

This chapter has highlighted the major differences between the original LWS technique, and our modified formulation. We began with a review of existing problems encountered by other authors, noting the particular attention that has been given to the sensitivity of the technique to initial conditions, and the problems with spurious feedback between recycle and inlet planes. This was followed by a detailed description of the specific differences between LWS and our formulation. We justified our use of $\delta^*$ instead of $\delta$ in the rescaling scheme, and demonstrated how our improvements of the initial conditions has led to faster settling of initial conditions, and suppressed a tendency of the simulation to initially relaminarise. We justified our different upper boundary conditions, we also investigated two suggestions by Liu & Pletcher (2006); the modified weighting function was found to make little difference to the accuracy of the recycling scheme, however fixing $\gamma = 1.0$ in the turbulent fluctuation calculation clearly led to more accurate turbulent statistics.

Finally, the effect of the location of the recycling plane was investigated, es-
especially in terms of spurious feedback between planes. We surmised from the literature, that one could either choose a recycling distance long enough to ensure streamwise structures naturally de-cohere, or one could implement an inflow mirroring or shifting technique to disrupt spurious feedback. We opted for the latter, and had the most success with an inflow mirroring technique.

The next chapter will present a simple flow-control test-case (steady jets in cross-flow), in order to appraise the suitability of our inflow condition for simulations of flow control devices, and perhaps suggest any further modifications to the LES code that may be necessary for more complex flows.
CHAPTER 7

Simulation of Steady Jets in Cross-Flow

7.1 Introduction

The Jet In Cross-Flow (JICF) is a commonly studied complex turbulent flow, relevant in a wide array of engineering problems. The 'kernel' case of a steady jet acting perpendicular to a turbulent flat plate boundary layer has applications ranging from understanding the flow of pollutants issuing from smokestacks, to designing V/STOL (Vertical/Short Take Off and Landing) aeroplanes. Our particular interest in JICF was driven by their potential for application in boundary layer control devices, where jets can be used to produce streamwise vortices in a similar manner to well established solid vortex generating devices.

Solid vortex generating devices (winglets and vanes) can often be seen applied along modern airliner wings (see Figure 7.1), designed to delay flow separation. They were developed in the early 1950s by H. Bruynes and H.D. Taylor at the United Aircraft Corporation, for the purpose of improving wind tunnel diffuser flow. Pearcey (1961) provides an early summary and guide to design methodology, and Gadelhak & Bushnell (1991) provide a more recent, extensive review of their use in separation control. Essentially, streamwise vortices generated upstream on the vanes, lay downstream along the wall, mixing and transport-
Figure 7.1: Vane-style micro vortex generators on the flap of a Piper Malibu Meridian, taken from NASA fact sheet FS-2000-06-52-LaRC (available on the NASA website).

ing streamwise freestream momentum across the layer, energising the near-wall boundary layer flow. With careful placement of these devices, flow separation can be delayed or prevented altogether.

These solid vortex generating devices are useful under certain flow conditions (for example in high-lift circumstances such as aircraft take-off and landing). However, during level cruise, solid vortex generating vanes can generate unwanted parasitic drag, reducing aircraft fuel efficiency. Furthermore, permanent vortex generating devices may be undesirable in other circumstances (such as when variable control is required). Vortex generating jets (VGJ) can be used to generate similar streamwise structures, however they can be switched on or off, pulsed, or varied in velocity according to requirement (the mechanical manipulation of solid vanes would pose considerably more difficulty than the simple adjustment of VGJ velocity).

A brief literature review reveals a wealth of academic literature on the subject, making the case of jets in crossflow an excellent candidate for assessing of the suitability of our spatially developing boundary layer simulation for the modelling
of more complex vortex generating devices. In particular, to highlight any further modifications to our numerical scheme necessary to enable the characterisation and parametric study of the mechanisms underlying separation delay indicated by established experimental work.

We begin with a review of jets in cross-flow, detailing the various vortical structures observed by other authors in the perpendicular and pitched & skewed jet test-cases (this includes a brief contextualisation - steady and pulsating jets for separation control). This is followed by the details of the configuration of our test-case simulations, and some preliminary time-averaged flow-field results, intended to give an impression of the structure of our jets.

We then look specifically at the perpendicular jet test-case, establishing the accuracy of our jet simulation by validating our data directly against the experimental results of Gopalan et al. (2004). This is used as a basis for comparison with the pitched and skewed test-case. The chapter is closed with a wider discussion of some of the code modifications that would be required to enable the simulation of pulsating jets, the development of these jets over negative pressure gradients, and their effect on separated regions.

7.2 Jets in cross-flow

The purpose of this review is to establish the structures and characteristics of JICF flow-fields, for later use in the appraisal of our test-case model. There are numerous published numerical and experimental investigations into high velocity ratio perpendicular jets in cross-flow, therefore this treatment is limited to the most cited, key sources. In particular the review paper by Margason (1993), the experiments of Fric & Roshko (1994), the simulations of Yuan et al. (1999) and the various ‘pitched and skewed’ jet studies by Johnston (eg. Johnston & Nishi
(1990)). The review is brought to a close with a brief, wider discussion of the use of JICF in the context of flow control, with reference to recent interest in pulsating jets.

7.2.1 Perpendicular jet in cross-flow

One of the most important parameters with regards to the configuration of jets in cross-flow is the velocity ratio ($VR$), i.e. the ratio between the centreline velocity of the steady jet, and the free-stream. Typically, investigations have ranged from $VR = 0.5$ to $VR = 10.0$, and orifice diameters are usually of the order of the

![Figure 7.2](image_url)

Figure 7.2: Illustration of the effect of velocity ratio on the trajectory of the jet - (a) $VR = 2.0$, (b) $VR = 4.0$, (c) $VR = 8.0$ (reprinted from Fric & Roshko (1994)).
boundary-layer thickness.

When the jet issues from the orifice, it is deflected by the crossflow boundary layer, following a curved path downstream. It's cross section changes, spreading laterally into an oval shape, the crossflow shearing the spanwise edges to form a kidney shaped cross-section. In high velocity ratio jets \( VR > 2.0 \) the centreline jet trajectory projects away from the wall, and through the boundary layer. The shearing folds the spanwise faces over themselves to form a counter-rotating vortex pair, which dominates the wake flow-field. The near-wall flow behind the jet is unsteady, and resembles a Von Karman vortex street.

At lower velocity ratios \( VR < 2.0 \), the wake region of the flow field is fundamentally different, the jet centreline tending to remain closer to the wall, well within the boundary layer. Fric & Roshko (1994) noted that, 'the close proximity of the jet to the wall makes it very difficult to distinguish between jet, boundary-layer, and wake fluid. In a sense, the jet is too close to produce well-defined wake structures, and it is not cleanly separated from the wall.' Indeed, the experimental work of Gopalan et al. (2004) indicated that at these lower velocity ratios, the flow instead forms a semi-cylindrical vortical layer (or 'shell') behind the jet, enclosing a domain with slow-moving reverse flow.

7.2.2 High velocity ratios - \( VR > 2.0 \)

Figure 7.3 is an illustration depicting some of the primary vortical structures associated with the perpendicular jet near-field (based largely on a diagram in Fric & Roshko (1994)). Occurring in a region where the 3D interaction between the jet and cross-flow is most intense, they typically show the following features;
7. Simulation of Steady Jets in Cross-Flow

Figure 7.3: Illustration of a jet acting perpendicular to a boundary layer.

1. Jet shear-layer vortices,

2. Horseshoe vortex,

3. Wake vortices,

4. The emerging counter-rotating vortex pair.

The jet shear-layer vortices (eg. Sykes et al. (1986)) dominate the initial portion of the jet, and are a result of the Kelvin-Helmholtz instability of the annular shear-layer separating from the edge of the jet orifice. The horseshoe vortex (eg. Krothapalli et al. (1990)) wraps around the base of the jet, distinct until just downstream of the jet orifice. Fric & Roshko (1994) compared the flow
7. Simulation of Steady Jets in Cross-Flow

to a wall-mounted cylinder test-case, where similar structures were noted. For both cases, the upstream boundary layer encounters an adverse pressure gradient ahead of the cylinder (or jet) and separates to form the horseshoe vortex.

The emerging vortex pair has been experimentally studied by numerous authors (e.g. Keffer & Baines (1963), Kamotani & Greber (1972) and Fearn & Weston (1974)), and its origins are still the subject of much debate. All authors seem to agree that the source of the vorticity is the jet shear-layer, but the means by which it re-aligns to produce the vortex pair is unclear. Many researchers model the jet as a series of vortex rings, postulating that the rings deform and stretch as they move downstream, leading to the eventual formation of the counter-rotating vortex pair (CVP). This hypothesis seems reasonable since vortex rings are known to be the dominant structures in the near field of a free jet, however recent work by Lim et al. (2001) has shown that the presence of the CVP inhibits the formation of vortex rings, and that the CVP in fact originates directly from the deformation of the cylindrical vortex sheet (or jet column).

Finally, Figure 7.3 shows the wake vortices, vertically aligned, 'tornado-like' structures that connect the jet body with the wall boundary layer. Fric & Roshko (1994) revealed that they arise from separation events of the wall boundary layer as it sweeps around the edge of the jet. Indeed, they emphasised that;

"The counter-rotating vortex pair, and the wake vortices are quite distinct, different structures, with different generic origins. The vortex pair is essentially a manifestation of the mean flow-field induced by impulse of the initial jet... The latter is a result of the entrainment of cross-flow fluid by the jet."

Fric & Roshko (1994)
Fundamentally, Fric & Roshko (1994) concluded that the boundary layer itself is the main source of vorticity for the structures in the wake of the jet, a conclusion driven by the evidence of spectral and velocity measurements, and flow visualisation.

7.2.3 Lower velocity ratios - $VR < 2.0$

Most of the previous studies of perpendicular jets in cross-flow have focused on high velocity ratio jets ($VR > 2.0$). The lower velocity ratio case has had much less attention, and whilst the few existing experimental investigations have explored the flow field in the wake of the jet in detail, there appear to be no numerical investigations. Gopalan et al. (2004) provides a detailed investigation into lower $VR < 2.0$ flow fields, noting that the flow structures are distinctly different from the high $VR$ cases detailed earlier, particularly in the wake behind the jet. Specifically, a semi-cylindrical vortical region forms behind the jet, enclosing a region with slow reverse flow, originating from the jet shear-layer. Figure 7.4 gives an impression of the formation of this vortical ‘shell’.

They argued that the formation of the semi-cylindrical vorticity layer occurs because the upstream and downstream edges of the jet experience very different conditions as the jet emerges from the orifice. Essentially the rear, downstream face of the jet is shielded from the free-stream by the jet itself, while the leading, upstream face is exposed directly to the crossflow. The sides experience the fastest streamwise flow, due to the ‘blockage’ caused by the jet - in much the same way as the high $VR$ cases.

