Turbulent inflow generation methods for Large Eddy Simulations

By

John S. Haywood

A Dissertation
Submitted to the Faculty of
Mississippi State University
in Partial Fulfillment of the Requirements
for the Degree of Doctor of Philosophy
in Aerospace Engineering
in the Department of Aerospace Engineering

Mississippi State, Mississippi

August 2019

Distribution Statement A. Approved for public release; distribution is unlimited.
Turbulent inflow generation methods for Large Eddy Simulations

By

John S. Haywood

Approved:

__________________________
Adrian Sescu
(Major Professor)

__________________________
J. Mark Janus
(Committee Member)

__________________________
Shanti Bhushan
(Committee Member)

__________________________
David S. Thompson
(Committee Member/Graduate Coordinator)

__________________________
Jason M. Keith
Dean
Bagley College of Engineering

Distribution Statement A. Approved for public release; distribution is unlimited.
With the increased application of large eddy simulations and hybrid Reynolds-averaged Navier-Stokes techniques, the generation of realistic turbulence at inflow boundaries is crucial for the accuracy of numerical results. In this dissertation research, two novel turbulence inflow generation methods are derived and validated.

The first method, the Triple Hill’s Vortex Synthetic Eddy Method, is a new type of synthetic eddy method, where the fundamental eddy is constructed through a superposition of three orthogonal Hill’s vortices. The amplitudes of the three vortices that form the fundamental eddy are calculated from known Reynolds stress profiles through a transformation from the physical reference frame to the principal-axis reference frame. In this way, divergence-free anisotropic turbulent velocity fields are obtained that can reproduce a given Reynolds stress tensor. The model was tested on isotropic turbulence decay, turbulent channel flow, and a spatially developing turbulent mixing layer. The Triple Hill’s
Vortex Synthetic Eddy Method exhibited a quicker recovery of the desired turbulent flow conditions when compared with other current synthetic turbulence methods.

The second method is the Control Forced Concurrent Precursor Method which combines an existing concurrent precursor method and a mean flow forcing method with a new extension of the controlled forcing method. Turbulent inflow boundary conditions are imposed through a region of body forces added as source terms to the momentum equations of the main simulation which transfer flow variables from the precursor simulation. Controlled forcing planes imposed in the precursor simulation, allow for specific Reynolds stress tensors and mean velocities to be imposed. A unique feature of the approach is that the proposed fluctuating flow controlled forcing method can be applied to multiple fluctuating velocity components and couple their calculation to amplify the existing fluctuations present in the precursor flow field so that prescribed anisotropic Reynolds stress tensors can be reproduced. The new method was tested on high and low Reynolds number turbulent boundary layer flows, where the proposed fluctuating flow controlled forcing method greatly accelerated the development of the turbulent boundary layers when compared with cases without controlled forcing and with only the original controlled forcing.

Distribution Statement A. Approved for public release; distribution is unlimited.
ACKNOWLEDGEMENTS

I wish to acknowledge my parents, Carl and Barbara, and my sister, Nicole, for all of their unwavering support. Thank you Courtney for all the laughter and encouragement. I am grateful for the many on topic and absurd conversations with my research group mates. Finally, I thank my committee for their comments on this dissertation, and I thank Adrian Sescu for calmly directing this research.

Effort sponsored by the Engineering Research & Development Center under Cooperative Agreement number W912HZ-17-C-0021. The views and conclusions contained herein are those of the author and should not be interpreted as necessarily representing the official policies or endorsements, either expressed or implied, of the Engineering Research & Development Center.

Reference herein to any specific commercial company, product, process, or service by trade name, trademark, manufacturer, or otherwise, does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States Government or the Department of the Army (DoA). The opinions of the author expressed herein do not necessarily state or reflect those of the United States Government or the DoA, and shall not be used for advertising or product endorsement purposes.

Distribution Statement A. Approved for public release; distribution is unlimited.
TABLE OF CONTENTS

ACKNOWLEDGEMENTS ................................................................. ii

LIST OF TABLES ................................................................. vi

LIST OF FIGURES ................................................................. vii

CHAPTER

1. INTRODUCTION ................................................................. 1

   1.1 Contributions ........................................................... 4

2. OVERVIEW OF TURBULENT INFLOW GENERATION METHODS ................. 6

   2.1 Precursor Methods ....................................................... 6
   2.1.1 Concurrent Methods .................................................. 7
   2.1.2 Database Methods ..................................................... 13

   2.2 Synthetic Turbulence Method ........................................... 20
   2.2.1 Random Fluctuation Method ....................................... 21
   2.2.2 Digital Filtering Methods ......................................... 23
   2.2.3 Diffusion Method .................................................... 26
   2.2.4 Fourier Methods ...................................................... 27
   2.2.5 Proper Orthogonal Decomposition Methods ....................... 32
   2.2.6 Wavelet Methods ..................................................... 34
   2.2.7 Vortex Method ........................................................ 35
   2.2.8 Random Turbulent Spot Method ................................... 37
   2.2.9 Synthetic Eddy Method .............................................. 38
   2.2.10 Source Methods ..................................................... 44

   2.3 Controlled Forcing Methods ............................................ 45

3. TRIPLE HILL’S VORTEX SYNTHETIC EDDY METHOD ........................... 50

   3.1 Governing Equations .................................................... 51
   3.1.1 Subgrid Scale Model ............................................... 53
   3.2 Numerical Algorithm .................................................... 55

Distribution Statement A. Approved for public release; distribution is unlimited.
## 3.2.1 Spatial Discretization

- 3.2.2 Time Marching
- 3.2.3 Spatial Filtering
- 3.2.4 Boundary Conditions

## 3.3 Proposed Synthetic Eddy Method

- 3.3.1 Hill's Spherical Vortex
- 3.3.2 Triple Hill's Vortex
- 3.3.3 Divergence of a Triple Hill's Vortex
- 3.3.4 Convection of a Triple Hill's Vortex
- 3.3.5 Inflow Velocity
- 3.3.6 Determination of the Amplitudes
- 3.3.7 Modification of the Target Reynolds Stresses
- 3.3.8 Generations of Triple Hill's Vortices

## 3.3.7.1 Reynolds Stresses for a Single THV

## 3.3.9 Near-wall THV Stretching

## 3.4 Results

- 3.4.1 Single Triple Hill's Vortex
- 3.4.2 Homogeneous Turbulent Flow
- 3.4.3 Turbulent Channel Flow
- 3.4.4 Turbulent Mixing Layer

## 3.5 Distorted Triple Hill's Vortex

- 3.5.1 Distortion of a Hill's Vortex
- 3.5.2 Results
- 3.5.2.1 Single Triple Hill's Vortex
- 3.5.2.2 Homogeneous Turbulent Flow
- 3.5.2.3 Turbulent Channel Flow
- 3.5.2.4 Turbulent Mixing Layer

## 3.6 Conclusions and Future Work

## 4. CONTROL FORCED CONCURRENT PRECURSOR METHOD

### 4.1 Governing Equations

- 4.1.1 Subgrid Scale Model
- 4.1.2 Wall Model
- 4.1.3 Immersed Boundary Method

### 4.2 Numerical Algorithm

- 4.2.1 Spatial Discretization
- 4.2.2 Time Marching
- 4.2.3 Boundary Conditions

### 4.3 Proposed Precursor Method

- 4.3.1 Concurrent Precursor Method
- 4.3.2 Fluctuating Flow Controlled Forcing
- 4.3.3 Mean Flow Controlled Forcing

### 4.4 Results

- 4.4.1 Single Triple Hill's Vortex
- 4.4.2 Homogeneous Turbulent Flow
- 4.4.3 Turbulent Channel Flow
- 4.4.4 Turbulent Mixing Layer

### 3.3.1 Hill's Spherical Vortex

### 3.3.2 Triple Hill's Vortex

### 3.3.3 Divergence of a Triple Hill's Vortex

### 3.3.4 Convection of a Triple Hill's Vortex

### 3.3.5 Inflow Velocity

### 3.3.6 Determination of the Amplitudes

### 3.3.7 Modification of the Target Reynolds Stresses

### 3.3.8 Generations of Triple Hill's Vortices

### 3.4.1 Single Triple Hill's Vortex

### 3.4.2 Homogeneous Turbulent Flow

### 3.4.3 Turbulent Channel Flow

### 3.4.4 Turbulent Mixing Layer

### 3.5.1 Distortion of a Hill's Vortex

### 3.5.2 Results

### 3.5.2.1 Single Triple Hill's Vortex

### 3.5.2.2 Homogeneous Turbulent Flow

### 3.6 Conclusions and Future Work

### 4. CONTROL FORCED CONCURRENT PRECURSOR METHOD

### 4.1 Governing Equations

### 4.1.1 Subgrid Scale Model

### 4.1.2 Wall Model

### 4.1.3 Immersed Boundary Method

### 4.2 Numerical Algorithm

### 4.2.1 Spatial Discretization

### 4.2.2 Time Marching

### 4.2.3 Boundary Conditions

### 4.3 Proposed Precursor Method

### 4.3.1 Concurrent Precursor Method

### 4.3.2 Fluctuating Flow Controlled Forcing

### 4.3.3 Mean Flow Controlled Forcing

### 4.4 Results

### 4.4.1 High Reynolds Number Turbulent Boundary Layer

### 4.4.2 Homogeneous Turbulent Flow

### 4.4.3 Turbulent Channel Flow

### 4.4.4 Turbulent Mixing Layer

### 4.5 Conclusions and Future Work
### LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1</td>
<td>Weights of the centered stencils</td>
<td>56</td>
</tr>
<tr>
<td>3.2</td>
<td>Example of five generations of THV’s</td>
<td>75</td>
</tr>
<tr>
<td>3.3</td>
<td>THV generations for the homogeneous isotropic case</td>
<td>81</td>
</tr>
<tr>
<td>3.4</td>
<td>THV generations for the channel case, where $-\delta &lt; y &lt; \delta.$</td>
<td>88</td>
</tr>
<tr>
<td>3.5</td>
<td>THV generations for the spatial turbulent mixing layer</td>
<td>100</td>
</tr>
<tr>
<td>4.1</td>
<td>Grid point dimensions of the high Reynolds number turbulent boundary layer domain.</td>
<td>131</td>
</tr>
<tr>
<td>4.2</td>
<td>Grid point dimensions of the low Reynolds number turbulent boundary layer domain.</td>
<td>138</td>
</tr>
<tr>
<td>FIGURE</td>
<td>DESCRIPTION</td>
<td></td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
<td></td>
</tr>
<tr>
<td>2.1</td>
<td>Diagrams showing the recycling method with a separate precursor domain (top) and a precursor region connected to the main domain (bottom).</td>
<td></td>
</tr>
<tr>
<td>2.2</td>
<td>Diagram of the temporal database method.</td>
<td></td>
</tr>
<tr>
<td>2.3</td>
<td>Diagram of the spatial database method.</td>
<td></td>
</tr>
<tr>
<td>2.4</td>
<td>Diagram of the spatiotemporal database method. $\Delta T$ corresponds to the time scale of the largest-scale turbulent motion.</td>
<td></td>
</tr>
<tr>
<td>2.5</td>
<td>Diagram of the spatiotemporal database method with blending.</td>
<td></td>
</tr>
<tr>
<td>2.6</td>
<td>Diagram showing the controlled forcing method.</td>
<td></td>
</tr>
<tr>
<td>3.1</td>
<td>An example of inlet plane: black) existing THV’s; red) new THV.</td>
<td></td>
</tr>
<tr>
<td>3.2</td>
<td>A diagram of the stretched THV imposed in the near-wall region: a) inclination angle; b) tilting angle.</td>
<td></td>
</tr>
<tr>
<td>3.3</td>
<td>Contours of the velocity magnitude on xy (left), xz (middle), and yz (right) planes through the center of a THV.</td>
<td></td>
</tr>
<tr>
<td>3.4</td>
<td>A single THV being generated at the inlet. Time is increasing with a constant time step from a) to d) and the inlet is on the left side of each frame.</td>
<td></td>
</tr>
<tr>
<td>3.5</td>
<td>Top) Isosurfaces of Q-criterion. Contours of vorticity magnitude: middle) xy-plane through the center of the domain; bottom) xz-plane through the center of the domain.</td>
<td></td>
</tr>
<tr>
<td>3.6</td>
<td>Two-point spatial correlations (a) and turbulent kinetic energy spectrum as a function of wavenumber (b), captured at the inlet, compared with the experimental data collected by Comte-Bellot and Corrsin [28]. a) ——) transverse; - - -) longitudinal; O) experimental transverse; X) experimental longitudinal. b) O) THV SEM; ——) experimental.</td>
<td></td>
</tr>
</tbody>
</table>

Distribution Statement A. Approved for public release; distribution is unlimited.
3.7 Turbulent kinetic energy (a) and velocity derivative skewness (b) in the streamwise direction compared with the experimental data collected by Comte-Bellot and Corrsin [27] and LES data from Jarrin et al. [47]: —— THV SEM; O) \(k^{-1.25}\) experimental fit; - - -) original SEM. 85

3.8 Turbulent kinetic energy in the streamwise direction (a) and longitudinal two-point spatial correlation at the inlet compared with the experimental data collected by Comte-Bellot and Corrsin [27] and LES data from Dietzel et al. [34]: black) THV SEM; O) experimental; blue) digital filtering method [66, 59]; red) diffusion method [58]; yellow) Fourier method [16, 30]. 86

3.9 Two-point spatial correlations (a) and turbulent kinetic energy spectrum as a function of wavenumber (b), captured downstream of the inlet, compared with the experimental data collected by Comte-Bellot and Corrsin [28]. a) ——) transverse; - - -) longitudinal; O) experimental transverse; X) experimental longitudinal. b) O) THV SEM; ——) experimental. 87

3.10 Contours for turbulent channel flow at the inlet plane (from the synthetic turbulence model): a) streamwise velocity; b) vorticity magnitude. 89

3.11 Isosurfaces of Q-criterion for turbulent channel flow. 90

3.12 Vertical profiles at the inlet compared with the DNS data collected by Moser et al. [105] of: a) mean streamwise velocity; b) Reynolds stresses; ——) THV SEM; - - -) DNS data from Moser et al. [105]. 91

3.13 Two-point spanwise spatial correlations, captured at the inlet and at three different heights from the wall, compared with the DNS data collected by Moser et al. [105]. blue) transverse; red) longitudinal; - - -) DNS transverse; -.-.-) DNS longitudinal. 92