However, in terms of vorticity, the rear face of the jet is unopposed, whilst the leading face has a strong interaction between the negative vorticity of the wall boundary layer, and the positive vorticity of the jet. At low velocity ratios, the vorticity at the leading face of the jet is essentially dissipated by the
boundary layer. The jet vortex ring is stretched along the rear face, which subsequently 'peels over' to enclose the region of slow-moving fluid, leading to the semi-cylindrical shell behind the jet. This is fundamentally different from the high velocity ratio case, where the higher jet momentum at the leading edge 'resists' dissipation, and allows the jet column to coherently bend away from the wall. The jet vorticity, although distorted, remains confined to the jet, eventually forming the counter-rotating vortex pair.

Gopalan noted that the transition from the type of structures illustrated in Figure 7.4 to the CVP and Von Karman vortex street type structures described earlier, occurs at $VR \approx 2.0$. This appears to be supported by Fric & Roshko
(1994), who observed that at $VR = 2.0$ (their lowest velocity ratio case), the close proximity of the jet to the wall made it difficult to distinguish the vortical structures noted in the higher $VR$ cases.

Meyer et al. (2007) investigated two velocity ratios, $VR = 3.3$ and $VR = 1.3$, establishing that in the higher $VR$ case, the wake vortices are the dominant dynamic flow structures, interacting strongly with the jet core. In the lower $VR$ case, they noted that shear-layer at the leading edge of the jet was the dominant structure, and that their results supported the observations of Gopalan et al. (2004). Furthermore, they argued that the vortical shell layer in actual fact contains a steady pair of rotationally opposed tornado-like vortices.

7.2.4 Pitched and skewed jets in cross-flow

Figure 7.5 illustrates the layout of a pitched and skewed jet in cross-flow. Again, existing investigations regarding these jets are experimental, and considerably rarer than those for the high $VR$ perpendicular jet kernel case. Indeed, published literature concerning this flow configuration is of limited depth, and lacks a detailed, structure-oriented description of the flow field in the jet’s wake. Compton & Johnston (1992) noted that a jet pitched by $45^\circ$ and skewed by $90^\circ$ (a common configuration derived from early work by Wallis (1960)) tends to form a dominant, ‘single’ vortex, rather than the vortex pair, or vortex shell seen in the high and low $VR$ perpendicular jet cases respectively.

Rixon & Johari (2003) qualified this observation with their experimental investigation into the decay of streamwise vorticity behind the jets, noting that they appear to create a counter-rotating vortex pair near the orifice exit, one of which being significantly stronger than the other. Zhang & Collins (1997) agreed with this observation, and added that the weaker secondary vortex dissipated within $10D$ (where $D$ is the orifice diameter); furthermore he noted the presence
of an upstream horseshoe vortex, in common with the perpendicular jet case.

One important observation by Rixon & Johari (2003) was that the position of the primary vortex core had a marked tendency to meander by up to 30% of the local boundary layer thickness, causing some degree of smearing when time-averaging the flow field. They applied conditional averaging based on a vortex-core centred technique, noting that in their case, the discrepancy between weaker peak vorticity from the standard time-averaged flow-field, and that of the conditional time-average was up to 36%.

In terms of VR, Compton & Johnston's jets were low velocity ratio ($VR \leq 1.3$),
Rixon & Johari’s jets crossed the velocity ratio threshold at $1.0 \leq VR \leq 3.0$, and Zhang’s were low ratio again - $0.5 \leq VR \leq 1.5$. Contrary to the description provided by Gopalan et al. (2004), of the formation of a vortex shell behind a low $VR$ perpendicular jet, there appears to be no apparent evidence of similar behaviour behind pitched and skewed jets. This is a topic for further investigation in our test-cases.

7.2.5 Application of jets for flow control

As mentioned in the introduction, jets in crossflow are of particular interest to engineers trying to generate streamwise vortices in order to entrain high momentum air into the boundary layer, and delay separation. Typically, pitched and skewed jets are investigated, as opposed to the perpendicular jet case.

"Compared to a trailing vortex pair from a [perpendicular] jet, a dominant vortex from a [pitched and skewed] interaction is much stronger and appears to provide much more effective momentum transfer across the wall boundary layer."

Johnston (1999)

This improvement over the perpendicular jet case had been observed in separation experiments by Johnston & Nishi (1990), who managed to show that they were able to substantially reduce large stalled region of turbulent separated flow. This was supported by the work of other authors, such as Selby et al. (1992) who demonstrated their efficacy in reducing separation over a rearward facing ramp, and Nishi et al. (1997) who successfully applied them to a conical diffuser.

Recent work has suggested that the addition of jet pulsing might provide more efficient and effective free-stream momentum mixing and transport than
the simpler steady jet case. The group led by Dr Keith McManus at Physical Sciences Inc., Andover, Massachusetts have produced a large body of work exploring pulsating jets in cross-flow (eg. McManus et al. (1994), McManus & Magill (1997), Magill & McManus (1998)). They have successfully demonstrated that with careful choice of jet-hole geometry, pulse frequency, duty cycle (the ratio of the jet switched on to jet turned off), and velocity ratio, significant improvement of separation delay can be obtained over the steady jet case.

Johari & McManus (1997) provided flow-visualisations of pulsating jets in crossflow in order to understand the mechanisms behind this improvement, and determined that the extra benefit stems from the pulsating jets forming coherent vortex-rings, with all of the corresponding increased penetration and mixing characteristics normally associated with these vortex-ring structures (eg. Glezer (1988), Gharib et al. (1998)).

7.3 Simulation configuration

Two configurations were simulated for our steady jet in crossflow test-cases;

- Perpendicular jet, $VR = 1.0$.
- Pitched & skewed jet, $VR = 1.0$, pitch $= 45^\circ$ and skew $= 90^\circ$.

The simulation dimensions were identical to the flat-plate main simulation domain (detailed in Section 5.7), and the inflow boundary layer data was identical to the data used in that case (See Figure 5.19). Specifically, dimensions were, $128\delta^* \times 32\delta^* \times 4\pi\delta^*$ in the streamwise, wall-normal and spanwise directions respectively, with grid resolutions of $200 \times 60 \times 96$ points, yielding $\Delta x^+ = 59$, $\Delta y_{wall}^+ \approx 1.2$, and $\Delta z^+ = 18$, applying hyperbolic tangent stretching in the wall-normal direction. $Re_\theta \approx 1500$ at the inlet, $Re_\theta \approx 1950$ at the domain exit.
7.3.1 Jet configuration

The jet was located on the spanwise centreline, $x_{jet} = 48\delta^* \approx 6\delta$ downstream of the domain inlet, (37.5% of the streamwise domain length, corresponding to $Re_\theta \approx 1650$ in the flat plate case), with a circular orifice of diameter $D_{jet} = 4\delta^* \approx 0.5\delta$.

This configuration compares well with the pitched and skewed experimental configuration of Compton & Johnston (1992), whose $VR = 1.0$, orifice diameter $D_{jet} \approx 0.45\delta$, and $Re_\theta \approx 1500$. The experimental work of Gopalan et al. (2004) (used to validate the perpendicular jet case) had a slightly different configuration, with $VR = 1.0$, $D_{jet} \approx 0.35\delta$, and $Re_\theta \approx 6100$. Fundamentally, this is still a fully developed turbulent boundary layer interacting with a perpendicular jet, whose diameter is of a similar order to our own - consequently we still considered it to be a useful basis for comparison.

Figure 7.6: Velocity profile across jet.
The velocity profile for the jet inflow was a top-hat profile with smooth edges (a hyperbolic tangent profile taken from Chung et al. (2002)).

\[ u_{jet} = \frac{1}{2} \left[ 1 + \tanh \left( \frac{0.5 - |x|}{2\theta_{jet}} \right) \right] \]  
(7.1)

Where \(|x|\) represents position across the jet from the jet centreline, \(\theta_{jet}\) representing the momentum thickness of the jet inflow, where the ratio of momentum thickness to the jet diameter; \(D_{jet}/\theta_{jet} = 20\). The resulting velocity profile is shown in Figure 7.6.

Due to the nature of the LES code's formulation, the existing rectilinear grid was unable to provide mesh refinement in a streamwise or spanwise direction across the jet. Thus the grid resolution across the orifice, based on the existing grid, was 6 nodes in a streamwise, \(x\) direction, and 31 in the spanwise, \(z\) direction. Results presented later demonstrate that for our purposes, this was not a significant issue, however further research would ideally include the addition of mesh refinement around the orifice.
7.4 Jet test-cases - time-averaged velocity fields

The purpose of the section is to present a number of time-averaged velocity flow-fields taken from the perpendicular, and pitched and skewed jet simulations, to give an impression of the development of the jets in a streamwise direction. For both cases, \( u \) velocity \( xy \) contour plots are given for both the centreline and an offset, followed by \( u \), \( v \) and \( w \) plots for three \( yz \) stations downstream of the jet.

7.4.1 Perpendicular jet

Figure 7.7 shows time-averaged 3D surface plots of the perpendicular jet, side and top views at \( u = 0.4 \) (an arbitrarily chosen velocity that gives a reasonable visual impression of the shape of the jet). Note that Rixon & Johari (2003) described the tendency of the wake region of the jets to meander. Since our plots are produced from a simple time-averaged flow-field, there may be some blending due to the movement of the wake. Axes in this particular plot are non-dimensionalised with respect to the orifice diameter, \( D \), for convenience - where \( x = 0 \) is the jet centreline.

Figure 7.8 shows \( u \) velocity contour lines for \( xy \) slices at (a) \( z = 9.2 \) - the jet centreline, and (b) \( z = 14.5 \approx 1D \) away from the jet centreline. The oscillations seen in the contour lines upstream of the jet are a result of the central differencing scheme, and are due to insufficient grid resolution. These oscillations could be smoothed by the application of an upwinding scheme, however this approach was avoided for the sake of accuracy.

Figures 7.9, 7.10 and 7.11 provide \( u \), \( v \) and \( w \) velocity contour plots and velocity profiles in the \( yz \) plane for (a,b) \( x = 48 \) - the jet centreline, (c,d) \( x = 55 \approx 2D \) downstream of the centreline, (e,f) \( x = 65 \approx 4D \) downstream of the centreline. Please note that \( Z = Av \). refers to a spanwise average velocity profile.
Figure 7.7: Perpendicular jet, time-averaged 3D surface plots of $u = 0.4$, (a) side view, (b) top view.
Figure 7.8: Perpendicular jet, u velocity xy contour fields for (a) $z = 9.2$ (centreline), (b) $z = 14.5$. 
Figure 7.9: Perpendicular jet centreline u velocity yz contour fields and velocity profiles, (a,b) $x = 48$ (jet centreline), (c,d) $x = 55$, (e,f) $x = 65$. 
Figure 7.10: Perpendicular jet centreline $v$ velocity $yz$ contour fields and velocity profiles, (a,b) $x = 48$ (jet centreline), (c,d) $x = 55$, (e,f) $x = 65$. 

7. Simulation of Steady Jets in Cross-Flow
Figure 7.11: Perpendicular jet centreline $u$ velocity $yz$ contour fields and velocity profiles, (a,b) $x = 48$ (jet centreline), (c,d) $x = 55$, (e,f) $x = 65$. 
7. Simulation of Steady Jets in Cross-Flow

7.4.2 Pitched and skewed jet

Figure 7.12 shows time-averaged 3D surface plots of the jet, (pitched by 45° and skewed by 90°), side and top views at \( u = 0.4 \) (an arbitrarily chosen velocity that gives a reasonable visual impression of the shape of the jet). Axes in this particular plot are non-dimensionalised with respect to the orifice diameter, \( D \), for convenience - where \( x = 0 \) is the jet centreline.

Figure 7.13 shows \( u \) velocity contour lines for \( xy \) slices at (a) \( z = 9.2 \) - the jet centreline, and (b) \( z = 14.5 \approx 1D \) away from the jet centreline, in the direction of jet skew.

Figures 7.14, 7.15 and 7.16 provide \( u, v \) and \( w \) velocity contour plots and velocity profiles in the \( yz \) plane for (a,b) \( x = 48 \) - the jet centreline, (c,d) \( x = 55 \approx 2D \) downstream of the centreline, (e,f) \( x = 65 \approx 4D \) downstream of the centreline. Please note that \( Z = Av. \) refers to a spanwise average velocity profile.
Figure 7.12: Pitched & skewed jet, time-averaged 3D surface plots of $u = 0.4$, (a) side view, (b) top view.
Figure 7.13: Pitched & skewed jet, $u$ velocity $xy$ contour fields, (a) $z = 9.2$ (centreline), (b) $z = 14.5$. 

7. Simulation of Steady Jets in Cross-Flow
Figure 7.14: Pitched & skewed jet, centreline \( u \) velocity \( yz \) contour fields and velocity profiles, (a,b) \( x = 48 \) (jet centreline), (c,d) \( x = 55 \), (e,f) \( x = 65 \).
Figure 7.15: Pitched & skewed jet, centreline $v$ velocity $yz$ contour fields and velocity profiles, (a,b) $x = 48$ (jet centreline), (c,d) $x = 55$, (e,f) $x = 65$. 
Figure 7.16: Pitched & skewed jet, centreline $w$ velocity $yz$ contour fields and velocity profiles, (a,b) $x = 48$ (jet centreline), (c,d) $x = 55$, (e,f) $x = 65$. 
7.5 Perpendicular jet validation

In this section, we draw on the experimental work of Gopalani et al. (2004), reproducing a number of key figures from their paper with our LES model results. This is done in order to establish the accuracy of our jet simulation test-case, and to provide a basis for the discussion of some of the interesting flow-features of the low velocity ratio perpendicular jet in cross-flow.

Figure 7.17: Velocity contour plots & streamlines - mean $\sqrt{(u^2 + v^2)}$ at the jet centre-line. - (a) & (c), (compared to figures reproduced from Gopalan et al. (2004) - (b) & (d)).
Figure 7.17 shows the mean velocity magnitude ($\sqrt{(u^2 + v^2)}$) and streamlines at spanwise ($z$) jet centreline - (a)&(c) are our LES results, and (b)&(d) are reproduced from Gopalan. Clearly our model compares favourably with the experimental data. In terms of the flow-physics, jet streamlines exiting from the orifice converge, indicating a spanwise spread. There is a region of weak reverse-flow behind the jet, persisting until $x/D = 3.0$. There is also some evidence of the different conditions the jet experiences on the leading and downstream faces of the jet, as it exits the orifice. Clearly a strong shear-layer persists along the rear face of the jet, but along the leading edge, the mean shear diminishes quickly, vanishing by $x/D \approx 1.0$ (slightly lower than in the experimental case).

Figure 7.18 shows the mean $\omega_z$ vorticity plots for the same location as Figure 7.17. Again, there is good agreement with the experimental data. The contour plot highlights a number of flow features, positive vorticity in the forward face of the jet, mild positive vorticity in the wake region behind the jet, negative vorticity in the rear-boundary of the jet, and negative vorticity in the boundary layer upstream of the jet. In our low velocity ratio case, the leading edge positive vorticity is much weaker than the negative vorticity at the rear of the jet, which extends far downstream. By $x/D = 1.0$, the positive vorticity from the leading edge is zero. This diminished leading edge vorticity is caused by the strong mixing with the negative vorticity in the boundary layer.
Figure 7.18: Vorticity contour plots - mean $\omega_z$ at the jet centreline - (a) & (c), (compared to figures reproduced from Gopalan et al. (2004) - (b) & (d)).
Figures 7.19 and 7.20 should be appraised together, showing both $u'^2$ and $v'^2$ at the jet centre-plane. There is fairly good agreement with the experimental data for $u'^2$, however there are certain discrepancies for the $v'^2$ case. Near the leading edge of the orifice, $v'^2$ is certainly slightly higher than $u'^2$ - consistent to some extent with the experimental case, however the relatively strong $v'^2$ fluctuations extend up the leading edge, and across to the jet wake region.

Figure 7.19: Distributions of $u'^2$ at the jet centreline - (a)&(c), (compared to figures reproduced from Gopalan et al. (2004) - (b)&(d)).
This is markedly different from the experimental case, which shows stronger $v'^2$ fluctuations at the trailing edge of the jet. Our jet is modelled with a laminar velocity profile, and the jet itself uncharacteristically lacks Turbulent Kinetic Energy as it is emitted from the orifice exit. More sophisticated, unsteady modelling of the jet may improve the TKE agreement between our case, and the experimental data.

Figure 7.20: Distributions of $v'^2$ at the jet centreline - (a)&(c), (compared to figures reproduced from Gopalan et al. (2004) - (b)&(d)).
7. Simulation of Steady Jets in Cross-Flow

Given that the experimental jet Reynolds number is $Re_{jet} = 1.9 \times 10^4$, accurate modelling would require the use of significantly more grid points around the orifice, perhaps necessitating an auxiliary simulation for the jet. In this case we would argue that our slight $\overline{v'^2}$ discrepancy is acceptable.

Figure 7.21 shows an $xz$ velocity contour plot & streamlines for the time-averaged $\sqrt{(u'^2 + w'^2)}$ at a wall normal distance of $y/D = 0.6$. Again, there is agreement with the experimental results, with a region of reversed flow extending up to $x/D \approx 2.0$. This reverse flow is consistent with the $xy$ plots shown earlier, and seems to indicate the existence of two spatially separated recirculating shear layers. This flow behaviour is markedly different from the Von Karman vortex street, 'wake vortices' described in the literature for the high velocity ratio cases. Indeed, Meyer et al. (2007) argued that this vortical shell layer in actual fact contains a steady pair of tornado-like vortices, and the reverse-flow streamlines in our plot seem to support this.

![Figure 7.21: Velocity contour plots & streamlines - mean $\sqrt{(u'^2 + w'^2)}$ at $y/D = 0.6$ - (a), (compared to a figure reproduced from Gopalan et al. (2004) - (b)).](image-url)
7. Simulation of Steady Jets in Cross-Flow

Figure 7.22: Vorticity contour plot - mean $\omega_y$ at $y/D = 0.6$ - (a), (compared to a figure reproduced from Gopalan et al. (2004) - (b)).

Figure 7.22 shows a time-averaged $\omega_y$ vorticity contour plot for an $xz$ plane placed at $y/D = 0.6$ - our results show reasonable agreement with Gopalan. When viewed in conjunction with Figure 7.18, we can begin to see the semi-cylindrical structure of the vortex ‘shell’. Figure 7.18 indicates that the downstream face of the jet column forms the upper section of this shell structure, and Figure 7.22 intersects the downstream sides, or ‘legs’ of the vortex shell.

Figure 7.23 shows the distributions of $\overline{u'^2}$ and $\overline{w'^2}$ at $y/D = 0.6$. $\overline{u'^2}$ has two distinct lobes, either side of the jet, $\overline{w'^2}$ spread broader and weaker - converging downstream but still initially located in the same region. Turbulent intensity appears to correspond with regions of high vorticity - i.e. the ‘legs’ of the vortex shell (see Figure 7.4).
Figure 7.23: Distributions of (a) \( \overline{u'^2} \) & (b) \( \overline{w'^2} \) at \( y/D = 0.6 \), (compared to figures reproduced from Gopalan et al. (2004) - (c)&(d)).
Finally, Figure 7.28 shows vorticity offset from the centreline plane, and gives an impression of the distortion of the jet column around the spanwise edge of the orifice, giving an indication of how the flow at the rear edge of the jet 'peels' around the sides to form the vortex shell. Despite the positive vorticity at the leading edge of the jet being slightly higher than expected, there appears to be good agreement with the experimental data.
7.6 Pitched and Skewed jet

The purpose of this section is to compare our validated perpendicular jet to the pitched and skewed case. As mentioned earlier, existing literature has characterised the low VR interaction of a pitched and skewed jet with a crossflow boundary layer in limited detail, so the direct comparison of our two test-cases may be instructive. It is worth noting that despite the perpendicular case demonstrating signs of the formation of a vortex 'shell' behind the jet, (containing weak reverse flow), no mention has been made of this being the case in the pitched and skewed configuration. Indeed authors usually describe the formation of a single, coherent streamwise vortex, even in the low VR cases.

Figure 7.25 is a direct comparison between velocity contour plots and streamlines for mean $\sqrt{(u^2 + v^2)}$ at the jet centreline. There seems to be reasonable qualitative agreement between the two cases in terms of the diminished velocity at the leading edge of the jets. However the plane for the downstream wake of the jet doesn't show any signs of the weak reverse-flow shown in the perpendicular case. Figure 7.14 (presented earlier), and Figures 7.27 and 7.29 (discussed later in the chapter) appear to confirm the lack of reverse-flow in the pitched and skewed case.