3.14 Turbulent kinetic energy spectra, captured at the inlet and at three different heights from the wall, compared with the DNS data collected by Moser et al. [105]. blue) transverse; red) longitudinal; - - -) DNS transverse; -.-.-) DNS longitudinal. 93

3.15 Downstream development of the friction coefficient compared with the DNS data collected by Moser et al. [105] and three different synthetic eddy methods: black) THV SEM; blue) Original SEM (OSEM) of Jarrin et al. [47]; red) Divergence-Free SEM (DFSEM) of Poletto et al. [125]; yellow) SEM (SSEM) of Skillen et al. [148]; - - -) DNS of Moser et al. [105]. 94

Distribution Statement A. Approved for public release; distribution is unlimited.
3.16 Vertical profiles at three different streamwise locations compared with the DNS data collected by Moser et al. [105] of: a) turbulent kinetic energy; b) Reynolds shear stress; blue) \( x = 0 \) (inlet); red) \( x = 2\delta \); yellow) \( x = 4\delta \); - - -) DNS data from Moser et al. [105]. 95

3.17 Two-point streamwise spatial correlations, captured at two different heights from the wall, compared with the DNS data collected by Moser et al. [105]. blue) transverse; red) longitudinal; - - -) DNS transverse; -.-.-) DNS longitudinal. 96

3.18 Downstream evolution of the two-point spanwise spatial correlations, captured at two different heights from the wall, compared with the DNS data collected by Moser et al. [105]. blue) \( x = 0 \) (inlet); red) \( x = 2\delta \); yellow) \( x = 4\delta \); - - -) DNS transverse. 97

3.19 Isosurfaces of Q-criterion for the turbulent mixing layer. 99

3.20 Vertical profile of the mean streamwise velocity at the inlet. blue) THV SEM; - - -) target value based on DNS data from Jiménez et al. [51]. 101

3.21 Vertical profiles of the non-zero Reynolds stresses at the inlet. blue) THV SEM; - - -) target values based on DNS data from Jiménez et al. [51]. 102

3.22 Vertical profile of the mean streamwise velocity far downstream of the inlet \( (x = 34\delta_{BL}) \) compared with the experimental data collected by Mehta [95]. \( U_c \) is the convection velocity \( (U_{high} + U_{low})/2 \). blue) THV SEM; O) experimental; - - -) error function, \( erf(\xi)/2 \). 103

3.23 Vertical profiles of the non-zero Reynolds stresses far downstream of the inlet \( (x = 34\delta_{BL}) \) compared with the experimental data collected by Mehta [95]. blue) THV SEM; O) experimental. 104

3.24 Contour of the streamlines of a single Hill’s vortex: a) without distortion; b) with distortion. 106

3.25 Contours of the velocity magnitude on xy (left), xz (middle), and yz (right) planes through the center of an undistorted THV (top) and a distorted THV (bottom). 108

3.26 A single undistorted THV (top) and a single distorted THV (bottom) being generated at the inlet. Time is increasing with a constant time step from a) to d) and the inlet is on the left side of each frame. 109

ix

Distribution Statement A. Approved for public release; distribution is unlimited.
3.27 Probability density function at the inlet calculated along the homogeneous: a) $y$-direction; b) $z$-direction; O) numerical; ——) Gaussian. 109

4.1 Example Control Forced Concurrent Precursor Method domains. 124

4.2 The physical dimensions of the domain used for comparison to the experimental results of Castro and Robins [21]. The blue line represents the single controlled forcing plane. 130

4.3 Turbulent statistics along the vertical direction from the precursor domain, using the proposed forcing method and three grid resolutions, sampled at the controlled forcing plane at conditions matching the momentum thickness Reynolds number of Castro and Robbins [21]. -.-.-) coarse; - - -) medium; ——) fine; X) experimental data 132

4.4 Contour of the instantaneous streamwise velocity normalized by the freestream velocity on an xz plane through the center of the precursor domain, at conditions matching the momentum thickness Reynolds number in Castro and Robins [21]. The black line represents the controlled forcing plane. left) “no forcing”; middle) “original forcing”; right) proposed forcing 133

4.5 Contour of the instantaneous streamwise velocity normalized by the freestream velocity on an xz plane through the center of the main domain, at conditions matching the momentum thickness Reynolds number in Castro and Robins [21]. left) “no forcing”; middle) “original forcing”; right) proposed forcing 134

4.6 Turbulent statistics along the vertical direction from the precursor domain sampled at the controlled forcing plane at conditions matching the momentum thickness Reynolds number of Castro and Robbins [21]. -.-.-) “no forcing”; - - -) “original forcing”; ——) proposed forcing; X) experimental data 135

4.7 Mean streamwise velocity along the vertical direction sampled at streamwise locations downstream of the cube. -.-.-) no forcing; - - -) original forcing; ——) proposed forcing; X) experimental data for the high Reynolds number case of Castro and Robins [21]. 136

4.8 Streamwise turbulence intensity along the vertical direction sampled at streamwise locations downstream of the cube. -.-.-) no forcing; - - -) original forcing; ——) proposed forcing; X) experimental data for the high Reynolds number case of Castro and Robins [21]. 137

Distribution Statement A. Approved for public release; distribution is unlimited.
4.9 The physical dimensions of the domain used for comparison to the experimental results of Jovic and Driver [53]. The blue line represents the single controlled forcing plane.

4.10 Turbulent statistics along the vertical direction from the precursor domain, using the proposed forcing method and three grid resolutions, sampled at the controlled forcing plane at conditions matching the momentum thickness Reynolds number of Jovic and Driver [53]. -.-.-) coarse; - - -) medium; —–) fine; X) experimental data.

4.11 Contour of the instantaneous streamwise velocity normalized by the freestream velocity on an xz plane through the center of the precursor domain, at conditions matching the momentum thickness Reynolds number in Jovic and Driver [53]. The black line represents the controlled forcing plane. top) “no forcing”; middle) “original forcing”; bottom) proposed forcing.

4.12 Contour of the instantaneous streamwise velocity normalized by the freestream velocity on an xz plane through the center of the main domain, at conditions matching the momentum thickness Reynolds number in Jovic and Driver [53]. The black line represents the controlled forcing plane. top) “no forcing”; middle) “original forcing”; bottom) proposed forcing.

4.13 Turbulent statistics along the vertical direction from the precursor domain sampled at the controlled forcing plane at conditions matching the momentum thickness Reynolds number of Jovic and Driver [53]. -.-.-) “no forcing”; - - -) “original forcing”; —–) proposed forcing; X) experimental data.

4.14 Mean streamwise velocity along the vertical direction sampled at streamwise locations downstream of the step. -.-.-) “no forcing”; - - -) “original forcing”; —–) proposed forcing; X) experimental data for the low Reynolds number case of Jovic and Driver [53].

4.15 2D turbulent kinetic energy along the vertical direction sampled at streamwise locations downstream of the step. -.-.-) “no forcing”; - - -) “original forcing”; —–) proposed forcing; X) experimental data for the low Reynolds number case of Jovic and Driver [53].

4.16 Reynolds shear stress along the vertical direction sampled at streamwise locations downstream of the step. -.-.-) “no forcing”; - - -) “original forcing”; —–) proposed forcing; X) experimental data for the low Reynolds number case of Jovic and Driver [53].

Distribution Statement A. Approved for public release; distribution is unlimited.
A.1 Isosurfaces of Q-criterion for the DNS and different SGS cases at the same non-dimensional time. .......................................................... 165

A.2 Temporal evolution of the momentum thickness of the mixing layer. The symbols all represent data from from Vreman et al. [163]. blue) no SGS model; red) Smagorinsky SGS model; yellow) Coherent Structure Model; O) no SGS model; Δ) Smagorinsky SGS model; +) Dynamic Smagorinsky SGS model. .......................................................... 166

A.3 Vertical profiles of the turbulence intensities, \( \sqrt{\langle \rho u'_i u'_i \rangle} \), and Reynolds shear stress, \( -\langle \rho u' v' \rangle \), for the temporal mixing layer at \( tU_\infty/\delta_{\omega(0)} = 70 \). yellow) Coherent Structure Model; +) Dynamic Smagorinsky SGS model from Vreman et al. [163]. .......................................................... 167
CHAPTER 1

INTRODUCTION

Increases in computing power have led to the application of Large Eddy Simulations (LES) and Direct Numerical Simulations (DNS) to a wider range of turbulent flows. The ability to simulate spatially-developing fully-turbulent flow over complex geometries using LES (and over simple geometries using DNS) has become more of a reality. Unlike temporally-developing flows, which can utilize periodic domains, a spatially-developing simulation requires a turbulent flow field be supplied at the inlet. Imposing high-quality realistic turbulent flow at an inlet is crucial for an efficient, accurate simulation. The behavior of the turbulent flow in the interior of the domain is highly dependent on the physical quality of the flow imposed at the inlet.

The development of different hybrid Reynolds-Averaged Navier-Stokes – Large Eddy Simulation (RANS-LES) approaches introduced another level of complexity into the field of turbulence generation. In these hybrid methods, the overall domain is divided into different RANS regions and LES regions as a means to decrease the computational cost as compared with using LES over the entire domain. The interface between the RANS and LES domains needs to efficiently turn the averaged flow statistics from the RANS domain into realistic turbulence for the LES domain, where the injection of improper fluctuations
can greatly increase the size of the required LES region and disrupt the overall results [144]. Even for temporally-developing flows, there is a need to generate realistic initial conditions to shorten the transition time from unrealistic to realistic turbulence in the whole domain.

Tabor and Baba-Ahmadi [158] outlined the following general requirements for any turbulent inflow generation method.

- Produce stochastically varying fluctuations.
- Produce fluctuations at all of the resolved temporal and spatial scales.
- Produce fluctuations that are consistent with the Navier-Stokes equations.
- The resultant fluctuating field should appear intrinsically like turbulence.
- The method should have the ability to use a wide range of given turbulent statistics as inputs.
- The method should be efficient and straightforward to implement.

Dhamankar [32] also added that a method should not depend on the inflow geometry or the spatial discretization of the grid.

The first four points on the list are about faithfully reproducing all of the characteristics of realistic turbulence at the inlet. This involves satisfying a wide array of single and multipoint statistical quantities that describe a range of physical phenomena [166]. Turbulent flow is comprised of coherent structures over a large range of length scales that interact with each other in a correlated fashion. The deformation and orientation of these structures are also key aspects of what makes a “physical” turbulent flow. If the imposed inflow for a wall-bounded flow has favorable statistical matching, but does not create the characteristic stretched and bursting fluid structures at the wall, then the inflow is not a completely

Distribution Statement A. Approved for public release; distribution is unlimited.
accurate representation of a realistic turbulent flow. When the imposed fluctuations are less than physical, a development region is required to allow for the transition to realistic turbulence. This development region is an increased length of the domain upstream of area of investigation, thus increasing the computational cost of the overall simulation.

The last two points on the list and the additional point by Dhamankar [32] address the applicability of a method over a wide range of problems. Depending on the application, different input statistics may be available. For example, a RANS solution of turbulent channel flow will provide a limited set of statistical quantities as compared to DNS results [136]. The inflow generation method should be adaptable enough to consider any subset of inputs and should not be the limiting factor when considering the overall computational cost of a simulation.

Various methods have been developed for specifying turbulent inflow boundary conditions that fall within two categories: precursor methods and synthetic turbulence methods. Precursor methods use a separate simulation to provide realistic inflow conditions that satisfy the Navier-Stokes equations, whereas synthetic turbulence methods use various degrees of modeling to approximate real turbulence. This work proposes new turbulent inflow generation methods in both categories: a synthetic turbulence method and a precursor method.

The next chapter outlines the different turbulent inflow generation methods that exist within the broad precursor and synthetic turbulence distinctions. In the subsequent chapters, each of the proposed methods are detailed. Each method’s chapter is split into an overview of the governing equations and numerical methods utilized, a description of the
proposed methodology, presentation of results, and a discussion of method specific conclusions and future work.

1.1 Contributions

The main contributions in this dissertation are:

- the creation of a new divergence-free synthetic eddy by superposing three orthogonal Hill’s vortices
- the introduction of a new divergence-free synthetic eddy method based on combination of coherent structures where the Reynolds stress tensor is exactly matched through a coordinated transform of the amplitudes of the synthetic eddies
- the application of the controlled forcing method to multiple velocity components and coupling their calculation in order to reproduce given anisotropic Reynolds stress tensors
- the combination of the mean and fluctuating flow controlled forcing methods with a concurrent precursor method

In support of the Triple Hill’s Vortex Synthetic Eddy Method, the secondary contributions are:

- the implementation of the THV amplitude calculation using the eigenvalues of the Reynolds stress tensor
- the implementation of the reference frame transformation using the eigenvectors of the Reynolds stress tensor
- the implementation of the multiscale generations of THV’s
- the implementation of the proportionally controlled amplitude scaling
- the implementation of the near-wall stretched THV’s
- the extension of the distortion from a single Hill’s vortex to a THV
- the implementation of the Smagorinsky SGS Model
- the implementation of the compressible Coherent Structure SGS Model
- the implementation of artificial viscosity outflow region based on the local SGS stress

Distribution Statement A. Approved for public release; distribution is unlimited.
In support of the Control Forced Concurrent Precursor Method, the secondary contributions are:

- the implementation of the concurrent precursor method
- the implementation of the recycling precursor method
- the implementation of the direct forcing immersed boundary method
- the implementation of the viscous diffusion term
- the implementation of non-zero wall velocity boundary condition
- three-dimensional snapshot proper orthogonal decomposition (POD) analyses
CHAPTER 2

OVERVIEW OF TURBULENT INFLOW GENERATION METHODS

The following is a broad overview of the many different kinds of inflow generation methods split into two major categories: Precursor Methods and Synthetic Turbulence Methods. Although not a method for generating fluctuations in itself, a section discussing the Controlled Forcing Methods is also included because of its association with multiple inflow generation methods, targeting the acceleration of the transition from imposed fluctuations to realistic turbulence.

2.1 Precursor Methods

The precursor methods provide turbulent inflow conditions by running a separate simulation. The fluctuations imposed at an inlet are actual solutions to the Navier-Stokes equations and capture all of the higher order spatial and temporal statistics along with the physical dynamics associated with the coherent flow structures. Accurate precursor methods generate the most realistic turbulent inflow condition, but with the added cost of running a separate simulation.