Figure 7.26 - vorticity contour plots showing mean $\omega_z$ at the jet centreline, again qualitatively compares well with the perpendicular case. We can see similar diminished positive vorticity at the leading edge, and stronger negative vorticity at the trailing edge of the jet, indicating vorticity dissipation at the leading edge. The pitch and skew of the jet in plots (c) and (d) means that we do not capture the entire wake region in this centreline plot. However it is worth noting that the pitched and skewed wake appears to sit closer to the wall than in the perpendicular case.
Figure 7.25: Velocity contour plots & streamlines - mean $\sqrt{u^2 + v^2}$ at the jet centre-line. (a)&(c) Perpendicular, (b)&(d) Pitched & Skewed.
Figure 7.26: Vorticity contour plots - mean $\omega_z$ at the jet centreline, (a)&(c) Perpendicular, (b)&(d) Pitched & Skewed.
Figure 7.27 shows velocity contour plots and streamlines for mean $\sqrt{\left(u^2 + w^2\right)}$ at $y/D = 0.6$, (a) Perpendicular, (b) Pitched & Skewed.

Figure 7.27 shows velocity contour plots and streamlines for mean $\sqrt{\left(u^2 + w^2\right)}$ at $y/D = 0.6$, in the $xz$ plane, for the wake region of the jet. Please note the $z$ axis offset for the pitched and skewed case. There seems to be little evidence of the region of weak reversed-flow seen behind the perpendicular jet case. When viewed in conjunction with the $xy$ plots shown in Figures 7.14 and 7.9, we can see that despite the perpendicular jet showing signs of weak reversed $u$ velocity at a number of stations in the jet’s wake, the pitched and skewed case merely indicates a reduction in centreline velocity.
The final comparison with the perpendicular jet test-case shows offset vorticity contour plots - mean $\omega_z$ at (a) & (c) $z = 0.6D$, (b) & (d) $z = 0.85D$. Strong positive vorticity can be seen above the orifice exit in (c), moving away from the wall in (d). This behaviour is consistent with the structure of a pitched and skewed jet, projecting in a spanwise direction.

Figure 7.28: Vorticity contour plots - mean $\omega_z$ at offsets of (a) & (c) $z = 0.6D$, (b) & (d) $z = 0.85D$. Where (a,b) Perpendicular, (c,d) Pitched & Skewed.
Finally, two offset velocity contour plots & streamlines for the pitched & skewed case are presented in Figure 7.29, showing mean $\sqrt{(u^2 + v^2)}$, (a)&(b) offset by $z = 0.6D$, (c)&(d) offset by $z = 0.85D$. When compared with figure 7.25, it becomes clear that there appears to be no region of weak reverse flow behind the pitched and skewed jet, as there is in the perpendicular case.

Figure 7.29: Velocity contour plots & streamlines for the pitched & skewed case - mean $\sqrt{(u^2 + v^2)}$. (a)&(b) offset by $z = 0.6D$, (c)&(d) offset by $z = 0.85D$.

When these plots are viewed in conjunction with Figures 7.9, 7.10 and 7.11 for the perpendicular case, and Figures 7.14, 7.15 and 7.16 for the pitched and...
skewed case, we can see clear evidence of the formation of a vortex shell with a region of weak reverse \( u \) velocity in the perpendicular jet case, with two spatially separated shear-layers within this shell. The pitched and skewed case however seems to indicate some slowing of the \( u \) velocity centreline, but no reverse flow. Furthermore, there appears to be evidence of just one streamwise rotating structure behind the jet.

Earlier we had mentioned that authors studying these pitched and skewed low VR jets had made no mention of the 'shell' structure seen in the perpendicular cases. Indeed our results appear to support this, we can see no region of reversed flow, and the results appear to show signs of the formation of a coherent streamwise vortex, consistent with the experimental observations of Rixon & Johari (2003) and Compton & Johnston (1992).

7.7 Discussion

The figures presented in this final chapter have demonstrated agreement between our model of a low VR perpendicular jet issuing into a cross-flow boundary layer, and established experimental data. We have velocity field agreement and vorticity agreement. The main improvement that could be made to our model would be to impose realistic turbulent fluctuations on the mean jet velocity profile at the orifice exit, in order to improve TKE agreement in Figure 7.20. However, as mentioned, Gopalan's experimental jet Reynolds number was \( Re_{jet} = 1.9 \times 10^4 \), and accurate modelling would require the use of significantly more grid points around the orifice. In this case we would argue that our slight \( \overline{u'^2} \) discrepancy is acceptable.

Meyer et al. (2007) established that in these low VR interactions, the jet shear-layer vortices at the leading edge of the jet are the dominant mechanism
for the formation of the vortex shell behind the jet. The differential dissipation of vorticity at the leading and downstream faces of the jet lead to the jet column effectively folding over itself to form the shell. This leading-edge shear-layer is highly unsteady, and driven by the cross-flow boundary layer. Given the good agreement between our numerical model, and the experimental data, we are satisfied that our boundary layer generation can provide a sufficiently accurate inflow condition for the simulation of these jets, and as such, our JICF test case has been a success.

The pitched and skewed test-case has supported the lack of vortex shell shown in Compton & Johnston (1992)'s experimental data. Furthermore it has demonstrated that our code has the potential to be used to simulate more complex flows. In terms of further work, we would firstly investigate a number of different pitches and skews to study the transition between the perpendicular vortex shell flow-field, and the results seen in our pitched and skewed case.

The next step would be to investigate higher velocity ratio flows, to try and replicate the well-documented flow structures for the \( VR > 2.0 \) case. This would require a grid-resolution increase in the boundary layer region of the simulation - clearly the higher velocity ratio jets project further into the boundary layer, and more points would be required to capture some of the more complex unsteady structures in the jet's wake. Particularly the tornado-like wake vortices, and the jet shear-layer vortices (which would presumably extend further along the jet column into the free-stream).

This again would be followed by a parametric investigation of jet pitch and skew, to explore the effect on the counter-rotating vortex pair, and to determine whether or not similar vortical structures appear for the pitched and skewed case. Some sort of comparative appraisal of peak vorticity, and wall shear stress behind the jet would follow, with a view to a later, more detailed investigation into sepa-
ration control. Finally, pulsating, or synthetic jets could easily be simulated with the simple superimposition of a time-dependent waveform on the jet centreline velocity.

Fundamentally, the code in its current configuration is restricted to flat-plate turbulent boundary layers. In order to directly investigate separation control using these jets, we would need to generate a separated boundary layer within our rectilinear computational domain. One possible option involves modifying the upper boundary condition of the simulation by applying a suction and blowing velocity distribution to create an adverse to favourable pressure gradient at the wall, which produces a closed separation bubble (Na & Moin (1998), Pauley et al. (1990)). Another method (the thesis topic of the author's colleague, Adam Preece Preece (2008)) involves modifying the solver to treat a submerged boundary as the wall. This involves calculating and applying appropriate source terms at points directly above, or below the submerged boundary, such that a no-slip condition is enforced (Kim et al. (2001), Tseng & Ferziger (2003)). This way, a bump can be placed within the domain, leading to a more consistent and physically relevant model of separation than in the suction and blowing case. This method could be adapted to our code fairly easily.
7.8 Conclusions

The purpose of this chapter was to investigate two jet in crossflow test-cases, in order to appraise the suitability of our boundary layer generating technique for providing inflow conditions for simulations of flow control devices. Jets in cross-flow were chosen due to large quantity of literature already published on the topic, and their potential suitability for delaying separation.

We began with a review of jets in crossflow, detailing the various vortical structures that one would expect to observe in their wake. We determined that for the perpendicular case, the low velocity ratio ($VR < 2.0$) flow-field is different from the higher velocity cases, demonstrating signs of weak streamwise reverse-flow immediately behind the jet, forming a vorticity 'shell' layer. We described the flow-field behind low $VR$ pitched and skewed jets, noting that there appears to be little evidence of a similar shell-like structure. The review then briefly described the potential application of vortex generating jets in separation control.

The configuration of our simulation was described, and a series of contour line plots were presented in order to give an overall impression of our time-averaged flow-fields. We then compared our perpendicular jet test-case directly to the experimental results of Gopalan et al. (2004), and found agreement - with the minor exception of Figure 7.20 - a discrepancy attributed to the lack of realistic turbulent fluctuations in the jet inflow. This perpendicular jet test-case was then compared to our pitched and skewed case, determining that our results supported the low $VR$ pitched and skewed jet literature that makes no mention of a vortex shell behind the jet, and instead describes the formation of a single streamwise vortex.

We closed the chapter with a discussion of the suitability of our numerical code for the simulation of jets in cross-flow. We concluded that given the agreement
seen in our test-case, with limited modifications, the code could be successfully used to simulate a wide variety of JICF configurations. A rigorous investigation of these jets interacting with a separated boundary layer however, would require the application of suction and blowing at the upper boundary, or an immersed boundary method.
CHAPTER 8

Conclusion

Two areas of novel research have been presented in this thesis. Firstly we have successfully described and validated a number of modifications to the Lund et al. (1998) turbulent boundary layer inflow generation method, addressing problems described by various authors regarding the stability and accuracy of the technique. Secondly we have successfully used this boundary layer to simulate simple jet in cross flow vortex generating devices, producing what we believe to be the first documented unsteady numerical simulation of the flow field behind a low velocity ratio pitched and skewed jet.

We began with Chapter 2, where we reviewed the various inflow generation techniques appropriate for unsteady turbulent boundary layer simulations. We began with a discussion of various methods of applying experimental data as an inflow condition, followed by a more in-depth discussion of the various numerical inflow generation methods, highlighting the long and costly spatial transients that are typical of the less accurate random fluctuation based schemes. The LWS rescaling and recycling technique was then introduced as a means to provide accurate and coherent inflow data, without having to sacrifice an upstream region of the main simulation to spatial transients. Problems were noted with the original LWS formulation, as described by Liu & Pletcher (2006), and Simens et al.
(2007) (among others) to be discussed in more depth at the beginning of Chapter 6.

The literature review was followed by two ‘methods’ chapters. Chapter 3 provided a brief description of the formulation and discretisation of our underlying LES code. The dynamic subgrid-scale model was discussed, in particular, its suitability for modelling the near-wall region of a boundary layer due to the automatic adjustment of the Smagorinsky constant. This was followed by a description of the fractional-step time-advancement scheme.

Chapter 4 detailed the final, fully-validated formulation of our LWS-based inflow generation technique. We began with a description of the initial conditions, moving on to a discussion of the algorithm itself. A description of our novel inflow-mirroring technique was included, used to suppress spurious durable spanwise flow variations. Finally a brief treatment of the method used to ensure spatial and temporal synchronisation between the inflow simulation and the main simulation was provided. The chapter was closed with a step-by-step summary of the inflow generation process.