The following precursor methods are further divided into concurrent and database methods depending on whether the inflow conditions are transferred directly to the main simulation or stored to disk first.
2.1.1 Concurrent Methods

The concurrent precursor method defines a separate precursor simulation that is run simultaneously with the main simulation. Turbulent data is transferred from the precursor simulation to the main simulation to be used as inflow conditions. It is not a requirement that the precursor simulation be computed on a separate domain. A single domain can be employed where the “outlet” of a distinct precursor region is also the “inlet” for the area of investigation. Inflow conditions for the precursor region are provided by mapping, or recycling, a slice of flow data from the “outlet” to the inlet of the domain. The precursor simulation techniques presented below are equally valid for separated or connected precursor and main domains. The character of the precursor simulation is dependent on the type of flow being studied.

For fully-developed turbulent flow, a periodic precursor domain provides an efficient means for generating inflow conditions over a relatively short domain length Baba-Ahmadi [158]. The most accurate concurrent method is one where the precursor domain is identical to and synchronized in time with the main domain. A slice of precursor data can be copied directly to the inlet of the main simulation at each time step. The inflow data is a solution of the Navier-Stokes equations for the flow conditions and geometry of the main simulation. For a main domain with periodic boundary conditions, like those required for spectral and pseudo-spectral discretizations, Stevens et al. [156] proposed a method that blends a region of the precursor flow field onto the end of the main domain. The precursor and main simulations are run on identical grids and are synchronized in time. The blending operation consists of copying the region of precursor flow into the main domain.
and then using a blending function to remove the discontinuity between the main flow and
the copied flow. This operation occurs outside of the solution of the Navier-Stokes equa-
tions and only produces a $C^0$ continuous flow field which was found to introduce spurious
oscillations in spectral and pseudo-spectral methods. Munters et al. [108] later replaced
the blending operation with a penalization region and expanded the method to account
for time-dependent inflow freestream direction. The penalization region forces the main
flow towards the corresponding precursor flow in that region through the addition of body
forces to the governing equations. Because the transfer of the precursor flow to the main
domain occurs inside the solution the Navier-Stokes equations, there is a more physical
transition from the main flow to the precursor flow in the main domain. Haywood and
Sescu [43] then applied mean and fluctuating flow controlled forcing within the precursor
domain to impose specific Reynolds stress tensors and mean velocities. If simulating the
exact same domain for the precursor simulations is cost prohibitive, several simplifications
can be made, but those require additional procedures to provide relevant turbulent fluctu-
atations. By using a precursor domain that is shorter in streamwise direction than the main
domain, oscillations can be introduced that can be removed using one of the techniques
presented below. Streamwise periodicity is also introduced into the main simulation by
using a shortened precursor domain because the flow structures do not have enough time
to evolve before the inlet data is repeated. An interpolation procedure would be required
if the precursor and main domains were of different dimensions or discretizations. If the
precursor and main simulations were at different flow conditions, the rescaling techniques

Distribution Statement A. Approved for public release; distribution is unlimited.
presented in the Database Methods section (Section 2.1.2) could be used to provide appropriate inflow conditions.

For spatially-developing turbulent boundary layers, the flow is inhomogeneous in the streamwise direction, thus making the use of a periodic domain inappropriate. In the context of equilibrium boundary layers, Spalart and Leonard [151] proposed a method based on the assumption that the mean flow and the Reynolds stresses satisfy similarity properties. In the method, the physical coordinate system, where the flow is non-periodic, is transformed into a self-similar coordinate system, where the flow is periodic, thus enabling the use of a spectral numerical solver. Numerous source terms are added to the momentum equations to facilitate this transformation. Spalart [152] suggested a refined method to calculate the source terms, but it required extra simulations. The last real attempt to utilize periodic domains for spatially-developing flows was by Spalart and Watmuff [153]. A fringe region is added, where source terms in the momentum equations are used to damp the growth of the boundary layer. While simpler and more widely applicable than the similarity-based methods, the fringe-based method still proved to be rather complicated.

Looking for a simpler method, Lund et al. [83] proposed the recycling/rescaling method. While the concept of rescaling the velocity using various similarity laws from Spalart and Leonard [151] is preserved, the recycling/rescaling method only applies the rescaling operation over a wall-normal plane, thus destroying any chance of using periodic boundary conditions. The method samples a plane of data normal to the streamwise direction near the outlet of the domain. Using appropriate similarity laws for each wall-normal region, the inner and outer velocity profiles are rescaled and reintroduced at the inlet. Spatially-

Distribution Statement A. Approved for public release; distribution is unlimited.
Figure 2.1

Diagrams showing the recycling method with a separate precursor domain (top) and a precursor region connected to the main domain (bottom).
developing turbulent boundary layers were simulated that compared exceedingly well to a canonical boundary layer and to results from Spalart and Leonard [151]. Also, no development length was found to be necessary when the recycling/rescaling method was used to generate inflow conditions for an LES study of flat plate boundary layers. Mayor et al. [93] applied the Lund et al. [83] recycling/rescaling method to the internal boundary layer formed by cold air moving over warm water. Ferrante and Elghobashi [36] tried to apply the original recycling/rescaling method to a DNS study of a flat plate boundary layer and failed to achieve proper development of the Reynolds stresses because the initial uncorrelated fluctuations present in the precursor domain were quickly dissipated. As a correction, an initial fluctuating velocity field for the precursor domain was created using the Fourier synthetic turbulence method of Le and Moin [75]. With the better correlated initial flow field, excellent agreement with experimental data was found. Liu and Pletcher [80] posed a different solution to the problem of poorly posed initial conditions. During the initial transient state, Liu and Pletcher [80] suggested that the location of the recycling plane be dynamically positioned so that it be consistently located in the most turbulent region of the precursor domain. Modest improvements in the development in the skin friction were seen over using a stationary recycling plane. In their study, Liu and Pletcher [80] only used an initial field made up of random fluctuations, so no comparison was made between the dynamic recycling plane method and the original stationary plane method with better correlated initial condition. Nikitin [111] showed that spatial periodicity in the main domain, whose frequency is inversely dependent on the length of the recycling region, can be introduced by a precursor simulation; which results in the streamwise repetition of the
precursor flow throughout the main domain. While utilizing a shortened recycling domain, both Simens et al. [146] and Lee and Sung [78] also found significant secondary peaks in the streamwise two point spatial velocity correlations in their simulations of turbulent boundary layer flow. To break the spatial periodicity, Spalart et al. [154] suggested a constant shift of the recycling plane flow field in a homogeneous direction in order to keep the fluctuations imposed at the inlet of the main domain out of phase with the fluctuations at the recycling plane. Jewkes et al. [50] proposed that the flow field from the recycling plane be mirrored about a homogeneous direction before being recycling back to the inlet to prevent the accumulation of errors caused by a short recycling region. Morgan et al. [104] applied a combination of a shift in a homogeneous direction and a reflection about a homogeneous direction to the flow field at the recycling plane. The shifting-reflection operation would occur randomly in time for every flow-through of the domain. Araya et al. [6] coupled the original recycling/rescaling method with a dynamic method for calculating the rescaling relationships to be able to simulate turbulent boundary layers with pressure gradients. Araya and Castillo [7] further extended the coupled method for use on thermal boundary layers. Xiao et al. [167] generalized the overall recycling/rescaling methodology to be applicable to non-equilibrium inflow conditions without a homogeneous direction. This was done by removing the similarity property-based rescaling and replacing it with a process that rescales the velocity fluctuations based on prescribed normal Reynolds stress profiles. The entire precursor domain is rescaled every certain amount of iterations. These rescaling procedures all occur outside of the solution of the governing equations. When applied to an
LES investigation of droplet dispersion across a turbulent mixing layer, promising results were seen when compared to experimental data.

Concurrent precursor simulations identical to the main simulations can provide the most realistic turbulent inflow conditions, but that comes with the doubled computational cost of running another complete simulation. Without quality initial fluctuations present in the precursor domain, the concurrent precursor methods suffer from long development times before the desired fully turbulent flow is reached. The recycling/rescaling method has the ability to provide the same level of realistic inflow conditions for a limited subset flows, namely boundary layer flows. The equilibrium requirement for the recycling region severely limits the use of the recycling/rescaling method on complex flows, while the introduction of spurious low-frequency disturbances limits their use on aeroacoustic applications.

2.1.2 Database Methods

The database precursor method utilizes an external library, or database, of turbulent flow fields as a source for the inflow conditions. The database is created by running a separate simulation using any of the other turbulence generation methods discussed here and writing instantaneous flow fields into memory. These flow fields should contain realistic instances of a turbulent flow. At each time step, a slice of the turbulent velocity field from the library is read from memory by the main simulation and imposed at the inlet. As with the concurrent method, the ideal database would be created using a simulation at the same

Distribution Statement A. Approved for public release; distribution is unlimited.
flow conditions and with the same domain as the main simulation. It should be noted that a database can also be created using sufficiently resolved experimental measurements [91].

Chung and Sung [26] classified database methods into two groups: temporal and spatial. For the temporal database method, instantaneous planes of data are sampled from the precursor simulation at one streamwise location in time. A slice of the flow field should be captured for each time step required by the main simulation. This enables an ideal representation of the statistical quantities and dynamic characteristics of the turbulent flow to be imposed at the inflow, but with a cost. For large domains over many time steps, the memory required to store all of the flow slices and communication time required to transfer the slices from disk storage to the main simulation are unwieldy. To work around the need for large storage and data transfer resources, the spatial database method was introduced.

![Diagram of the temporal database method.](image)

Figure 2.2

Diagram of the temporal database method.
The central assumption of the spatial database method is that Taylor’s frozen turbulence hypothesis is applicable for constant convection (Lee et al. [77]). In the original spatial method, a single three-dimensional turbulent flow field is written to memory. As the main simulation is evolved in time, normal planes of the database flow field are sampled along the streamwise direction, with the sampling location moving downstream according to the convection velocity. Evolution in time for the main simulation corresponds to evolution in space for the spatial database. For main simulations requiring many time steps or a large convection velocity, a precursor simulation with a prohibitively large domain would have to be carried out; but if a shortened precursor domain is employed, the main simulation would have to sweep over the spatial database multiple times to provide enough inflow planes, thus introducing spatial periodicity into the main simulation.

![Diagram of the spatial database method.](image)

Several techniques have been proposed to correct for the introduction of spatial periodicity while still being able to use smaller spatial databases. Na and Moin [109] utilized the amplitude jittering method developed by Mahesh et al. [87]. In their proposed method, a three-dimensional flow field is transformed into the frequency domain using a Fourier transform.
transform. The transformed field is multiplied by a random amplitude, in order to preserve the realistic phase angle behavior, and then transformed back to the physical domain using an inverse Fourier transform. The main simulation sweeps through the modified database field until it reaches the end, then a new database field is created using another random amplitude. The reduction in spatial periodicity is directly related to the magnitude of the amplitude, but if the amplitude is too large, the flow field will no longer provide appropriate inflow conditions for the main simulation. Chung and Sung [26] mentioned replacing amplitude jittering with the phase jittering method developed by Lee et al. [77]. Instead of modifying the amplitude, the phase angles are shifted randomly. In tests of the two, it was seen that amplitude jittering better preserved the turbulent structures in the original spatial database and was thus concluded to be the superior method. Both of the jittering methods require the Fourier and inverse Fourier transforms of the entire database domain which may be computationally expensive or not feasible depending on the domain.

As a simpler method that does not require the use of Fourier transforms, Chung and Sung [26] proposed a spatiotemporal database that looked to take advantage of the behavior in time of a temporal database while utilizing the smaller spatial database sizes. The spatiotemporal database method saves multiple three-dimensional flow fields at different time steps corresponding to the time scale of the largest-scale turbulent motion to ensure that each flow field is independent from each other. The main simulation sweeps through the first database flow field sampling inflow planes. When the end of the first domain is reached, the second database flow field is loaded and the sweep continues at the beginning of that domain. The loading and sweeping of new database domains continues until the
Diagram of the spatiotemporal database method. $\Delta T$ corresponds to the time scale of the largest-scale turbulent motion.

main simulation is complete. By only imposing each database flow field once, no spatial periodicity is introduced; but discontinuities in the flow are introduced at the transfer point by switching database flow fields. Xiong et al. [169] sought to remove the discontinuities by combining all of the spatial flow fields into one continuous field that could be swept straight through. Each of the independent spatial database flow fields is blended together, outlet to inlet, in such a way to retain the second-order statistics of the flow fields being blended. The blending operation introduces an additional dilatation term in the blending region that is corrected for by solving a Poisson equation over the entire combined spatial database domain. Solving a Poisson equation over an extremely large combined domain could be computationally costly. Seeking to reduce the blending procedure cost, Larsson
[74] used a least-squares approach to localize the removal of the dilatation term to each of the separate blending regions. Not only does this allow for reduced computational costs, but it also allows for the blending of additional database flow fields while the main simulation is still running.

For flows that are not fully-developed or have unsteady mean velocity profiles, like those seen at RANS-LES interfaces, Schlüter et al. [136] proposed a method based on scaling the velocity fluctuations extracted from a database flow field. The velocity field imposed at the inlet of the main simulation, shown below, is a given time-dependent mean

---

Figure 2.5

Diagram of the spatiotemporal database method with blending.
velocity profile combined with the fluctuations sampled from a database flow field plane scaled to match given RMS velocity profiles.

\[ u_i(t) = \langle u_i \rangle_{given}(t) + [(u_i)_{DB}(t) - \langle u_i \rangle_{DB}] \frac{\sqrt{\langle u_i' u_i' \rangle_{given}(t)}}{\sqrt{\langle u_i' u_i' \rangle_{DB}}} \]  \hspace{1cm} (2.1)

In the formulation by Schlüter et al. [136], the given profiles are provided by a RANS solution, hence why time-averaged quantities are shown to be time-dependent. Keating et al [56] compared simulations of turbulent channel flow using a temporal database with simulations that used the Schlüter et al. [136] method. Two different spatial databases were considered: one at a Reynolds number close to the main simulation Reynolds number and one at an inappropriate Reynolds number. The case with the appropriate Reynolds number compared quite well with a temporal database case, while the inappropriate database case required a modest development length that fell between the lengths required for the Fourier synthetic turbulence method of Batten et al. [10] with and without the controlled forcing method of Spille-Kohoff and Kaltenbach [155]. Schlüter et al. [138] then combined this database inflow generation method with the outflow control forcing method of Schlüter et al. [137] to study the multicomponent effects in gas turbines.