The next chapters were ‘results and discussion’ oriented. Chapter 5 demonstrated the successful implementation of our modified inflow generation technique. Temporal development of the inflow simulation was investigated, establishing the time-cost of starting transients, demonstrating mass conservation, and verifying realistic behaviour for a number of the key rescaling-procedure parameters. Time-averaged integral thickness data was presented for the inflow simulation, in addition to boundary layer profile data and turbulent fluctuation profiles; in all cases, agreement with well cited DNS data was established. Instantaneous contour plots indicated the formation of realistic turbulent structures, particularly with regards to low-speed streak spacing. Finally we established that inflow data was being successfully passed from the inflow simulation to the main
simulation domain, with a smooth transition between inflow and main simulation domains for the integral thickness plots, and time-averaged profile agreement.

Chapter 6 highlighted the major differences between the original LWS technique, and our modified formulation. We began with a review of existing problems encountered by other authors, noting the particular attention that has been given to the sensitivity of the technique to initial conditions, and the problems with spurious feedback between recycle and inlet planes. This was followed by a detailed description of the specific differences between LWS and our formulation. We justified our use of $\delta^*$ instead of $\delta$ in the rescaling scheme, and demonstrated how our improvements of the initial conditions has led to faster settling of initial conditions, and suppressed a tendency of the simulation to initially relaminarise. We discussed our different upper boundary conditions, we also investigated a number of suggestions by Liu & Pletcher (2006).

The effect of the location of the recycling plane was investigated, especially in terms of spurious feedback between planes. We surmised from the literature, that one could either choose a recycling distance long enough to ensure streamwise structures naturally de-cohere, or one could implement an inflow mirroring or shifting technique to disrupt spurious feedback. We opted for the latter, and had the most success with an inflow mirroring technique.

Finally, chapter 7 investigated two jet in crossflow test-cases, in order to appraise the suitability of our boundary layer generating technique for providing inflow conditions for simulations of flow control devices. Jets in cross-flow were chosen due to their potential suitability for delaying separation, and the wealth of literature available on perpendicular jets in crossflow.

We began with a review of jets in crossflow, detailing the various vortical structures that one would expect to observe in their wake. We determined that for the perpendicular case, the low velocity ratio ($VR < 2.0$) flow-field is different
from the higher velocity cases, demonstrating signs of weak streamwise reverse-flow immediately behind the jet, forming a vorticity 'shell' layer. We described the flow-field behind low VR pitched and skewed jets, noting that there appears to be little evidence of a similar shell-like structure. The review then briefly described the potential application of vortex generating jets in separation control.

The configuration of our simulation was described, and a series of contour line plots were presented in order to give an overall impression of our time-averaged flow-fields. We then compared our perpendicular jet test-case directly to the experimental results of Gopalan et al. (2004), and found agreement. This perpendicular jet test-case was then compared to our pitched and skewed case, finding that our results supported the low VR pitched and skewed jet literature that makes no mention of a vortex shell behind the jet, and instead describes the formation of a single streamwise vortex.

This thesis has addressed two problems; firstly the generation of inflow conditions for the simulation of a spatially developing turbulent boundary layer, and secondly the simulation of low velocity ratio jets interacting with the boundary layer. Our modifications to the Lund et al. (1998) inflow generation technique have resulted in a stable and accurate method, that validates well against well established DNS data. Secondly we have found agreement in our perpendicular jet in cross flow test-case, and have produced what we believe to be the first documented unsteady numerical simulation of the flow field behind a low velocity ratio pitched and skewed jet. In terms of the original goals of our research, we have managed to demonstrate the suitability of our numerical code for the simulation of jets in cross-flow. We conclude that given the agreement seen in our test-case, with limited modifications, our code could be successfully used to simulate a wide variety of JICF configurations.
Bibliography


Part IV

Appendices
APPENDIX A

Summary of the Original LWS Formulation

The original LWS rescaling subroutine formulation, for comparison with Section 4.8.

1. Inflow simulation flow-field is initialised with a spatially developing Spalding profile, with random fluctuations (maximum amplitude, 10%$u_\infty$) superimposed.

2. Simulation begins, rescaling subroutine called at the end of each time-step.

3. Temporally, and spatially averaged boundary layer profile is sampled at the recycle plane, instantaneous turbulent fluctuations are obtained.

4. $\delta_{\text{resc}}, \theta_{\text{resc}}$ and $u_{r,\text{resc}}$ are calculated from the mean profile.

5. $\delta_{\text{inlt}} = 1.0$ fixed, $u_{r,\text{inlt}}, \theta_{\text{inlt}}$ are calculated for the inlet.

6. $\gamma = \frac{u_{r,\text{inlt}}}{u_{r,\text{resc}}}, \eta = \frac{y}{\delta}$ are produced for use in step 7.

7. $U, u'$ and $v$ velocities at the recycle plane are mapped to those at the inlet via a linear interpolation. Law of the wall for the inner boundary layer, defect law for the outer region.
A. Summary of the Original LWS Formulation

8. Composite velocity profile produced.

9. Resultant $\delta_{\text{init}}$ is measured from the composite profile, and stretched such that $\delta_{\text{init}} = 1.0$, $\theta_{\text{init}}$ is adjusted accordingly. This new $\theta_{\text{init}}$ is substituted into step 5, an improved $u_{r,\text{init}}$ is calculated and the process is iterated until resultant composite profile produces $\delta_{\text{init}} = 1.0$ at step 8.

10. Sample planes are saved to disk, for later use as the inlet condition for the main simulation.
APPENDIX B

Rescaling Subroutine

This appendix contains the Fortran 77 implementation of the rescaling formulation, called at the end of each time-step in the inflow simulation.

```fortran
SUBROUTINE RESCALE (U, V, W, P, X, Y, Z, DX, DY, DZ, UINLET, VINLET, WINLET, 
1 XC, YC, ZC, DXS, DYS, DZS, TIME, II, UAVOLD)

! THIS SUBROUTINE IS USED TO GENERATE TURBULENT INFLOW DATA FOR A SPATIALLY DEVELOPING BOUNDARY LAYER

James Jewkes, last edit 13/05/2006

! DECLARATIONS

INTEGER RKSTEP, ITER
REAL LENGTH, U(-1: NX+2, -1: NY+2, -1: NZ+2), V(-1: NX+2, -1: NY+2, -1: NZ+2), 
1 W(-1: NX+2, -1: NY+2, -1: NZ+2), P(0: NXT, 0: NYT, 0: NZT), X(NXT), Y(NYT), 
1 Z(NZT), DX(-1: NX+2), DY(-1: NY+2), DZ(-1: NZ+2), UINLET(NYT, NZT, 3,2,2), 
1 VINLET(NXT, NZT, 3,2,2), XC(0: NXT), YC(0: NYT), 
1 ZC(0: NZT), DXS(NXT), DYS(NYT), DZS(NZT), 
1 URECY(NY+1, NZ+1), VAECY(NY+1, NZ+1), 
1 WRECY(NY+1, NZ+1), UAVZ(NY+1), UAVNEW(NY+1), UAVOLD(NY+1), 
1 UFLRECY(NY+1, NZ+1), VFLRECY(NY+1, NZ+1), VFLININ(NYT, NZT), 
1 VFLINOU(NYT, NZT), TIMEINT, UTRECY, UTINLT, UPLRECY(NYT), YPPLRECY(NYT), 
1 MTHRECY, MTHINLT, GAMMA, ETAINLT(NYT), ETAVNLT(NYT), ETAECY(NYT), 
1 ETAVRECY(NYT), UAVVINLT(NYT), UAVVINOU(NYT), VPLINLT(NYT), 
1 YPPLINLT(NYT), LININT, UFLININ(NYT, NZT), UFLINOU(NYT, NZT), 
1 WEIGHT(NYT), UINAV(NYT), DTHRECY, DTHINLT, UINTEST(NYT), YCOUNT, 
1 ZSHIFT(NYT, NZ, 3)

COMMON/PARA1/LENGTH, HEIGHT, WIDTH, ROH, RE,
1 CFL, DT, NSTEP, NPRINT, RKSTEP
```

163
B. Rescaling Subroutine

SAMPLE PLANE AND TIME AVERAGE

First step is to take a slice from a yz plane at 40%x(NXT).
Places u, v, w into recycle point variables.

```
I=INT(0.4*REAL(NX))
DO 1 K=1,NZ
   DO 1 J=1,NYT
      UAECY(J, K) = U(I, J, K)
      VRECY(J, K) = V(I, J, K)
      WRECY(J, K) = W(I, J, K)
   1 CONTINUE

Now to spanwise average instantaneous u.

DO 2 J=1,NYT
   UAVZ(J) = 0.0
   DO 3 K=1,NZ
      UAVZ(J) = UAVZ(J) + URECY(J, K)
   3 CONTINUE
   UAVZ(J) = UAVZ(J) / REAL(NZ)
2 CONTINUE

Produce weighted u time-average.
TIMEINT is the time interval used to reduce transients.
Initial slow increase of TIMEINT until TIME=800 or 100BL/UINF

IF(TIME.LT.4.0E1) THEN
   TIMEINT = TIME
ELSE IF(TIME.LE.8.0E2) THEN
   TIMEINT = 40.0
ELSE IF(TIME.GT.8.0E2) THEN
   TIMEINT = 80.0 + TIME - 800.0
ENDIF
DO 4 J=1,NYT
   IF(TIME.EQ.0.0) THEN
      UAVOLD(J) = UAVZ(J)
      UAVNEW(J) = UAVOLD(J)
   ELSE
      UAVNEW(J) = (((DT/TIMEINT)*UAVZ(J)) + ((1.0-(DT/TIMEINT))*UAVOLD(J)))
      UAVOLD(J) = UAVNEW(J)
   ENDIF
4 CONTINUE

Isolate the fluctuations in u...

DO 5 J=1,NYT
   DO 5 K=1,NZ
      UFLRECY(J, K) = URECY(J, K) - UAVNEW(J)
5 CONTINUE
```
B. Rescaling Subroutine

```fortran
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
ccc
```
B. Rescaling Subroutine

Linear interpolations for:
  u mean & fluc, inner & outer,
  v inner & outer,
  w is not rescaled.