Pierce and Moin [119] introduced a method to create swirling flow from fully-developed pipe flow using a body force added to the momentum equations. Wang et. al [165] utilized this method to generate a database at a single swirl number. That database was then applied to main simulations over a range of swirl numbers using a rescaling method, introduced by Pierce [120], that matched measured experimental statistics. It was seen that as
the difference between the swirl number of the database and the swirl number of the main simulation grew larger, the simulated flow deviated further from the experimental results.

When the turbulent flow contained in a database is similar to the turbulent flow required at the inflow of a main simulation, high-quality realistic turbulence can be imposed. The main issues arise from using a limited database and trying to apply databases to domains and flows with vastly different conditions. Extra-long development lengths and spatial repetition of the database flow in the main domain can occur without proper treatment of the database. While databases have been successfully used on the standard canonical flows, for the complex flow conditions and geometries seen in general applications, a database cannot be created to satisfy the necessary inflow conditions or would require prohibitively large computer resources to be practical.

2.2 Synthetic Turbulence Method

The synthetic turbulence methods forgo the explicit solution of the Navier-Stokes equations and impose fluctuations that model real turbulence. The synthetic fluctuations are transformed over a certain development length by the simulated Navier-Stokes equations into realistic turbulence. Synthetic turbulence methods generally have lower computation cost than the precursor methods, but at the cost of imposing inflow conditions that are less realistic.

Three main classes of methods exist: statistical, decompositional, and physical. The statistical methods (Random Fluctuation, Digital Filtering, Diffusion) modify fields of random fluctuations as a means to match as many turbulent statistics as possible. The de-
compositional methods (Fourier, Proper Orthogonal Decomposition) decompose a turbulent flow, either by frequency or energy, and then use limited combinations of statistically modified modes to create the synthetic inflow. The physical methods (Wavelet, Random Turbulent Spot, Vortex, Synthetic Eddy) try to model the coherent structures present in turbulence through the combination of discrete synthetic structures. There is a trade-off between a statistical description of the synthetic turbulence and a physical description for each of the classes.

2.2.1 Random Fluctuation Method

The simplest method for modeling the turbulent fluctuations is using pure white noise, where independent random fields, $r_i$ are added to a mean velocity profile, $\langle u_i \rangle$.

$$u_i = \langle u_i \rangle + r_i \sqrt{\frac{2}{3} k}$$  \hspace{1cm} (2.2)

The random fields are defined to have zero mean and unit variance and can be scaled by the turbulent kinetic energy, $k$. Inflow generated with this method reproduces prescribed mean velocity profiles and turbulent kinetic energy levels; but otherwise produces an uncorrelated, random flow, i.e. the velocity cross-correlations and the two-point spatial and temporal correlations are zero.
In an effort to ensure the cross-correlations of the velocity components are non-zero, Lund et al. [83] introduced an improvement based on the Cholesky decomposition of the Reynolds stress tensor.

\[ a_{ij} = \begin{bmatrix}
\sqrt{R_{11}} & 0 & 0 \\
\frac{R_{21}}{a_{11}} & \sqrt{R_{22} - a_{21}^2} & 0 \\
\frac{R_{31}}{a_{11}} & \frac{R_{32} - a_{21} a_{31}}{a_{22}} & \sqrt{R_{33} - a_{31}^2 - a_{32}^2}
\end{bmatrix} \]  \hspace{1cm} (2.3)

\( R_{ij} \) are the components of the Reynolds stress tensor. Three independent random fields with zero mean and unit variance are again created, but they are transformed by the Cholesky decomposition of the Reynolds stress tensor instead of scaled by the turbulent kinetic energy. The transformed fluctuations are then added to the proper mean velocity profiles.

\[ u_i = \langle u_i \rangle + a_{ij} r_j \]  \hspace{1cm} (2.4)

This improved method has the ability to match a given Reynolds stress tensor, but it still produces a flow field that is uncorrelated in space and time. In addition, the turbulent kinetic energy is spread equally over all wavenumbers, which is in opposition to the observed energy spectrum of real turbulence [66]. Since the quickly dissipating small scale turbulent structures contain an excessive amount of energy while the more stable large, supposedly energy-containing, structures are energy deficient, the fluctuations introduced by a random fluctuation method are quickly dissipated near the inlet. This has been shown by Klein et al. [66], Aider et al. [5], and Rana et al. [126], amongst others. In terms of turbulent boundary layer flows, Klein et al. [66] even suggested it would be more efficient to start with a laminar profile than to use a random fluctuation method because of

Distribution Statement A. Approved for public release; distribution is unlimited.
how close relaminarization occurred to the inlet. In comparing the improvement proposed by Lund et al. [83] with a pure white noise random fluctuation method, Rana et al. [126] observed that the flow took slightly longer to relaminarize.

2.2.2 Digital Filtering Methods

The origins of the use of digital filters in the generation of synthetic turbulence can be traced back to the Fourier method of Béchara et al.[12] (Wu [166]). In their initial study, Béchara et al.[12] found that each spatially correlated flow field in the time series of inflow data was independent, resulting in a white noise signal in time. As a correction for the temporal correlation, a convolution of the time series and a Gaussian filter was performed. The concept of applying digital filters to introduce correlations in otherwise uncorrelated data can also be applied spatially. This is the aim of synthetic turbulence inflow methods based on digital filtering.

The method proposed by Klein et al.[66] looked to remedy the uncorrelated nature of the inflow generated by the random fluctuation method of Lund et al.[83] by filtering the random fluctuations. The instantaneous velocity is decomposed as follows,

\[ u_i = \langle u_i \rangle + a_{ij} U_j \]  

(2.5)

where \( \langle u_i \rangle \) is the time-averaged velocity, \( a_{ij} \) is the Cholesky decomposition of the Reynolds stress tensor, and \( U_j \) is the digital filtered velocity. \( U_j \) is defined such that: \( \langle U_j \rangle = 0 \) and
\( \langle U_j U_j \rangle = 1 \). In one-dimension, the convolution in physical space of a random series and a digital linear non-recursive filter is defined as

\[
 u_m = \sum_{n=-N}^{N} b_n r_{m+n}
\]  

(2.6)

where \( u_m \) is one component of the velocity at one point in space or time, \( N \) is the length scale of the filter, \( b_n \) are the filter coefficients, and \( r_m \) is a random series of data with \( \langle r_m \rangle = 0 \) and \( \langle r_m r_m \rangle = 1 \). A relation between the filter coefficients and the two-point autocorrelation function of \( u_m \) can be found.

\[
 \frac{\langle u_m u_{m+k} \rangle}{\langle u_m u_m \rangle} = \frac{\sum_{j=-N+k}^{N} b_j b_{j-k}}{\sum_{j=-N}^{N} b_j b_j}
\]  

(2.7)

The procedure extends simply to two- or three-dimensions, depending on whether a plane or field of inflow data is generated, through the convolution of two or three one-dimensional filters. Kempf et al. [59] was able to show a significant reduction in computational cost by defining the filter as a tensor product of one-dimensional filters. Di Mare et al. [33] showed that the filter coefficients can be calculated from any known autocorrelation function. Because the complete autocorrelation function is not generally known for all cases, multiple approximations have been proposed. Klein et al.[66] suggested a Gaussian form dependent only on a single length scale that needed to be prescribed, while Xie and Castro [168] found that an exponential form based on the integral length scale fit better for turbulent shear flows. The exponential function was updated after the study of turbulent channel flows by Kim et al. [63]. Veloudis et al. [162] investigated Gaussian approximations based on multiple length scales and for non-uniform grids.
To overcome the computationally expensive three-dimensional filtering required by the method proposed by Klein et al. [66], Xie and Castro [168] suggested an alternative method. Two-dimensional filtering is employed and the data generated at the next time step is correlated to the current time step data through the following exponentially decaying relation,

\[ \mathbf{U}_j = u_m(t + \Delta t) = u_m(t) \exp\left(-\frac{\pi \Delta t}{2T}\right) + u_{m\beta}(t) \sqrt{1 - \exp\left(-\frac{\pi \Delta t}{T}\right)} \]  \hspace{1cm} (2.8)

where \( T \) is the Lagrangian timescale and \( u_{m\beta} \) is calculated in the same manner as \( u_m \), but from an independent set of random data.

It should also be noted that Kim et al. [63] developed a divergence-free formulation. After the momentum equations are solved, the filtered fluctuations are imposed on a plane near the inlet. When the pressure-corrector step is completed, the flow field with the synthetic fluctuations satisfies the divergence-free condition. By using the pressure-corrector step to modify the synthetic turbulence to enforce the divergence-free condition, the statistics of the fluctuations are also modified such that they do not exactly match their imposed values. While reductions in spurious pressure fluctuations were seen, the overall development length remained largely unchanged [63, 64].

Dietzel et al. [34] showed that, for homogeneous isotropic turbulence generated using the Gaussian approximation and a single length scale, the initial synthetic velocity compared poorly with experimental data and only match the experimental velocity correlations and the energy spectrum after a lengthy development time. If given a high quality autocorrelation function, a digital filter-based method has the possibility of quickly producing
realistic turbulence. Also, the filtering is only applicable to uniformly space grids; varying grid spacings require additional filtering operations at the different filtering scales.

### 2.2.3 Diffusion Method

The diffusion method is based on the proof by Kempf et al. [58] that the homogeneous diffusion of \( u(x, t) \) is equivalent to the convolution of \( u_0(x) \) with a Gaussian filter, i.e. a digital filter method with a Gaussian filter can be replicated without the grid restrictions. In their proposed method, three fields of white noise, \( U_i \), are correlated spatially by the solution in time of the unsteady diffusion equation at each grid point until desired length scales are achieved.

\[
\frac{\partial U_i}{\partial t} = D \frac{\partial^2 U_i}{\partial x_j^2} \tag{2.9}
\]

The local length scale, \( L \), grows like

\[
L \propto \sqrt{nD} \tag{2.10}
\]

where \( D \) is the local diffusion coefficient and \( n \) is the number of diffusion time steps. The method is not restricted to a homogeneous field of diffusion coefficients; they can be varied spatially to account for the changing length scales in wall-bounded flows, for example. Once the diffused fields are normalized, they are then transformed using the Cholesky decomposition of the Reynolds stress tensor. Slices of the field are then saved to be used as inlet conditions. Kempf et al. [58] only offer a suggestion of generating three-dimensional fields, instead of slices, and projecting them to satisfy the divergence-free condition.

In the same evaluation, Dietzel et al. [34] found that the diffusion method suffered from the same long development time as the digital filtering method with Gaussian autocorre-
lation approximation for homogeneous isotropic turbulence, which is not surprising considering the theoretical equivalencies of the methods. The clear advantage of the diffusion method is that it does not have any grid restrictions; it is suitable for complex geometries and unstructured grids, but at a large cost. A two-dimensional (or three-dimensional, in the case of a divergence-free method) unsteady diffusion equation must be iterated over at for each time step of the main simulation, which for large or complex inlets would significantly increase the computational cost.

2.2.4 Fourier Methods

The Fourier methods are a frequency-based attempt to model the energy containing structures using Fourier analysis. The fluctuating velocity components are represented by a superposition of a finite number, \( N \), of random Fourier modes with amplitudes, \( \hat{u}^n \), that allow for the matching of a prescribed energy spectrum.

\[
\begin{align*}
  u_i'(x_j) &= \sum_{n=1}^{N} \hat{u}^n \cos \left( \kappa_j^n x_j + \psi^n \right) \sigma_i^n \\
  &\quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \quad \q
modes and prescribed energy spectrum, but the correlation in time is still zero. As mentioned above, Béchara et al. [12] introduced temporal correlation using a filtering operation in the frequency domain, while Bailly and Juvé [8] added a dependence in time through the inclusion of a constant convection velocity. Davidson [30] applied an asymmetric time filter to impose specific time correlations.

Lee et al. [77] developed a Fourier method based on the inverse Fourier transform of a prescribed three-dimensional energy spectrum combined with random phase angles. To break the periodicity in time of the fluctuating velocity field introduced by the inverse Fourier transform, the phase angles were randomly shifted over time. However, if the phase angles are shifted too much, the prescribed energy spectrum will no longer be matched. Le and Moin [75] extended the method to inhomogeneous flows through scaling the synthetic velocities to match given Reynolds stresses. Both Le and Moin [75] and Le et al. [76] observed quick dissipation of the synthetic field followed by a long development length for turbulent channel flow and turbulent flow over a backward-facing step, respectively. This is consistent with the observations seen in other methods that use non-physical methods to alter the temporal correlation.

The initial method proposed by Kraichnan [72], and those that followed [55, 12, 8, 16, 30], ensure the divergence-free condition is satisfied for isotropic turbulence by requiring orthogonality between the wavenumber vector and the velocity unit vector for each mode. Building upon that, Smirnov et al. [150] introduced the Random Flow Generation (RFG) method to preserve the divergence-free aspect while also being able to generate anisotropic turbulence. Fluctuating fields are created by superposing harmonic functions.
with random amplitudes and phase angles using imposed turbulent length scales and time scales. The mode wavenumbers and frequencies are sampled from a prescribed energy spectrum. These fields then undergo a scaling and orthogonal transformation procedure to allow for matching of a given velocity correlation tensor. Smirnov et al. [150] proved that this method is divergence-free for homogeneous turbulence, but only approximately divergence-free for slowly spatially varying velocity correlation tensors. This is because the superposition of harmonic functions creates a global flow field, while the orthogonal transformation procedure becomes a local operation for inhomogeneous turbulence. Batten et al. [10] simplified the orthogonal transformation procedure by basing the transformation on the Cholesky decomposition of the Reynolds stress tensor. A side effect of using this simplification was that the divergence-free condition was violated. In tests by Keating et al. [56] on turbulent channel flow and Keating et al. [57] on flat plate boundary layer flow, realistic turbulence was slow to develop after the synthetic fluctuations quickly dissipated because of low levels of Reynolds shear stress production away from the walls caused by a lack of phase information. Huang et al. [45] improved the method of Smirnov et al. [150] to allow for the prescription of a different energy spectrum in each of the three dimensions, which also enabled the divergence-free condition to be approximately satisfied for turbulent flows with greater inhomogeneity, and suggested more realistic model energy spectra. Higher levels of turbulent intensity were seen immediately downstream of the inlet as compared to the original RFG method in a study of a turbulent boundary layer flow around a building. Castro and Paz [22] reformulated the definition of the mode amplitudes so that the imposed statistics did not vary with different discretizations of the prescribed energy
spectra. Patruno and Ricci [115] then introduced a correction that guarantees the generation of divergence-free homogeneous anisotropic fluctuations that also reproduce arbitrary spatial and temporal energy spectra. Like with the original method of Smirnov et al. [150], the method proposed by Patruno and Ricci [115] is only approximately divergence-free for slowly spatially varying inhomogeneous turbulence. Within the general framework of the RFG method, Yu and Bai [172] developed a method that is divergence-free for all cases of inhomogeneous anisotropic turbulence through the use of the velocity potential. Though not divergence-free, Shur et al. [144] presented a heavily modified version of the Batten et al. [10] method coupled with a damping layer at the inlet that was shown to successfully damp the spurious noise created by the synthetic turbulence generation in aeroacoustic simulations of a turbulent mixing layer and the flow over a wing-flap configuration.