Inner u.

\[
\begin{align*}
&\text{DO 10 } J=1,\text{NYT} \\
&\quad \text{IF}(\text{YPLINLT}(J) .GE. \text{YPLRECY}(\text{NYT})) \text{ THEN} \\
&\quad \quad \text{UAVININ}(J)-\text{UAVNEW}(\text{NYT})+\text{GAMMA} \\
&\quad \quad \text{DO 30 } K=1,\text{NZ} \\
&\quad \quad \quad \text{UFLININ}(J, K)=0.0 \\
&\quad \quad \text{ELSE} \\
&\quad \quad \quad \text{DO 9 } L=1,\text{NYT} \\
&\quad \quad \quad \quad \text{IF}((\text{YVPLRECY}(L) .LE. \text{YVPLINLT}(J)) \text{ AND.} (\text{YVPLRECY}(L+1) .GT. \text{YVPLINLT}(J))) \text{ THEN} \\
&\quad \quad \quad \quad \quad \text{LININT} = ((\text{YVPLINLT}(J)-\text{YVPLRECY}(L))/((\text{YVPLRECY}(L+1)-\text{YVPLRECY}(L)))) \\
&\quad \quad \quad \quad \quad \text{UAVININ}(J)-\text{GAMMA}*(\text{UAVNEW}(L)+\text{LININT}*(\text{UAVNEW}(L+1)-\text{UAVNEW}(L))) \\
&\quad \quad \quad \quad \quad \text{DO 18 } K=1,\text{NZ} \\
&\quad \quad \quad \quad \quad \quad \text{UFLININ}(J, K)=(\text{UFLRECY}(L, K)+\text{LININT}*(\text{UFLRECY}(L+1, K)-\text{UFLRECY}(L, K))) \\
&\quad \quad \quad \quad \text{ENDIF} \\
&\quad \quad \quad \text{9 CONTINUE} \\
&\quad \text{10 CONTINUE} \\
&\text{ENDIF} \\
&\text{100 CONTINUE} \\
\end{align*}
\]

Inner v - staggered grid.

\[
\begin{align*}
&\text{DO 100 } J=1,\text{NYT} \\
&\quad \text{IF}(\text{YVPLINLT}(J) .GE. \text{YVPLRECY}(\text{NYT})) \text{ THEN} \\
&\quad \quad \text{DO 300 } K=1,\text{NZ} \\
&\quad \quad \quad \text{VFLININ}(J, K)=0.0 \\
&\quad \quad \text{ELSE} \\
&\quad \quad \quad \text{DO 90 } L=1,\text{NYT} \\
&\quad \quad \quad \quad \text{IF}((\text{YVPLRECY}(L) .LE. \text{YVPLINLT}(J)) \text{ AND.} (\text{YVPLRECY}(L+1) .GT. \text{YVPLINLT}(J))) \text{ THEN} \\
&\quad \quad \quad \quad \quad \text{LININT} = ((\text{YVPLINLT}(J)-\text{YVPLRECY}(L))/((\text{YVPLRECY}(L+1)-\text{YVPLRECY}(L)))) \\
&\quad \quad \quad \quad \quad \text{UAVININ}(J)-\text{GAMMA}*(\text{UAVNEW}(L)+\text{LININT}*(\text{UAVNEW}(L+1)-\text{UAVNEW}(L))) \\
&\quad \quad \quad \quad \quad \text{DO 180 } K=1,\text{NZ} \\
&\quad \quad \quad \quad \quad \quad \text{VFLININ}(J, K)=(\text{VRECY}(L, K)+\text{LININT}*(\text{VRECY}(L+1, K)-\text{VRECY}(L, K))) \\
&\quad \quad \quad \quad \text{ENDIF} \\
&\quad \quad \quad \text{90 CONTINUE} \\
&\quad \text{100 CONTINUE} \\
\end{align*}
\]
B. Rescaling Subroutine

DO 16 J=1, NYT
IF(ETAINLT(J).GE.ETARECY(NYT)) THEN
  UAVINOU(J)=(UAVNEW(NYT)*GAMMA)+(1.0-GAMMA)
DO 31 K=1,NZ
  UFLINOU(J,K)=0.0
CONTINUE
ELSE
DO 17 L=1, NYT
  IF((ETARECY(L).LE.ETAINLT(J)).AND.(ETARECY(L+1).GT.ETAINLT(J))) THEN
    LININT=((ETAINLT(J)-ETARECY(L))/(ETARECY(L+1)-ETARECY(L)))*
    1.0+LININT(UAVNEW(L),LININT(UAVNEW(L+1)-UAVNEW(L))+
    ((1.0-GAMMA)))
  DO 21 K=1,NZ
    UFLINOU(J,K)=(UFLRECY(L,K)+LININT(UFLRECY(L+1,K))
    -UFLRECY(L,K))
  CONTINUE
  ENDIF
CONTINUE
ENDIF
CONTINUE

DO 160 J=1, NYT
IF(ETAVINLT(J).GE.ETAVRECY(NYT)) THEN
  DO 310 K=1,NZ
    VFLINOU(J,K)=0.0
CONTINUE
ELSE
DO 170 L=1, NYT
  IF((ETAVRECY(L).LE.ETAVINLT(J)).AND.(ETAVRECY(L+1).GT.
    ETAVINLT(J))) THEN
    LININT=((ETAVINLT(J)-ETAVRECY(L))/(ETAVRECY(L+1)-ETAVRECY(L)))*
    1.0+LININT(VRECY(L,K)+LININT(VRECY(L+1,K))
    -VRECY(L,K))
  CONTINUE
  ENDIF
CONTINUE
ENDIF
CONTINUE
ENDIF

END

END
rescaling subroutine

rescaling subroutine

composite weighting function

begin

... (code continues)...

end
B. Rescaling Subroutine

Average UINLET for more rigorous test of displacement thickness

\[ K = \text{INT(REAL(NZ)/2.0)} \]
\[ \text{DO 29 J=1, NYT} \]
\[ \text{UINAV(J)=0.0} \]
\[ \text{DO 42 K=1, NZ} \]
\[ \text{UINAV(J)=UINAV(J)+UINLET(J,K,1,1,2)} \]
\[ \text{CONTINUE} \]
\[ \text{UINAV(J)=UINAV(J)/REAL(NZ)} \]
\[ \text{CONTINUE} \]

Test resultant displacement thickness and adjust UTINLT

Use UINTEST to begin with.

\[ \text{IF(TIME.LT.200) THEN} \]
\[ \text{MTHINLT=0.0} \]
\[ \text{DTHINLT=0.0} \]
\[ \text{DO 41 J=1, NYT} \]
\[ \text{DTHINLT=DTHINLT+((1.0-UINTEST(J))*(DYS(J)))} \]
\[ \text{MTHINLT=MTHINLT+((1.0-UINTEST(J))*(DYS(J)))} \]
\[ \text{CONTINUE} \]
\[ \text{UTINLT=SQR((1/RE)*((UAVNEW(1)))/(YC(1)/DTHINLT)))} \]
\[ \text{ELSE} \]
\[ \text{Use UINAV later.} \]
\[ \text{MTHINLT=0.0} \]
\[ \text{DTHINLT=0.0} \]
\[ \text{DO 411 J=1, NYT} \]
\[ \text{DTHINLT=DTHINLT+((1.0-UINAV(J))*(DYS(J)))} \]
\[ \text{MTHINLT=MTHINLT+((1.0-UINAV(J))*(DYS(J)))} \]
\[ \text{CONTINUE} \]
\[ \text{END IF} \]
\[ \text{UTINLT=UTINLT*DTHINLT} \]
\[ \text{GOTO 69} \]
\[ \text{END IF} \]

Exit subroutine once dispth at inlet=1.0

RETURN
APPENDIX C

Further JICF Results

C.1 Steady Jet VR=0.5, perpendicular to flow

Figure C.1: $VR = 0.5$, perpendicular jet, 3D surface plot of $u = 0.4$. 
C. Further JICF Results

Figure C.2: $VR = 0.5$, perpendicular jet, $u$ contour fields for $xy$ plane, (a) $z = 9.2$ (centreline), (b) $z = 13.2$. 
Figure C.3: $VR = 0.5$, perpendicular jet, $u$ contour fields and velocity profiles for $yz$ plane, (a,b) $x = 48$ (jet centreline), (c,d) $x = 55$, (e,f) $x = 65$. 
C. Further JICF Results

Figure C.4: VR = 0.5, perpendicular jet, v contour fields and velocity profiles for yz plane, (a,b) x = 48 (jet centreline), (c,d) x = 55, (e,f) x = 65.
Figure C.5: \( VR = 0.5 \), perpendicular jet, \( w \) contour fields and velocity profiles for \( yz \) plane, (a,b) \( x = 48 \) (jet centreline), (c,d) \( x = 55 \), (e,f) \( x = 65 \).
C. Further JICF Results

C.2 Steady Jet VR=0.5, pitched and skewed

Figure C.6: VR = 0.5, pitched & skewed jet, 3D surface plot of $u = 0.4$. 
Figure C.7: $VR = 0.5$, pitched & skewed jet, $u$ contour fields for $xy$ plane, (a) $z = 9.2$ (centreline), (b) $z = 13.2$. 
Figure C.8: $VR = 0.5$, pitched & skewed jet, $u$ contour fields and velocity profiles for $yz$ plane, (a,b) $x = 48$ (jet centreline), (c,d) $x = 55$, (e,f) $x = 65$. 
Figure C.9: VR = 0.5, pitched & skewed jet, v contour fields and velocity profiles for yz plane, (a,b) x = 48 (jet centreline), (c,d) x = 55, (e,f) x = 65.
C. Further JICF Results

Figure C.10: $VR = 0.5$, pitched & skewed jet, $w$ contour fields and velocity profiles for $yz$ plane, (a,b) $x = 48$ (jet centreline), (c,d) $x = 55$, (e,f) $x = 65$. 
APPENDIX D

Theoretical Model of a Laminar 2D Jet

D.1 Introduction

This section of the appendices presents some early PhD work; a 2D theoretical model of the development of a laminar jet issuing from an orifice into quiescent air. The configuration of the model can be seen in Figure D.1 (essentially a half-jet model). Centreline velocity decay results are compared to the theory of Schlichting (2003), pp. 177-180.

D.2 Theory

D.2.1 Variables

The 2D half-jet model is split into an inner 'mass-conservation' region (denoted by an 'i' subscript), and an outer mixing region (denoted by an 'o' subscript).

- $u_i$ - Velocity in the inner region
- $u_o$ - Velocity in the outer region
- $u_{mo}$ - Centreline velocity at the orifice exit
**D. Theoretical Model of a Laminar 2D Jet**

Figure D.1: Diagram of the 2D steady laminar jet configuration.