A method to introduce the intrinsic non-Gaussian statistical quantities observed in realistic turbulence into a synthetic velocity field was proposed by Rosales and Meneveau [132]. The Multiscale Minimal Lagrangian Map (MMLM) approach starts with a Gaussian divergence-free synthetic velocity field created from the superposition of random Fourier modes and a prescribed energy spectrum. A Langrangian map is first applied on the largest length scales, then on successively smaller scales to distort the original field. Through this multiscale operation, the larger scales continue to be distorted by the operations on the smaller scales, thus modeling the multiscale characteristics seen in realistic turbulence. Using this method, reproduction of realistic skewness and non-Gaussian probability density functions were observed for homogeneous isotropic turbulence. Further investigation into the behavior of the MMLM approach in the inertial range was carried out in Rosales and
Meneveau [133]. Rosales [130] extended the MMLM approach to include the transport and mixing of passive scalars.

In the context of atmospheric boundary layer simulations, Muñoz-Esparza et al. [106] proposed a method that uses perturbations applied to the potential temperature in order to generate turbulence. A field of potential temperature fluctuations are created using a superposition of random harmonic modes where the amplitudes follow a normalized energy spectrum multiplied by the maximum allowed potential temperature perturbation. The potential temperature fluctuations are transferred to the velocity components through buoyancy, technically making this a divergence-free method. Muñoz-Esparza et al. [107] updated the definition of the maximum allowed potential temperature perturbation to induce larger vertical turbulent heat fluxes, thus decreasing the development time. When compared with the filtering method of Xie and Castro [168], the Muñoz-Esparza et al. [107] method exhibited a much shorter development length and produced more realistic turbulent structures throughout the domain for a neutrally stratified atmospheric boundary layer.

Fourier methods allow for a large amount of control over the synthetic velocity field because individual wavelengths can be varied to reproduce a wide range of turbulent statistics. However, the quality of the synthetic turbulence generated is strongly tied to the energy spectra imposed, which may not be known for all flows.
2.2.5 Proper Orthogonal Decomposition Methods

Proper orthogonal decomposition (POD) is an energy-based modal decomposition approach. The POD analysis decomposes the flow into orthogonal modes whose corresponding eigenvalues represent the contribution of each mode to the total turbulent kinetic energy of the flow. Lumley [82] was the first to apply POD analysis to turbulent flows. The large amount of data required limited the application to two-dimensional flow fields, until Sirovich [147] introduced the snapshot POD method. Using an ensemble of uncorrelated instantaneous flow fields, the snapshot POD method determines a set of basis functions (modes) ordered by decreasing eigenvalue magnitude (turbulent kinetic energy contribution). Each of the snapshots can be exactly recreated using a linear combination of all of the modes or partially recreated using a subset of the modes,

\[
    u_i(x, y, z, t) = \langle u_i(x, y, z) \rangle + \sum_{k=1}^{N} a^k(t_n) \psi_i^k(x, y, z),
\]

where \( \langle u_i \rangle \) is the time-averaged velocity field, \( N \) represents the number of snapshots, \( a^k \) are time-coefficients, and \( \psi_i^k \) represent the POD modes. The partial recreation enables an optimal modeling of the highest energy containing turbulent structures with a small number of modes.

The method introduced by Druault et al.[35] utilized the recreation of the largest coherent turbulent structure through POD analysis. At a plane normal to the mean flow, the following decomposition of the instantaneous velocity was considered,

\[
    u_i(x_0, y, z, t) = \langle u_i(x_0, y, z) \rangle + \hat{u}_i(x_0, y, z, t) + u'_i(x_0, y, z, t)
\]
where $\langle u_i \rangle$ is the time-averaged velocity, $\hat{u}_i$ is the velocity of the coherent fluctuations, and $u'_i$ is the velocity of the incoherent fluctuations. The coherent fluctuations were modeled using snapshot POD analysis of turbulent mixing layers for either hot wire experimental measurements or DNS precursor data. Linear stochastic estimation was also employed to correct for the lack of spatial resolution within the hot wire data. Uncorrelated, random noise was used for the incoherent fluctuations. Perret et al.\cite{118} used much the same approach, but based the POD analysis on experimental measurements collected using particle image velocimetry (PIV). Their analysis suffered from the opposite problem as the analysis using hot wire measurements; the PIV data provides good spatial resolution, but poor temporal resolution. The time coefficients for the coherent structures were solved for using a set of ordinary differential equations (ODEs). Lower energy modes with random time coefficients were added as incoherent fluctuations in place of the random noise to decrease the development distance. Johansson and Andersson \cite{52} also found the need to add random low energy modes during their study of turbulent channel using a library of DNS data for the POD analysis.

The ability of a POD synthetic turbulence method to provide realistic turbulent inflow conditions is dependent on having a large well-resolved, both spatially and temporally, dataset of the flow being simulated \cite{166}. This hinders the application of a POD-base inflow generation method to all but the most simple flows.
2.2.6 Wavelet Methods

The wavelet methods are rooted in the concept of the energy cascade first proposed by Richardson [127]; large turbulent structures breakdown into smaller and smaller structures, transferring energy along the way. In the one-dimensional wavelet method first proposed by Juneja et al. [54], the fluctuations caused by the different scale eddies are modeled as wavelets. Various methods have used different wavelet shapes: tent function [54], Meyer wavelet [65], Haar wavelet [170], polynomial form based on the vector potential [88], and wavelet orthogonal basis [174]. Wavelets of different scales are then linearly superposed following a model for the energy dissipation. Meneveau and Sreenivasan [97] proposed a simple multifractal model for the energy cascade in the inertial region called the $p$ model, which was derived from the multifractal description of the energy dissipation rate proposed by Frisch and Parisi [38] and confirmed by Meneveau and Sreenivasan [96] experimental data. The $p$ model consists of multiple levels, $n$, that contain $2^n$ units on each level. The amplitude and size of the first level correspond to the mean dissipation and a characteristic length scale, respectively. The length scale of each successive level is halved, while the amplitude is based on the dissipation at that level’s length scale calculated from a given energy spectrum. It is at each of these units a wavelet is placed.

Juneja et al. [54] showed a favorable comparison of one-dimensional statistics between their method and experimental data. By improving the the wavelet form, from a tent function to a Meyer wavelet, Kitagawa and Nomura [65] were able to generate one-dimensional synthetic velocities that mimicked the intermittency of experimental wind data. Zhou et al. [174] developed and included an energy cascade model for the dissipation region in one
dimension, which was later extended to two dimensions by Zhou et al. [175]. Malara et al. [88] separately developed a three-dimensional method based on the modified $p$ model.

Zhou et al. [175] included two suggestions on how to make their method divergence-free, either through a Helmholtz decomposition or the use of a divergence-free wavelet, but neither suggestion was implemented. Since the three-dimensional wavelet was defined using a vector potential, the method of Malara et al. [88] is inherently divergence-free.

The main advantage of the wavelet-based synthetic turbulence method is the ease with which they are able to reproduce the natural intermittency of a turbulent flow. One-dimensional comparisons with experimental data have generally been favorable; but with only recent extensions to two and three dimensions, it is still relatively unknown how a wavelet-based methods would compare against the more mature synthetic turbulence generation techniques when applied to realistic problems.

2.2.7 Vortex Method

Sergent [139] introduced the vortex method as an attempt to model the vortical structures present in turbulent flows using a plane of two-dimensional vortices normal to the flow. The method begins by creating a random distribution of two-dimensional vortices over the inlet plane. The vorticity at each point on the inlet plane is equal to the summation of the vorticity contributions of each vortex. Using the Biot-Savart law, the in-plane fluctuating velocity components can be solved from the total vorticity at each point.

Each vortex is characterized by a circulation, a shape function, a length scale, a time scale, and a random rotation. The circulation is related to the local turbulent kinetic energy,
Γ ∝ \sqrt{k}, and Benhamadouche et al. [14] suggested a modified Gaussian form for the shape function, while Martha et al. [89] further tuned the shape functions to include counter-rotating vortex pairs as an attempt to model the hairpin vortices found in the boundary layer of the splitter plate at the inlet of a planar mixing layer. In the original formulation by Sergent [139], the vortex length scale was arbitrarily assigned, but Mathey et al. [92] suggested using a turbulent mixing length hypothesis to determine the vortex size, \( \sigma \propto \frac{k^{3/2}}{\epsilon} \). The time scale \( \tau = \frac{k}{\epsilon} \) represents the lifespan of the vortex before it is destroyed and a new vortex is randomly created. To introduce fluctuations in time, the vortices randomly walk around the inflow plane.

So far, velocity fluctuations are only introduced on planes normal to streamwise direction. Sergent [139] generated streamwise velocity fluctuations by solving a Langevin equation with the assumption that all three fluctuating velocity components satisfied the equation. The Langevin solution for the streamwise fluctuations provided spatial and temporal correlations to already correlated normal fluctuating velocity planes. Benhamadouche et al. [14] used a simplified form that only assumed the streamwise fluctuations satisfied a Langevin equation. As a further simplification, Mathey et al. [92] used a linear kinematic model to generate streamwise velocity fluctuations.

Benhamadouche et al. [14], Mathey et al. [92], and Penttinen and Nilsson [117] all showed a significant improvement over the random fluctuation method of Lund et al. [83] and favorable comparisons with DNS data over a range of test cases. Noticeable development lengths were still present; this is an artifact of the nonphysical method of generating streamwise fluctuations. The vortex method can be equally applied to structured or...
unstructured grids, but no direct control of prescribed Reynolds stress tensors has been implemented.

2.2.8 Random Turbulent Spot Method

The random turbulent spot method, first proposed by Kornev and Hassel [69], was developed as a more physically intuitive model of turbulence than the digital filtering based methods. The central idea is that the coherent structures in a turbulent flow can be modeled as a random field of turbulent spots. A component of the velocity fluctuations imposed by the spots is defined as the sum of a product of three unknown normalized shape functions.

$$u'_i(x, y, z, t) = \sum_{n=1}^{N} \prod_{k=1}^{3} f_{u'_i}^{w_k}(x_k, (x_k)_n, (\rho_k)_n) \text{sign}(d_i - 0.5)$$

(2.14)

The random center of a turbulent spot is $(x_k)_n$, the size of the spot is $(\rho_k)_n$, and $d_i$ is a random number between zero and one. Each of the nine shape functions are solved for using two-point autocorrelation functions; Kornev and Hassel [69] noted the impracticality of using autocorrelation functions for general turbulent flows and offered a simplified model based on the integral length scales. After being transformed using the Cholesky decomposition of the Reynolds stress tensor, the turbulent spots are convected with the mean flow using Taylor’s hypothesis.

Kornev and Hassel [70] updated the random spot method to be divergence-free by defining the shape functions using a vector potential. The resultant turbulent spot from their formulation, considering simplifying assumptions, took the form of a dipole vortex with an amplitude based on a prescribed energy spectrum. The difference argued by the authors between this method and a typical vortex method is that the random spot method...
reproduces an energy spectrum automatically. In an LES study of decaying isotropic turbulence, Kornev et al. [71] found little difference between the development lengths for the non-divergence-free and the divergence-free formulations.

The random turbulent spot method can generate anisotropic fluctuations based on prescribed Reynolds stress tensors, length scales, and autocorrelation functions without any grid restrictions. When the divergence-free formulation is considered, a prescribed energy spectrum can also be reproduced. This method shares the same flaw as other methods formulated using autocorrelation functions: how are these functions specified for general problems when they are not always readily known? Kornev et al. [71] showed that the quality of the synthetic turbulence was strongly affected by the quality of autocorrelation functions used. While the random turbulent spot method takes a more physical approach to modeling the turbulent flow, it still requires a long development time (Kornev et al. [71]).

2.2.9 Synthetic Eddy Method

The synthetic eddy method extends the idea of the vortex method into three dimensions. Whereas the vortex method relied upon a separate means to generate streamwise fluctuations, the synthetic eddy method aims to generate fluctuations in all three dimensions using three-dimensional synthetic coherent structures, or synthetic eddies. The method was formalized for inflow generation by Jarrin et al. [47] and then later applied to RANS-LES coupling by Jarrin et al. [48].
The imposed fluctuating velocity is generated by the combination of a uniform distribution of synthetic eddies.

\[
u'_{i}(x, y, z, t) = \sum_{k=1}^{N} \tilde{u}^{k}_{i}(x, y, z, t)
\]  \hspace{1cm} (2.15)

\(N\) is the total number of eddies and the contribution to the overall fluctuating velocity by a single eddy is \(\tilde{u}^{k}_{i}\). The eddies are convected through the inlet by the mean flow using Taylor’s Hypothesis and once an eddies passes completely through the inlet, a new eddy is randomly created. Each eddy is characterized by a center, \((x^{k}, y^{k}, z^{k})\), a length scale, \(\sigma\), a shape function, \(f_{\sigma}\), and a random orientation, \(\epsilon_{i} = \pm 1\).

\[
\tilde{u}^{k}_{i}(x, y, z, t) = \frac{\epsilon_{i}}{\sqrt{N}} f_{\sigma} \left( \frac{x - x^{k}(t)}{\sigma}, \frac{y - y^{k}(t)}{\sigma}, \frac{z - z^{k}(t)}{\sigma} \right)
\]  \hspace{1cm} (2.16)

The \(1/\sqrt{N}\) factor is included to ensure that the overall fluctuating velocity components satisfy the \(\left<u'_{i}u'_{i}\right> = 1\) condition. The shape function is defined to have compact support and to satisfy the normalization condition. Jarrin et al. [47] suggested that the shape function could be calculated from a two-point autocorrelation function, if complete knowledge of one existed for the inflow being imposed, otherwise the shapes of the eddies should be chosen a priori. In Jarrin et al. [47], the product of three one-dimensional Gaussian functions is chosen, while Jarrin et al. [48] uses the product of three one-dimensional tent functions. Before being combined with the mean velocity profiles, the fluctuating velocities are transformed using the Cholesky decomposition of the Reynolds stress tensor;

\[
u_{i} = \langle u_{i} \rangle + a_{ij}u'_{j}
\]  \hspace{1cm} (2.17)

thus ensuring that a given Reynolds stress tensor is reproduced. Jarrin et al. [48] was able to show a significant reduction in development length compared to the Fourier method of...
Batten et al. [10] for channel and duct flows, but Abboud et al. [1] showed only minor improvements over the digital filtering method of Klein et al. [66] for coaxial jets.