- $u_m$ - Centreline velocity
- $u_d$ - Velocity along the mass conservation interface
- $\eta_i$ - Spanwise position in the inner region
- $\eta_o$ - Spanwise position in the outer region
- $\delta_i, \gamma_d$ - Thickness of the inner region
- $\delta_i$ - Thickness of the outer region
- $\gamma_o$ - Thickness of entire jet
- $x$ - Distance along the centreline

**D.2.2 Approximate forms for velocity profiles**

Velocity profiles were approximated using quadratic functions for both the inner and outer sections of the jet.
D. Theoretical Model of a Laminar 2D Jet

\[ u = u_i = a_i + b_i \eta_i + c_i \eta_i^2; \quad \eta_i = \frac{y}{\delta_i} \]  \hspace{1cm} (D.1)
\[ u = u_o = a_o + b_o \eta_o + c_o \eta_o^2; \quad \eta_o = \frac{y - y_d}{\delta_o} \]  \hspace{1cm} (D.2)

D.2.3 Boundary conditions

\[ u_i = u_m \quad \text{at} \quad \eta_i = 0 \]  \hspace{1cm} (D.3)
\[ u_i = u_d \quad \text{at} \quad \eta_i = 1 \]  \hspace{1cm} (D.4)
\[ \frac{\partial u_i}{\partial \eta_i} = 0 \quad \text{at} \quad \eta_i = 0 \]  \hspace{1cm} (D.5)
\[ u_o = u_d \quad \text{at} \quad \eta_o = 0 \]  \hspace{1cm} (D.6)
\[ u_o = 0 \quad \text{at} \quad \eta_o = 1 \]  \hspace{1cm} (D.7)
\[ \frac{\partial u_o}{\partial \eta_o} = 0 \quad \text{at} \quad \eta_o = 1 \]  \hspace{1cm} (D.8)
\[ \frac{\partial u_i}{\partial y} = \frac{\partial u_o}{\partial y} \quad \text{at} \quad y = y_d \]  \hspace{1cm} (D.9)

Applying to Equations (D.1) and (D.2) yields;

\[ a_i = u_m \]  \hspace{1cm} (D.10)
\[ b_i = 0 \]  \hspace{1cm} (D.11)
\[ c_i = u_d - u_m \]  \hspace{1cm} (D.12)
\[ a_o = u_d \]  \hspace{1cm} (D.13)
\[ b_o = -2u_d \]  \hspace{1cm} (D.14)
\[ c_o = u_d \]  \hspace{1cm} (D.15)
Thus;

\[ u_i = u_m(1 - \eta_i^2) + u_d \eta_i^2 \]  \hspace{1cm} (D.16)

\[ u_o = u_d(1 - \eta_o)^2 \]  \hspace{1cm} (D.17)

From Equation (D.9);

\[
\frac{\partial u_i}{\partial y} = \frac{\partial u_o}{\partial y} \hspace{1cm} \text{at} \hspace{1cm} y = y_d
\]  \hspace{1cm} (D.18)

\[
\frac{\partial u_i}{\partial y} = \frac{\partial \eta_i}{\partial y} \frac{\partial u_i}{\partial \eta_i} = \frac{1}{\delta_i} \frac{\partial u_i}{\partial \eta_i}
\]  \hspace{1cm} (D.19)

\[
\frac{\partial u_o}{\partial y} = \frac{\partial \eta_o}{\partial y} \frac{\partial u_o}{\partial \eta_o} = \frac{1}{\delta_o} \frac{\partial u_o}{\partial \eta_o}
\]  \hspace{1cm} (D.20)

\[ \Rightarrow \left. \frac{1}{\delta_o} \frac{\partial u_o}{\partial \eta_o} \right|_{\eta_o = 0} = \left. \frac{1}{\delta_i} \frac{\partial u_i}{\partial \eta_i} \right|_{\eta_i = 1} \]  \hspace{1cm} (D.21)

From Equations (D.16) and (D.17);

\[ \frac{1}{\delta_o} (-2u_d) = \frac{1}{\delta_i} (-2u_m + 2u_d) \]  \hspace{1cm} (D.22)

Rearranging;

\[ u_d = \frac{\delta_o}{\delta_o + \delta_i} u_m \]  \hspace{1cm} (D.23)

D.2.4 Conservation of mass

Four parameters: \( u_m, u_d, \delta_i, \delta_o \). Flow rate between \( y = 0 \) and \( y = y_d \) conserved.

\[
\int_{0}^{y_d} u_i dy = \text{const}
\]  \hspace{1cm} (D.24)

\[ \Rightarrow \frac{d}{dx} \left( \delta_i \int_{0}^{1} u_d \eta_i \right) = 0 \]  \hspace{1cm} (D.25)
D. Theoretical Model of a Laminar 2D Jet

From (D.16):

\[ \int_0^1 u_i \, d\eta_i = u_m \int_0^1 (1 - \eta_i^2) \, d\eta_i + u_d \int_0^1 \eta_i^2 \, d\eta_i \]

\[ = \frac{2}{3} u_m + \frac{1}{3} u_d \] \hspace{1cm} (D.26)

So, from Equation (D.25):

\[ \frac{d}{dx} (2u_m \delta_i + u_d \delta_i) = 0 \] \hspace{1cm} (D.27)

From Equation (D.23):

\[ \frac{d}{dx} \left( 2u_m \delta_i + \frac{\delta_o \delta_i}{\delta_o + \delta_i} u_m \right) = 0 \] \hspace{1cm} (D.28)

Rearranging:

\[ \frac{d}{dx} \left( u_m \left( 2\delta_i + \frac{\delta_o \delta_i}{\delta_o + \delta_i} \right) \right) = 0 \] \hspace{1cm} (D.29)

D.2.5 Conservation of momentum

For the inner flow; From the Navier-Stokes equations (assume incompressible):

\[ \frac{\partial u_i^2}{\partial x} + \frac{\partial u_i v_i}{\partial y} = \mu \frac{\partial^2 u_i}{\partial y^2} \]

\[ \int_0^{v_d} \frac{\partial u_i^2}{\partial x} \, dy + \int_0^{v_d} \frac{\partial u_i v_i}{\partial y} \, dy = \mu \int_0^{v_d} \frac{\partial^2 u_i}{\partial y^2} \, dy \]

\[ \frac{d}{dx} \int_0^{v_d} u_i^2 \, dy - u_i^2 \{y_d\} \frac{dy_d}{dx} + u_i v_i \{y_d\} = \mu \int_0^{v_d} \frac{\partial^2 u_i}{\partial y^2} \, dy \]
From the streamline definition;

\[
\frac{d}{dx} \int_0^{y_d} u_i^2\,dy - u_i^2\{y_d\} \frac{dy_d}{dx} + u_i^2\{y_d\} \frac{dy_d}{dx} = \mu \int_0^{y_d} \frac{\partial^2 u_i}{\partial y^2} \,dy
\]

\[
\frac{d}{dx} \int_0^{y_d} u_i^2\,dy = \mu \int_0^{y_d} \frac{d^2 u_i}{dy^2} \,dy
\]

Now;

\[
\mu \int_0^{y_d} \frac{d^2 u_i}{dy^2} \,dy = \tau_d = \frac{\mu}{\delta_i} \frac{du_i}{\delta_i \,d\eta_i} \bigg|_{\eta_i=1}
\]

From Equation (D.16),

\[
\frac{\mu}{\delta_i} \frac{du_i}{\delta_i \,d\eta_i} \bigg|_{\eta_i=1} = \frac{2\mu}{\delta_i} (u_d - u_m)
\]

Thus inserting into Equation (D.33);

\[
\frac{d}{dx} \left( \delta_i \int_0^1 u_i^2\,d\eta_i \right) = \frac{2\mu}{\delta_i} (u_d - u_m)
\]

From Equation (D.16),

\[
u_i^2 = u_m(1 - \eta_i^2) + ud\eta_i^2)(u_m(1 - \eta_i^2) + ud\eta_i^2)
\]

\[
u_i^2 = u_m^2 + 2u_m(u_d - u_m)\eta_i^2 + (u_d - u_m)\eta_i^4
\]

\[
\int_0^1 u_i^2\,d\eta_i = u_m^2 + \frac{2}{3} u_m(u_d - u_m) + \frac{1}{5} (u_d - u_m)^2
\]

From Equation (D.23);

\[
(u_d - u_m) = \left( \frac{\delta_o}{\delta_o + \delta_i} u_m - u_m \right) = -u_m \left( \frac{\delta_i}{\delta_o + \delta_i} \right)
\]
Thus applying (D.39) and (D.40) to (D.36);

$$\frac{d}{dx} \left( \delta_i u_m^2 \left[ 1 - \frac{2}{3} \left( \frac{\delta_i}{\delta_o + \delta_i} \right) + \frac{1}{5} \left( \frac{\delta_i}{\delta_o + \delta_i} \right)^2 \right] \right) = -\frac{2\mu u_m}{\delta_o + \delta_i} \tag{D.41}$$

For the outer flow; From the Navier-Stokes equations (assume incompressible);

$$\frac{\partial u_o^2}{\partial x} + \frac{\partial u_o v_o}{\partial y} = \mu \frac{\partial^2 u_o}{\partial y^2}$$

$$\int_{y_o}^{y_v} \frac{\partial u_o^2}{\partial x} dy + \int_{y_o}^{y_v} \frac{\partial u_o v_o}{\partial y} dy = \mu \int_{y_o}^{y_v} \frac{\partial^2 u_o}{\partial y^2} dy$$

$$\frac{d}{dx} \int_{y_d}^{y_o} u_o^2 dy = -u_o v_o \left. \frac{dy_o}{dx} \right|_{y_o}^{y_d} + u_o v_o \left. \frac{dy_o}{dx} \right|_{y_o}^{y_d} + u_o v_o \left. \frac{dy_o}{dx} \right|_{y_o}^{y_d} = \mu \int_{y_o}^{y_v} \frac{\partial^2 u_o}{\partial y^2} dy$$

Applying the streamline definition and cancelling;

$$\frac{d}{dx} \int_{y_d}^{y_o} u_o^2 dy = \mu \int_{y_o}^{y_v} \frac{d^2 u_o}{dy^2} dy \tag{D.42}$$

Now;

$$\mu \int_{y_d}^{y_v} \frac{d^2 u_o}{dy^2} dy = -\tau_d = -\frac{2\mu}{\delta_i} (u_d - u_m) \tag{D.44}$$

Thus inserting into Equation (D.43);

$$\frac{d}{dx} \left( \delta_o \int_0^1 u_o^2 d\eta_o \right) = -\frac{2\mu}{\delta_i} (u_d - u_m) \tag{D.45}$$

From Equation (D.17),

$$u_o^2 = (u_d(1 - \eta_o)^2)(u_d(1 - \eta_o)^2) \tag{D.46}$$

$$= u_d^2(1 - 4\eta_o + 6\eta_o^2 - 4\eta_o^3 + \eta_o^4) \tag{D.47}$$

$$\int_0^1 u_o^2 d\eta_o = \left[ \frac{u_o^2(\eta_o - 2\eta_o^2 + 2\eta_o^3 - \eta_o^4 + \frac{1}{5} \eta_o^5)}{u_d} \right]_0^1 \tag{D.48}$$
Thus applying (D.48) and (D.23) to (D.45);

\[
\frac{d}{dx} \left( \frac{\delta_i^2 u_m^2}{5(\delta_o + \delta_i)^2} \right) = \frac{2\mu u_m}{\delta_o + \delta_i} 
\]  \hspace{1cm} (D.49)