In the LES study of plane channel flow, Jarrin et al. [47] observed that the size, shape, and distribution of eddies near the wall had a significant affect on the production of velocity fluctuations normal to the wall, which increased the development time of realistic turbulence. For wall-bounded flows, the dominant coherent structures that are seen vary with the normal distance from the wall [128] and Pamiès et al. [113] looked to better model that behavior. The inlet plane was separated into different regions moving away from the wall, where the shape function in each region was tuned to match the dominant structures. For example, in the region closest to the wall, the shape functions were inclined and stretched in the streamwise direction, according to the observations seen by Jeong et al. [49], in order to model the elongated turbulent structures growing at an angle from the wall. Significant improvements over the original synthetic eddy method were seen in the LES study of a flat plate turbulent boundary layer. Roidl et al. [129] improved the shape functions in the log layer and the outer layer, while still using the shape functions tuned by Pamiès et al. [113] in the near-wall region; and was able to show a notable improvement in the matching of the near-wall velocity profile and a more realistic turbulent production near the wall. Skillen et al. [148] replaced the $\frac{1}{\sqrt{\lambda}}$ term in the definition of the velocity contribution by a single eddy with a scale factor based on the local eddy population density. This accounts for inhomogeneous eddy distributions on the inlet plane, i.e. the clustering of small eddies near the wall with larger eddies farther into the freestream in boundary
layer flows. Better matching of the Reynolds stresses close to the wall was shown when compared to the original synthetic eddy method.

Motivated by the fact that the previous synthetic eddy methods are not able to directly reproduce any arbitrary energy spectra, Luo et al. [85] proposed the Multi-Scale Synthetic Eddy Method. This method generates anisotropic fluctuations using Gaussian synthetic eddies created over a discrete range of sizes based on given one-dimensional energy spectra and integral length scales. The given spectra are split into discrete wavenumber ranges each containing a portion of the total energy. The amplitudes of the eddies created are then scaled proportionally to the energy contained in their respective wavenumber range. For the LES of a turbulent boundary layer, Luo et al. [85] was able to match the given length scales and energy spectra at the inlet and showed improvement in the downstream development of the length scales and spectra as compared with Jarrin et al. [47]. Luo et al. [86] then used the Multi-Scale Synthetic Eddy Method to generate turbulent inflow for the LES of a turbulent boundary layer flow over a “high-rise building”. Comparing again with the synthetic eddy method of Jarrin et al. [47], Luo et al. [86] was able to show better agreement with reference results generated using the recycling/rescaling method of Lund et al. [83] for the pressure distribution and wind load characteristics on the building.

There has been significant work in the area of divergence-free synthetic eddy methods, with the key aspect of creating a divergence-free method being the imposition of an eddy that satisfies the divergence-free condition. Poletto et al. [124] applied the synthetic eddy framework to the vorticity field and solved for the fluctuating velocity components using the Biot-Savart law. From this solution process, restrictions on the shape functions were
found that were then used to select a function that satisfied the divergence-free condition. This formulation only includes one shape function, based on a sine function, and one characteristic eddy length scale, and as such, could only match a limited range of anisotropic Reynolds stress tensors. To be able to reproduce a wide range of anisotropic turbulence, Poletto et al. [125] reformulated the method to allow for separate shape functions and lengths scales for each dimension. This destroyed the satisfaction of the divergence-free condition guaranteed by the previous formulation, but by taking the divergence of the prospective eddy velocity field sufficient conditions were found to be able to select divergence-free shape functions, based on a quadratic polynomial. Poletto et al. [125] showed that pressure fluctuations at the inlet, caused by the divergence-free condition not being satisfied, were greatly reduced when compared with the original synthetic eddy method for plane channel flow. They also reported much quicker recovery of the friction coefficient and imposed Reynolds stress profiles than the original synthetic eddy method, the vortex method, and the source method of Davidson and Billson [29].

Sescu and Hixon [140] approached the development of a divergence-free method from a computational aeroacoustics direction. To account for the extremely low tolerance for spurious acoustic waves, the formulation needed to take more of the physics into account. The synthetic eddy is defined such that it satisfies the Euler equations linearized around the mean flow, ensuring that each eddy is convected at the correct velocity through the inlet; Lee et al. [77] proved that Taylor’s Hypothesis is not a good approximation for purely compressible motion. Then, to enforce the divergence-free condition, the fluctuating velocity field is represented by a vector potential. The Gaussian function, Mexican hat wavelet, and
Morlet wavelet were chosen as shape functions due to them having more realistic acoustic profiles. Plots of acoustic pressure and divergence of velocity in Sescu and Hixon [140] and Sescu and Hixon [141] for decaying homogeneous isotropic turbulence show a lack of spurious waves introduced at the inlet. In the LES study by Sescu and Hixon [141] on decaying homogeneous isotropic turbulence, their divergence-free method was able to reproduce the experimental spatial rate of turbulent kinetic energy decay, while the original synthetic eddy method showed a long development time before the decay rate was reached.

Kim and Haeri [61] further developed the method to include a scheme that optimizes the size, strength, distribution, and probability of shape function type of the synthetic eddies in order to match the von Kármán spectra for homogeneous isotropic turbulence. Since the synthetic eddies were defined to satisfy the linearized Euler equations, spurious waves could be created when they are injected into a domain that solves the non-linear Euler equations. To account for this, Kim and Haeri [61] convected the synthetic eddies through a sponge layer at the inlet. Results from airfoil-turbulence interaction simulations showed that the spurious noise measured was around four orders of magnitude less than physical mechanism of noise generation.

The synthetic eddy method can be applied to any type of discretization and complex geometry; Pavlidis et al. [116] conducted an LES study of an atmospheric flow over a bluff body using an unstructured tetrahedral mesh and were able to show favorable comparisons to reference data. Given realistic eddy shapes and quality estimates of length scales, the synthetic eddy method has the ability to closely model physical turbulence. That close
modeling results in the development of synthetic turbulence into realistic turbulence over a very short distance, thus allowing for shorter computational grids and reduced costs.

### 2.2.10 Source Methods

The source methods mainly differ from the other methods reviewed by how the synthetic fluctuations are imposed in the domain. For the other synthetic methods, the generated turbulence is imposed at the inlet or at an interface through the boundary conditions. Whereas for a source method, the velocity fluctuations are introduced through a source term in the momentum equations over a plane or region in the domain.

Billson et al. [16] proposed a source method used for jet noise prediction that generated synthetic isotropic turbulence using a Fourier method consistent with the methods of Béchara et al.,[12] and Bailly and Juvé,[8], but with an improved mechanism for temporal correlation. Instead of only filtering the synthetic velocity after it has been created[12] or only including a constant convective term [8], the generated field of random Fourier modes is filtered with the solution of the convection equation for the convection of the previous time steps’s filtered synthetic velocity. The solution of the convection equation provides a more physically consistent mechanism to correlate in time. This method is only divergence-free for homogeneous isotropic turbulence, but by using the same scaling and orthogonal transformation procedure in Smirnov et al. [150], Billson et al. [17] extended the method to reproduce divergence-free homogeneous anisotropic turbulence. When applied to a hybrid RANS-LES study of turbulent channel flow, Davidson and Billson [29] found that the source method quickly transitioned to realistic turbulence over a short de-
development length, but also generated an overproduction of turbulent kinetic energy. It was shown that the overproduction could be reduced by increasing the imposed streamwise turbulent length scale. Using a different method to generate synthetic fluctuations, Schmidt and Breuer [135] implemented a source method using the digital filtering method of Klein et al. [66] for a hybrid URANS-LES study of the laminar separation bubble over an airfoil.

Source methods allow for the inject of synthetic turbulence at any location within a domain, not just at a boundary, which gives increased flexibility in the types of flows that are able to be simulated. Whatever advantages or disadvantages exist for the method used to generate the synthetic turbulence will still be present when the fluctuations are injected into the domain using a source method.

2.3 Controlled Forcing Methods

At first glance, the controlled forcing method appears similar to the source method because they both act through an additional term added to one or all of the momentum equations, be it a body force for the controlled forcing method or a source term for the source method. But, there is a clear fundamental difference between the two methods: the controlled forcing method, by itself, does not generate additional fluctuations, it only acts on fluctuations that already exist within the flow. So, for a fully-laminar flat plate boundary layer flow with no outside disturbances, the controlled forcing method cannot cause the boundary layer to transition into becoming turbulent. If fluctuations are already present, for example, through the application of a synthetic turbulence method; the body
forces within the controlled forcing method can be defined to accelerate the transition from synthetic to realistic turbulence.

![Diagram showing the controlled forcing method.](image)

Spille-Kohoff and Kaltenbach [155] first proposed a method in the context of wall-bounded walls. After observing that the wall-normal Reynolds stress dominantly affected the production of the wall-normal Reynolds shear stress, a body force acting on a plane normal to the streamwise direction was added to the vertical momentum equation. The force is defined as the product between a magnitude and the streamwise velocity fluctuations at the forcing plane, where the magnitude of the body force is adjusted using a proportional-integral (PI) controller based on the error between the target and calculated Reynolds shear stress profiles. The intent of the body force is to amplify or dampen local flow events that contribute to the overall Reynolds shear stress [56]. Since the velocity fluctuations are forced directly, thresholds for when the body force is applied are set in order to prevent unrealistic flow events. Multiple forces can be added to the vertical momentum equation, corresponding to multiple forcing planes in the streamwise direction. Keating et al. [56]

Distribution Statement A. Approved for public release; distribution is unlimited.
compared the Fourier method of Batten et al. [10] with and without controlled forcing planes near the inlet for turbulent channel flow. It was observed that the controlled forcing method corrected the intrinsically low Reynolds shear stress production present in the Batten et al. [10] method causing the rapid growth of realistic turbulent structures. Recovery of the friction coefficient and Reynolds shear stress profile also occurred at half the distance of the Batten et al. [10] method without control forcing. Keating et al. [57] further confirmed the ability of the Spille-Kohoff and Kaltenbach [155] controlled forcing method to accelerate the transition from synthetic to realistic turbulence in a hybrid RANS-LES study of turbulent boundary layers over various pressure gradients. Laraufie et al. [73] modified the method so that the PI controller was now based on the error between the target and calculated wall-normal Reynolds stresses, instead of the Reynolds shear stress. Comparisons between the two forcing methods were then carried out using hybrid RANS-LES calculations of a flat plate turbulent boundary layer. The tuned synthetic eddy method of Pamiès et al. [113] was used to generate synthetic fluctuations upstream of the forcing planes. While the original forcing method only shortened the development length a small amount as compared to using no forcing at all, it was found that the updated definition of the forcing magnitude reduced the transition from synthetic to realistic turbulence by around two-thirds.

Lundgren [84] proposed the linear forcing method to force the formation of stationary isotropic turbulence. A force, operating on its corresponding fluctuating velocity component at each grid point in the domain, is added to each of the three momentum equations. Since the desired flow is supposed to be isotropic, the forcing needs to be applied isotrop-
ically. The amplitude of each force corresponds to the negative of the turbulent kinetic energy production term for its respective direction. By not letting the energy grow or decay, the turbulence is kept stationary. In tests by Rosales and Meneveau [131], it was shown that stationary isotropic turbulence could be achieved. de Laage de Meux et al. [31] proposed an extended formulation of the linear forcing method in order to force stationary anisotropic turbulence. In the new formulation the amplitude of each of the forces is related to the product of a matrix, based on the error between the target and calculated Reynolds stress tensors, and the vector of fluctuating velocity components. Each component of the force vector is added to its respective momentum equation and applied over the whole domain. The method was able to reproduce the stationary isotropic turbulence results of Rosales and Meneveau [131] and stationary anisotropic turbulence. Preliminary results were also shown coupling the anisotropic linear forcing method, applied over a region at the inlet, with the original synthetic eddy method of Jarrin et al. [47] for a hybrid RANS-LES turbulent channel flow validation case.

For flows that have varying mean velocity profiles, Schlüter et al. [137] proposed the addition of a body force to appropriate momentum equation to match a prescribed velocity profile. The force is a proportional controller based on the error between the given and calculated mean velocity profiles. This method was developed for use at the outflow of an LES domain that feeds into a RANS domain, but it would also be applicable elsewhere in the domain.

The controlled forcing has been shown to rapidly accelerate the development of realistic turbulence from a synthetic inflow. Most of the methods are relatively simple to
implement and have the potential to greatly reduce the length of domain required, thus reducing computational cost.
CHAPTER 3
TRIPLE HILL’S VORTEX SYNTHETIC EDDY METHOD

The proposed synthetic eddy method exists within the general domain of the original method proposed by Jarrin et al. [47] and of the methods that followed. Turbulent fluctuations are modeled using a combination of individual synthetic eddies which are convected by a mean flow. The amplitudes of these synthetic eddies are controlled in order to match desired Reynolds stresses. The key difference between the Triple Hill’s Vortex Synthetic Eddy Method (THV SEM) and the existing methods is in the definition of the synthetic eddy. The individual Triple Hill’s Vortex (THV) was first created as a more realistic, divergence-free synthetic eddy. The THV SEM was then developed around the THV adapting the overall philosophy of the synthetic eddy methods while preserving the divergence-free nature of the THV.