D.2.6 Summary of the governing equations

\[
\frac{d}{dx} \left( \frac{u_m \left( 2\delta_i + \frac{\delta_o \delta_i}{\delta_o + \delta_i} \right)}{\delta_o + \delta_i} \right) = 0 
\]  \hspace{1cm} (D.50)

\[
\frac{d}{dx} \left( \delta_i u_m^2 \left[ 1 - 2 \left( \frac{\delta_i}{\delta_o + \delta_i} \right) + \frac{1}{5} \left( \frac{\delta_i}{\delta_o + \delta_i} \right)^2 \right] \right) = -\frac{2\mu u_m}{\delta_o + \delta_i} 
\]  \hspace{1cm} (D.51)

\[
\frac{d}{dx} \left( \frac{\delta_i^2 u_m^2}{5(\delta_o + \delta_i)^2} \right) = \frac{2\mu u_m}{\delta_o + \delta_i} 
\]  \hspace{1cm} (D.52)

Adding Equation (D.51) to Equation (D.52) yields;

\[
\frac{d}{dx} \left( \delta_i u_m^2 \left[ 1 - 2 \left( \frac{\delta_i}{\delta_o + \delta_i} \right) + \frac{1}{5} \left( \frac{\delta_i}{\delta_o + \delta_i} \right)^2 \right] + \frac{\delta_i^2 u_m^2}{5(\delta_o + \delta_i)^2} \right) = 0 
\]  \hspace{1cm} (D.53)

D.2.7 Non-dimensional form of the governing equations

Let

\[
U_m = \frac{u_m}{u_{mo}} 
\]  \hspace{1cm} (D.54)

\[
U_d = \frac{u_d}{u_{mo}} 
\]  \hspace{1cm} (D.55)

\[
\Delta_{i,o} = \frac{\delta_{i,o}}{h} 
\]  \hspace{1cm} (D.56)

\[
\xi = \frac{10x}{h Re} 
\]  \hspace{1cm} (D.57)

\[
Re = \frac{u_{mo} h}{\mu} 
\]  \hspace{1cm} (D.58)
Applying to Equations (D.50), (D.52) and (D.53) yields;

\[
\frac{d}{d\xi} \left( U_m \left( 2\Delta_i + \frac{\Delta_o \Delta_i}{\Delta_o + \Delta_i} \right) \right) = 0 \quad \text{(D.59)}
\]

\[
\frac{d}{d\xi} \left( \frac{\Delta_o^3 U_m^2}{(\Delta_o + \Delta_i)^2} \right) = \frac{U_m}{\Delta_o + \Delta_i} \quad \text{(D.60)}
\]

\[
\frac{d}{d\xi} \left( \Delta_i U_m^2 - \frac{2\Delta_i^2 U_m^2}{3(\Delta_o + \Delta_i)} + \frac{\Delta_o^3 U_m^2}{5(\Delta_o + \Delta_i)^2} + \frac{\Delta_i^3 U_m^2}{5(\Delta_o + \Delta_i)^2} \right) = 0 \quad \text{(D.61)}
\]

### D.2.8 Solution for small values of $\xi$

Initial values when $\xi = 0$

\[
U_m = 1 \quad \text{(D.62)}
\]

\[
\Delta_i = 1 \quad \text{(D.63)}
\]

\[
\Delta_o = 0 \quad \text{(D.64)}
\]

Applying these values to Equations (D.59, D.61) yields;

\[
U_m \left( 2\Delta_i + \frac{\Delta_o \Delta_i}{\Delta_o + \Delta_i} \right) = 2 \quad \text{(D.65)}
\]

\[
\Delta_i U_m^2 - \frac{2\Delta_i^2 U_m^2}{3(\Delta_o + \Delta_i)} + \frac{\Delta_o^3 U_m^2}{5(\Delta_o + \Delta_i)^2} + \frac{\Delta_i^3 U_m^2}{5(\Delta_o + \Delta_i)^2} = \frac{8}{15} \quad \text{(D.66)}
\]

For the second step, let;

\[
U_m = 1 + \alpha \xi^\lambda \quad \text{(D.67)}
\]

\[
\Delta_i = 1 + \beta \xi^\mu \quad \text{(D.68)}
\]

\[
\Delta_o = \gamma \xi^\nu \quad \text{(D.69)}
\]
Applying to Equation (D.65) and neglecting higher order terms;

\[
U_m \left( 2\Delta_i + \frac{\Delta_o \Delta_i}{\Delta_o + \Delta_i} \right) = 2
\]

\[
(1 + \alpha \xi^\lambda) \left( 2(1 + \beta \xi^\mu) + \frac{(\gamma \xi^\nu)(1 + \beta \xi^\mu)}{(\gamma \xi^\nu) + (1 + \beta \xi^\mu)} \right) = 2
\]

\[
(1 + \alpha \xi^\lambda) \left( \frac{2(1 + \beta \xi^\mu)(1 + \gamma \xi^\nu + \beta \xi^\mu) + (\gamma \xi^\nu)(1 + \beta \xi^\mu)}{(1 + \gamma \xi^\nu + \beta \xi^\mu)} \right) = 2
\]

\[
2 + 3\gamma \xi^\nu + 4\beta \xi^\mu + 2\alpha \xi^\lambda = 2 + 2\gamma \xi^\nu + 2\beta \xi^\mu
\]

\[
\gamma \xi^\nu + 2\beta \xi^\mu + 2\alpha \xi^\lambda = 0
\]

Applying to Equation (D.66) and neglecting higher order terms;

\[
\Delta_i U_m^2 - \frac{2\Delta_o^2 U_m^2}{3(\Delta_o + \Delta_i)} + \frac{\Delta_i^3 U_m^2}{5(\Delta_o + \Delta_i)^2} + \frac{\Delta_o^3 U_m^2}{5(\Delta_o + \Delta_i)^2} = \frac{8}{15}
\]

\[
15\Delta_i U_m^2(\Delta_o + \Delta_i)^2 - 10(\Delta_o + \Delta_i)\Delta_i^3 U_m^2 + 3\Delta_i^3 U_m^2 + 3\Delta_o^3 U_m^2
\]

\[
= 8(\Delta_o + \Delta_i)^2
\]

\[
15(1 + \beta \xi^\mu)(1 + \alpha \xi^\lambda)^2(1 + \gamma \xi^\nu + \beta \xi^\mu)^2
\]

\[
-10((1 + \gamma \xi^\nu + \beta \xi^\mu)(1 + \beta \xi^\mu)^2(1 + \alpha \xi^\lambda)^2
\]

\[
+3(1 + \beta \xi^\mu)^3(1 + \alpha \xi^\lambda)^2 + 3(\gamma \xi^\nu)^3(1 + \alpha \xi^\lambda)^2
\]

\[
= 8(1 + \gamma \xi^\nu + \beta \xi^\mu)^2
\]
\[ 15(1 + 3\beta \xi^\mu + 2\alpha \xi^\lambda + 2\gamma \xi^\nu) \]
\[ -10(1 + \gamma \xi^\nu + 3\beta \xi^\mu + 2\alpha \xi^\lambda) \]
\[ + 3(1 + 3\beta \xi^\mu + 2\alpha \xi^\lambda) \]
\[ = 28(1 + 2\gamma \xi^\nu + 2\beta \xi^\mu) \] (D.76)

\[ 16\alpha \xi^\lambda + 8\beta \xi^\mu + 4\gamma \xi^\nu = 0 \] (D.77)

Now, from Equation (D.60), applying initial values to RHS, and second step values to LHS, neglecting higher order terms;

\[ \frac{d}{d\xi} \left( \frac{\Delta_o^2 U_m^2}{(\Delta_o + \Delta_i)^2} \right) = \frac{U_m}{\Delta_o + \Delta_i} \]
\[ \frac{d}{d\xi} \left( \frac{(\gamma \xi^\nu)^3(1 + \alpha \xi^\lambda)^2}{((\gamma \xi^\nu) + (1 + \beta \xi^\mu))^2} \right) = 1 \] (D.78)
\[ \frac{d}{d\xi} (\gamma^3 \xi^{3\nu}) = 1 \] (D.79)

Differentiating LHS;

\[ 3\nu \gamma^3 \xi^{3\nu-1} = 1 \] (D.80)

Thus;

\[ \nu = \frac{1}{3} \] (D.81)
\[ \gamma^3 = 1 \] (D.82)

Applying to Equation (D.69);

\[ \Delta_o = \gamma \xi^\nu \]
\[ \Delta_o = \xi^{\frac{1}{3}} \] (D.83)
Applying Equation (D.81) to Equations (D.73, D.77);

\[ 2\alpha \xi^\lambda + 2\beta \xi^\mu = -\gamma \xi^{\frac{1}{3}} \]  
(D.84)

\[ 4\alpha \xi^\lambda + 2\beta \xi^\mu = -\gamma \xi^{\frac{1}{3}} \]  
(D.85)

Thus;

\[ \alpha \xi^\lambda = 0 \]  
(D.86)

\[ \mu = \frac{1}{3} \]  
(D.87)

\[ \beta = -\frac{1}{2} \gamma = -\frac{1}{2} \]  
(D.88)

In Summary;

\[ U_m = 1 \]  
(D.89)

\[ \Delta_i = 1 - \frac{1}{2} \xi^{\frac{1}{3}} \]  
(D.90)

\[ \Delta_o = \xi^{\frac{1}{3}} \]  
(D.91)

D.3 Results & discussion

The governing equations were solved using a simple Runge-Kutta method, initialised with the above solution for small values of \( \xi \).
Results were compared to Schlichting (2003), his self-similar model (detailed in his book, pp. 179, Equation D.92) has a virtual origin behind the orifice exit. The experimental work of Andrade (1939) confirms Schlichting’s theoretical model to be accurate in the far field.

"Here it has been assumed that the jet has approximately the profile of a fully developed channel flow at the outlet in the wall, that is, does not yet possess the profile it demonstrates further downstream. However, in the far-field, similar profiles are to be expected, since the effect of the start of the jet dies away. Therefore the similar solution shown is a fictitious flow, although one which does describe a real flow in the far-field."

Schlichting (2003)
D. Theoretical Model of a Laminar 2D Jet

Figure D.3: Comparison between our theoretical model, and Schlichting's self-similar solution.

\[ u = 0.4543 \left( \frac{K^2}{\nu x} \right)^{\frac{1}{3}} (1 - \tanh^2 \eta) \]  \hspace{1cm} (D.92)

where, \( \eta = 0 \), and the remaining terms are constants for the centreline, such that \( u \propto x^{-\frac{1}{3}} \).

From Figure D.2, we can see that our model produces more plausible behaviour than Schlichting's in the near-field, whilst maintaining excellent agreement in the far field (Figure D.3).