Beginning with an overview of the governing equations and numerical algorithm, the Triple Hill’s Vortex Synthetic Eddy Method is presented. Following results shown from the simulation of a single Triple Hill’s Vortex, spatially decaying homogeneous isotropic turbulence, turbulent channel flow, and a turbulent mixing layer; conclusions and a direction for future work are outlined.
3.1 Governing Equations

The governing equations consist of the Favre-filtered full compressible Navier-Stokes equations written in curvilinear coordinates and conservative form. A generalized curvilinear coordinate transformation in the three-dimensional form \( \xi = \xi(x, y, z) \), \( \eta = \eta(x, y, z) \), \( \zeta = \zeta(x, y, z) \), is considered, where \( \xi \), \( \eta \), and \( \zeta \) are the spatial coordinates in the computational space, and \( x \), \( y \), and \( z \) are the spatial coordinates in physical space. As a note, a tilde (\( \tilde{\cdot} \)) represents Favre-filtering at the grid level \( \Delta \). In conservative form, the Navier-Stokes equations are written as

\[
Q_t + F_\xi + G_\eta + H_\zeta = S \tag{3.1}
\]

where the vector of conservative variables is given by

\[
Q = \frac{1}{J} \left\{ \bar{\rho}, \quad \bar{p}\tilde{u}_i, \quad \bar{p}\tilde{E} \right\}^T, \quad i = 1, 2, 3 \tag{3.2}
\]

\( \bar{\rho} \) is the mean density of the fluid, \( \tilde{u}_i = (\tilde{u}, \tilde{v}, \tilde{w}) \) is the filtered velocity vector in physical space, and \( \bar{p}\tilde{E} \) is the total energy. The flux vectors, \( F \), \( G \), and \( H \), are given by

\[
F = \frac{1}{J} \left\{ \bar{p}U, \quad \bar{p}\tilde{u}_i U + \xi_{x_i}(\bar{p} + \tau_{i1}), \quad \bar{p}\tilde{E}U + \bar{p}U + \xi_{x_i}\Theta_i \right\}^T, \tag{3.3}
\]

\[
G = \frac{1}{J} \left\{ \bar{p}V, \quad \bar{p}\tilde{u}_i V + \eta_{x_i}(\bar{p} + \tau_{i2}), \quad \bar{p}\tilde{E}V + \bar{p}V + \eta_{x_i}\Theta_i \right\}^T, \tag{3.4}
\]

\[
H = \frac{1}{J} \left\{ \bar{p}W, \quad \bar{p}\tilde{u}_i W + \zeta_{x_i}(\bar{p} + \tau_{i3}), \quad \bar{p}\tilde{E}W + \bar{p}W + \zeta_{x_i}\Theta_i \right\}^T \tag{3.5}
\]

where the contravariant velocity components are given by
\[ U = \xi x_i \tilde{u}_i, \quad V = \eta x_i \tilde{u}_i, \quad W = \zeta x_i \tilde{u}_i \]  

with the Einstein summation convention applied over \( i = 1, 2, 3 \). The shear stress tensor and the heat flux are given as

\[ \tau_{ij} = \frac{\tilde{\mu}}{Re} \left[ \left( \frac{\partial \xi_k}{\partial x_j} \frac{\partial \tilde{u}_i}{\partial x_k} + \frac{\partial \xi_k}{\partial x_i} \frac{\partial \tilde{u}_j}{\partial x_k} \right) - \frac{2}{3} \delta_{ij} \frac{\partial \xi_i}{\partial x_k} \frac{\partial \tilde{u}_k}{\partial x_l} \right] + \tau^{sgs}_{ij} \]  

\[ \Theta_i = \tilde{u}_j \tau_{ij} + \frac{\tilde{\mu}}{(\gamma - 1) M_\infty^2 Re Pr} \frac{\partial \xi_i}{\partial x_i} \frac{\partial \tilde{T}}{\partial \xi_i} + q^{sgs}_i \]  

respectively. \( \tau^{sgs}_{ij} \) and \( q^{sgs}_i \) are the subgrid scale stress and subgrid scale heat flux terms, which are defined in Section 3.1.1. The pressure \( \tilde{p} \), the temperature \( \tilde{T} \), and the density of the fluid are combined in the equation of state, \( \tilde{p} = \tilde{p} \tilde{T} / \gamma M_\infty^2 \). Other notations include the dynamic viscosity \( \mu \), Reynold’s number \( Re = \rho_\infty V_\infty L / \mu \) based on a characteristic velocity \( V_\infty \), and a characteristic length \( L \), the free-stream Mach number \( M_\infty = V_\infty / a \) (with \( a \) being the speed of sound), Prandtl’s number \( Pr = C_p \mu / k \) (where \( k \) is thermal conductivity), the specific heat at constant pressure \( C_p \), and the ratio between the specific heats \( \gamma \). The Jacobian of the curvilinear transformation from the physical space to computational space is denoted by \( J \). The derivatives \( \xi, \xi_y, \xi_z, \eta, \eta_y, \eta_z, \zeta, \zeta_y, \zeta_z \) and \( \xi_z \) represent grid metrics. The variables are non-dimensionalized by their respective freestream variables, except for the pressure which is non-dimensionalized by \( \rho_\infty V_\infty^2 \). The thermal conductivity, \( k \), is obtained from the Prandtl number and the dynamic viscosity is linked to the temperature using the Sutherland’s equations in dimensionless form.
\[ \tilde{\mu} = \tilde{T}^{3/2} \frac{1 + C_1/T_\infty}{T + C_1/T_\infty} \]  \tag{3.9} 

where for air at sea level, \( C_1 = 110.4 K \) and \( T_\infty \) is a reference temperature.

### 3.1.1 Subgrid Scale Model

The SGS stress is modeled using the Coherent Structure Model (CSM) developed by Kobayashi [67] for incompressible flow and later extended to compressible flows by Hadjadj et al. [41] and Ben-Nasr et al. [15]. The CSM is based on the assumption that the SGS dissipation is small at the center of a coherent eddy and that the energy transfer between the resolved scales and the SGS occurs around the edge of this coherent eddy [67, 68]. The model parameter is dynamically calculated based on a function of the local velocity gradients. Unlike the traditional Dynamic Smagorinsky Model of Germano et al. [39] and Lilly [79], the CSM does not require test filtering or averaging. Kobayashi [67], Kobayashi et al. [68], and Onodera et al. [112] all showed for various incompressible flow cases that the CSM performed just as well as the Dynamic Smagorinsky Model. For a compressible turbulent boundary layer, Ben-Nasr et al. [15] was able to show results that were equivalent to the Dynamic Smagorinsky Model and the Wall-Adapting Local Eddy-viscosity Model of Nicoud and Ducros [110].

The SGS stress tensor, \( \tau_{ij}^{sgs} = \overline{\rho} (\tilde{u}_i\tilde{u}_j - \tilde{u}_i\tilde{u}_j) \), is defined using an eddy-viscosity model as follows

\[ \tau_{ij}^{sgs} = -2\mu_{sgs} \left( \tilde{S}_{ij} - \frac{1}{3} \tilde{S}_{kk} \delta_{ij} \right) - \frac{1}{3} \tau_{kk}^{sgs} \delta_{ij} \]  \tag{3.10}
where $\tilde{S}_{ij}$ is the resolved strain-rate tensor. The SGS viscosity, $\mu_{sgs}$, is defined as

$$
\mu_{sgs} = \tilde{\rho} C_s \Delta^2 |\tilde{S}| \tag{3.11}
$$

where $\Delta$ is the grid scale, $C_s$ is the dynamically calculated Smagorinsky coefficient, and $|\tilde{S}|$ is the strain-rate magnitude. The isotropic part of the SGS tensor, $\tau_{kk}^{sgs}$, is modeled using the following relationship proposed by Yoshisawa [171]

$$
\tau_{kk}^{sgs} = 2\tilde{\rho} C_I \Delta^2 |\tilde{S}|^2 \tag{3.12}
$$

where $C_I$ is another dynamically calculated model coefficient.

For the CSM, the Smagorinsky coefficient is defined as

$$
C_s = C_{csm} |F_{cs}|^{3/2} (1 - F_{cs}) \tag{3.13}
$$

where $C_{csm}$ is the CSM model coefficient, taking values to between $1/30$ [112, 41, 15] and $1/22$ [67, 68], and $F_{cs}$ is the coherent structure function.

$$
F_{cs} = \frac{\tilde{Q}}{E} \tag{3.14}
$$

$\tilde{Q}$ is the second invariant of the resolved velocity gradient and $\tilde{E}$ is the magnitude of the resolved velocity gradient tensor.

$$
\tilde{Q} = \frac{1}{2} \left( \tilde{W}_{ij} \tilde{W}_{ij} - \tilde{S}_{ij} \tilde{S}_{ij} \right) = -\frac{1}{2} \frac{\partial \tilde{u}_j}{\partial x_i} \frac{\partial \tilde{u}_i}{\partial x_j} \tag{3.15}
$$

$$
\tilde{E} = \frac{1}{2} \left( \tilde{W}_{ij} \tilde{W}_{ij} + \tilde{S}_{ij} \tilde{S}_{ij} \right) = \frac{1}{2} \frac{\partial \tilde{u}_j}{\partial x_i} \frac{\partial \tilde{u}_i}{\partial x_j} \tag{3.16}
$$

$\tilde{S}_{ij}$ is the resolved strain-rate tensor and $\tilde{W}_{ij}$ is the resolved vorticity tensor.

$$
\tilde{S}_{ij} = \frac{1}{2} \left( \frac{\partial \tilde{u}_j}{\partial x_i} + \frac{\partial \tilde{u}_i}{\partial x_j} \right) \tag{3.17}
$$

$$
\tilde{W}_{ij} = \frac{1}{2} \left( \frac{\partial \tilde{u}_j}{\partial x_i} - \frac{\partial \tilde{u}_i}{\partial x_j} \right) \tag{3.18}
$$

Distribution Statement A. Approved for public release; distribution is unlimited.
While Kobayashi [67] only proposed the CSM for incompressible flow and Hadjadj et al. [41] and Ben-Nasr et al. [15] chose not to the model the isotropic part of the SGS stress tensor, this work dynamically models the coefficient for isotropic part of the SGS tensor, \(C_I\), in a similar fashion to the Smagorinsky coefficient.

\[
C_I = C_{csm} |F_{cs}|^{3/2} (1 - F_{cs}) \quad (3.19)
\]

\(C_{csm}\) is the CSM isotropic model coefficient and is chosen here such that the maximum value of \(C_I\) is between 0.005 [103] and 0.0066 [173].

It should be noted that the value of the coherent structure function \(F_{cs}\) is bounded between -1 and 1, which results in the model coefficients \(C_s\) and \(C_I\) to also be bounded.

\[
0 \leq C_s \leq C_{csm} \quad (3.20)
\]

\[
0 \leq C_I \leq C_{csm} \quad (3.21)
\]

The SGS heat flux term, \(q_{i}^{sgs}\), is also modeled using an eddy-viscosity model.

\[
q_{i}^{sgs} = -\frac{\mu_{sgs}}{(\gamma - 1) M_{\infty}^2 Re Pr_{sgs}} \frac{\partial \bar{T}}{\partial x_i} \quad (3.22)
\]

The SGS Prandtl number, \(Pr_{sgs}\), is given the constant value of 0.7 [173, 90].

A validation of this implementation of the Coherent Structure Model is shown in Appendix A.

### 3.2 Numerical Algorithm

The compressible Navier-Stokes equations are solved in the framework of Large Eddy Simulations, where the Coherent Structure Model is applied to account for the missing
sub-grid scale energy. The numerical algorithm uses high-order finite difference approximations for the spatial derivatives and explicit time marching.

### 3.2.1 Spatial Discretization

The spatial derivatives are discretized using dispersion-relation-preserving schemes of Tam and Webb [159] or a high-resolution 9-point dispersion-relation-preserving optimized scheme of Bogey et al. [18]. The first derivative at the \( l \)th node is approximated using \( M \) values of \( f \) to the right and \( N \) values of \( f \) to left of the node.

\[
\left( \frac{\partial f}{\partial x} \right)_l \approx \frac{1}{\Delta x} \sum_{j=-N}^{M} a_j f_{l+j} 
\]  

By taking the Fourier transform of the above equation, the coefficients \( a_j \) are found by minimizing the integrated error of the difference between the wavenumber of the finite difference scheme and the wavenumber of the Fourier transform of the finite difference scheme. The coefficients \( a_j \) are given in Table 3.1.

<table>
<thead>
<tr>
<th>Stencil</th>
<th>( a_1 = -a_{-1} )</th>
<th>( a_2 = -a_{-2} )</th>
<th>( a_3 = -a_{-3} )</th>
<th>( a_4 = -a_{-4} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>DRP</td>
<td>0.77088238</td>
<td>-0.16670590</td>
<td>0.02084314</td>
<td>0</td>
</tr>
<tr>
<td>FD9p</td>
<td>0.84157012</td>
<td>-0.24467863</td>
<td>0.05946358</td>
<td>-0.00765090</td>
</tr>
</tbody>
</table>

Table 3.1

Weights of the centered stencils

Distribution Statement A. Approved for public release; distribution is unlimited.
3.2.2 Time Marching

The time integration is either performed using a third-order TVD Runge-Kutta method [143] written in the form

\[
\begin{align*}
Q^{(0)} &= Q^n \\
Q^{(1)} &= Q^{(0)} + \Delta t L(Q^{(0)}) \\
Q^{(2)} &= \frac{3}{4} Q^{(0)} + \frac{1}{4} Q^{(1)} + \frac{1}{4} \Delta t L(Q^{(1)}) \\
Q^{n+1} &= \frac{1}{3} Q^{(0)} + \frac{2}{3} Q^{(1)} + \frac{2}{3} \Delta t L(Q^{(2)}) ,
\end{align*}
\] (3.24)

or a fully-explicit second-order Adams-Bashforth scheme [20] of the form

\[
Q^{n+1} = Q^n + \Delta t \left[ \frac{3}{2} L(Q^n) - \frac{1}{2} L(Q^{n-1}) \right] 
\] (3.25)

where \( L(Q) \) is the residual and \( n \) is the current time level.

3.2.3 Spatial Filtering

To damp out the unwanted high wavenumber waves from the solution, high-order spatial filters, as developed by Kennedy and Carpenter [60], are used. Consider the eighth order accurate explicit central-difference operator for the \((2n)\)th-order derivative of a function \( f \)

\[
f_i^{(2n)} = \gamma f_i + a \frac{f_{i+1} - f_{i-1}}{(\Delta x)^2} + b \frac{f_{i+2} - f_{i-2}}{(\Delta x)^2} + c \frac{f_{i+3} - f_{i-3}}{(\Delta x)^2} + d \frac{f_{i+4} - f_{i-4}}{(\Delta x)^2} 
\] (3.26)

The filter function is applied a vector \( u \) as follows

\[
\hat{u} = (1 + \alpha_D D) u
\] (3.27)
where \( \hat{u} \) is the filtered vector, \( \alpha_D = (-1)^{n+1}2^{-2n} \) for a \((2n)\)th-order filter, and \( D \) is a filter matrix whose values are related to the coefficients of the central-difference operator. The eigenvalues of \( D \) are negative which ensures that the filter is completely dissipative. For eighth order accurate filters, \( \gamma = -70, a = 56, b = 28, c = 8, \) and \( d = -1. \)

### 3.2.4 Boundary Conditions

The Triple Hill’s Vortex Synthetic Eddy Method (THV SEM) was applied at the inlet for all cases. No slip boundary condition for velocity and adiabatic condition for temperature are imposed wherever a wall is present and periodic boundary conditions are utilized in homogeneous directions.

Standard outflow boundary conditions are applied at the outlet along with a region of artificially increased viscosity immediately upstream of the outlet to damp spurious waves.

The dynamic viscosity, \( \mu_m \), in the exit region is smoothly increased in the streamwise direction by an artificial viscosity, \( \mu_{BC} \), and a weighting function, \( w_\mu \).

\[
\mu(x) = \mu_m + w_\mu(x) \mu_{BC}
\]  

(3.28)

The weighting function, \( w_\mu \), is defined as

\[
w_\mu(x) = \begin{cases} 
0 & ; x \leq x_s \\
 \max \left[ 5 \left( \frac{x-x_s}{x_e-x_s} \right)^4 - 4 \left( \frac{x-x_s}{x_e-x_s} \right)^5 \right] & ; x_s \leq x \leq x_e \\
 \max & ; x_e < x \leq L_x 
\end{cases}
\]

(3.29)

where \( x \) is the streamwise coordinate, \( x_s \) is the location of the start of the increased viscosity region, \( x_e \) is the location of the end of the transition region, \( L_x \) is the location of the...
outlet, and $w_{max}$ is maximum amplitude of the artificial viscosity. Walchshofer et al. [164] proposed defining the artificial viscosity, $\mu_{BC}$, using a Smagorinsky type eddy-viscosity model as an improvement over the constant definition of $\mu_{BC}$ proposed by Liu and Liu [81].

$$\mu_{BC} = \rho C_s \Delta^2 \| \tilde{S} \| = \mu_{sgs}$$

(3.30)

$\mu_{sgs}$ is the SGS viscosity calculated by the SGS model in Section 3.1.1. By locally defining the artificial viscosity based on the instantaneous flow field, the increased viscous dissipation is applied only to the areas where it is needed and spurious reflections off of the increased viscosity region are greatly reduced.

### 3.3 Proposed Synthetic Eddy Method

Hill’s spherical vortex (Hill [44]) represents one of the best-known examples of a steady rotational solution to the classical Euler equations, modeling an inviscid incompressible flow. It is characterized by one amplitude, and by one degree of freedom associated with the translation along its axis, which is a fundamental property of the Hill’s spherical vortex as a consequence of its definition in relation to a uniform flow. Thus, a synthetic turbulence model that is based on a combination of eddies representing single Hill’s vortices would only have the freedom to reproduce isotropic turbulence. By considering a new vortex structure, one that is composed of three Hill’s spherical vortices with the axes perpendicular to each other, two more associated amplitudes are introduced and thus, two more degrees of freedom that can be utilized to match a given Reynolds stress tensor are made available.
A synthetic turbulence model consisting of such Triple Hill’s Vortices (THV) has enough freedom to reproduce anisotropic turbulence.

### 3.3.1 Hill’s Spherical Vortex

The base component of the proposed synthetic eddy method is the Hill’s spherical vortex. It is a steady, axisymmetric solution to the Euler equations for an incompressible flow. Derived from the incompressible Euler equations, the Helmholtz equation for vorticity is

\[
\frac{\partial \omega}{\partial t} + (\mathbf{u} \cdot \nabla)\omega = (\omega \cdot \nabla)\mathbf{u}
\]

where \( \omega = (\omega_r, \omega_\theta, \omega_z) \). It can be combined with the continuity equation for an incompressible flow, \( \nabla \cdot \mathbf{u} = 0 \), and through using the definition of a streamfunction for an axisymmetric flow,

\[
\begin{align*}
  u_r(r, z) &= \frac{1}{r} \frac{\partial \psi}{\partial r}, \\
  u_z(r, z) &= -\frac{1}{r} \frac{\partial \psi}{\partial z}
\end{align*}
\]

and the fact that \( \omega_\theta / r \) is constant along a streamline and only depends on the value of the stream function (i.e. \( \partial (\omega_\theta / r) / \partial t = 0 \)),

\[
\frac{\omega_\theta}{r} = f(\psi),
\]

the following equation governing the streamfunction is obtained.

\[
\frac{\partial^2 \psi}{\partial z^2} + \frac{\partial^2 \psi}{\partial r^2} - \frac{1}{r} \frac{\partial \psi}{\partial r} = -r^2 f(\psi)
\]

Considering the assumption that \( \omega_\theta / r = f(\psi) = A = \text{const} \) inside a sphere of radius \( a \), an exact solution to equation (3.34) can be found. This solution represents the Hill’s
spherical vortex [44]. Using the boundary condition at the surface of the sphere, $\psi = 0$, the streamfunction inside the sphere ($r^2 + z^2 < a^2$) is

$$\psi(r, z) = \frac{Ar^2}{10} \left(a^2 - z^2 - r^2\right)$$

(3.35)

The streamfunction outside the sphere ($r^2 + z^2 > a^2$),

$$\psi(r, z) = -\frac{u_0 r^2}{2} \left[1 - a^3 \left(r^2 + z^2\right)^{-3/2}\right],$$

(3.36)

corresponds to the potential flow around a solid sphere of radius $a$ in a uniform flow of speed $u_0$ in the negative $z$-axis direction. By matching the two streamfunction solutions at the surface of the sphere, the constant $A$ can be found as

$$A = \frac{15u_0}{2a}$$

(3.37)

From equation (3.32), the velocities inside the sphere

$$u_r(r, z) = \frac{3}{2} u_0 \frac{zr}{a^2}, \quad u_z(r, z) = \frac{3}{2} u_0 \left(1 - \frac{z^2 + 2r^2}{a^2}\right)$$

(3.38)

and outside the sphere

$$u_r(r, z) = \frac{3}{2} u_0 \frac{zr}{a^2} \left(\frac{a^2}{z^2 + r^2}\right)^{5/2}, \quad u_z(r, z) = u_0 \left[\left(\frac{a^2}{z^2 + r^2}\right)^{5/2} \frac{2z^2 - r^2}{2a^2} - 1\right]$$

(3.39)

can be found.

### 3.3.2 Triple Hill’s Vortex

Instead of viewing the fundamental synthetic eddy as a distinct Hill’s vortex structure, this method considers a more complex flow structure: the Triple Hill’s Vortex (THV). The
THV is a combination of three independent Hill’s vortices that all share the same center \((x_0, y_0, z_0)\) and the same outer radius \(a\). The rotation axis of each Hill’s vortex is orthogonal to the rotation axis of the other two Hill’s vortices. In Cartesian coordinates, the rotation axis of one Hill’s vortex with amplitude \(u_0 = u_0^x\) is oriented in the \(x\)-direction, and has the associated velocity components given as

\[
\begin{align*}
    u^x(x, y, z) &= u_z(r, z) \\
v^x(x, y, z) &= u_r(r, z) \sin(\theta) \tag{3.40} \\
w^x(x, y, z) &= u_r(r, z) \cos(\theta)
\end{align*}
\]

\[
r = \sqrt{y^2 + z^2}, \quad z = x, \quad \theta = \arctan\left(\frac{y}{z}\right)
\]

The rotation axis of another Hill’s vortex with amplitude \(u_0 = u_0^y\) is oriented in the \(y\)-direction, and has the form

\[
\begin{align*}
    u^y(x, y, z) &= u_r(r, z) \cos(\theta) \\
v^y(x, y, z) &= u_z(r, z) \tag{3.41} \\
w^y(x, y, z) &= u_r(r, z) \sin(\theta)
\end{align*}
\]

\[
r = \sqrt{x^2 + z^2}, \quad z = y, \quad \theta = \arctan\left(\frac{z}{x}\right)
\]

The rotation axis of the third Hill’s vortex with amplitude \(u_0 = u_0^z\) is oriented in the \(z\)-direction, and has the form

\[
\begin{align*}
    u^z(x, y, z) &= u_z(r, z) \sin(\theta) \\
v^z(x, y, z) &= u_r(r, z) \cos(\theta) \tag{3.42} \\
w^z(x, y, z) &= u_z(r, z)
\end{align*}
\]

Distribution Statement A. Approved for public release; distribution is unlimited.
\[ r = \sqrt{x^2 + y^2}, \quad z = z, \quad \theta = \arctan \left( \frac{x}{y} \right) \]

In equations (3.40)-(3.42), \( r \) and \( z \) are the local polar coordinates associated with the vortex (the \( z \) coordinate is in the direction of the rotation axis), and \( u_r \) and \( u_z \) are the velocity components in the local polar coordinate system. The velocity components \( u_r \) and \( u_z \) of each of the vortices can be found from equations (3.38) and 3.39). The superscript signifies the orientation direction of the Hill’s vortex rotation axis (for example, \( u^y \) is the \( x \)-component of velocity for the Hill’s vortex with the rotation axis oriented in the \( y \)-direction). Thus, the velocity components for the THV can be obtained by summation according to

\[
\begin{align*}
\tilde{u} &= u^x + u^y + u^z \\
\tilde{v} &= v^x + v^y + v^z \\
\tilde{w} &= w^x + w^y + w^z
\end{align*}
\]

(3.43)

where the tilde (\( \tilde{\cdot} \)) signifies the entire THV.

### 3.3.3 Divergence of a Triple Hill’s Vortex

The proof that the THV is divergence-free is straightforward. Consider the divergence of the velocity field for a single THV.

\[
\nabla \cdot \tilde{u} = \frac{\partial \tilde{u}}{\partial x} + \frac{\partial \tilde{v}}{\partial y} + \frac{\partial \tilde{w}}{\partial z}
\]

\[
= \frac{\partial (u^x + u^y + u^z)}{\partial x} + \frac{\partial (v^x + v^y + v^z)}{\partial y} + \frac{\partial (w^x + w^y + w^z)}{\partial z}
\]

(3.44)

\[
= \left( \frac{\partial u^x}{\partial x} + \frac{\partial v^x}{\partial y} + \frac{\partial w^x}{\partial z} \right) + \left( \frac{\partial u^y}{\partial x} + \frac{\partial v^y}{\partial y} + \frac{\partial w^y}{\partial z} \right) + \left( \frac{\partial u^z}{\partial x} + \frac{\partial v^z}{\partial y} + \frac{\partial w^z}{\partial z} \right)
\]

\[
= (\nabla \cdot u^x) + (\nabla \cdot u^y) + (\nabla \cdot u^z)
\]
where, for example, $u^x$, $v^x$ and $w^x$ are the velocity components in the local coordinate system that is associated with the vortex with its axis aligned with the $x$ axis (of the global coordinate system). Therefore, since each Hill’s vortex satisfies the divergence-free condition in its local coordinate system,

$$\nabla \cdot \mathbf{u} = 0 \quad (3.45)$$

the THV is divergence-free. Appendix B shows the calculation of the divergence of velocity for each Hill’s vortex mapped onto the global coordinate system.

### 3.3.4 Convection of a Triple Hill’s Vortex

The THV’s are convected through the inlet using Taylor’s frozen turbulence hypothesis

$$x' = (x - x_0) - U(t - t_0)$$

$$y' = (y - y_0) - V(t - t_0) \quad (3.46)$$

$$z' = (z - z_0) - W(t - t_0)$$

where $(U,V,W)$ are the mean velocity components and $(x_0,y_0,z_0,t_0)$ are the space and time coordinates of the center of the THV. The divergence of velocity for a single THV is only identically zero when $(U,V,W)$ are constant over the entire THV (see Appendix C for more details). For spatially varying inflow velocity profiles, such as in channel flow, the mean inflow velocity vary continuously across the THV. So, the following assumption was
made: the \((U,V,W)\) for a single THV are calculated at the center of the THV and applied over the entire THV.

\[
x' = (x - x_0) - U(x_0, y_0, z_0)(t - t_0)
\]

\[
y' = (y - y_0) - V(x_0, y_0, z_0)(t - t_0)
\]

\[
z' = (z - z_0) - W(x_0, y_0, z_0)(t - t_0)
\]

(3.47)

3.3.5 Inflow Velocity

The calculation of the THV Synthetic Eddy Method is similar to the method proposed by Jarrin et al. [47], except the matching between the imposed and given Reynolds stress tensors is performed differently, while maintaining the divergence-free condition. The turbulent inflow is viewed as a collection of eddies added to a mean flow. In the following, the velocities in equations (3.40), (3.41), and (3.42) are assumed to be products of a constant amplitude multiplied by a shape function,

\[
\begin{bmatrix}
u^x \\
u^y \end{bmatrix} = \begin{bmatrix}
u_0^x f_u^x \\
u_0^y f_v^x \end{bmatrix}, \quad \begin{bmatrix}
u^y \\
u^z \end{bmatrix} = \begin{bmatrix}
u_0^y f_v^y \\
u_0^z f_w^y \end{bmatrix}, \quad \begin{bmatrix}
u^z \end{bmatrix} = \begin{bmatrix}
u_0^z f_w^z \end{bmatrix}
\]

(3.48)

where, instead of using Gaussian shape functions or other simple functions (as in Jarrin et al. [47], Poletto [124], and Skillen et al. [148]), THV shape functions are utilized (extracted from equations (3.38) and (3.39)). The three amplitudes associated with the THV are the means by which the Reynolds stress tensor components of a prescribed inflow

Distribution Statement A. Approved for public release; distribution is unlimited.
velocity are matched. The inflow velocity components, \((u_{in}, v_{in}, w_{in})\), are composed of the mean base flow and the fluctuating components

\[
\begin{align*}
    u_{in} &= U + \sum_{j=1}^{N} \tilde{u}_j \\
    v_{in} &= V + \sum_{j=1}^{N} \tilde{v}_j \\
    w_{in} &= W + \sum_{j=1}^{N} \tilde{w}_j
\end{align*}
\]

(3.49)

where \((U,V,W)\) is the mean velocity vector, and \((\tilde{u}_j, \tilde{v}_j, \tilde{w}_j)\) is the velocity vector of the \(j\)th THV. Each THV has an independent random center, radius, and amplitude.

### 3.3.6 Determination of the Amplitudes

In the original SEM of Jarrin et al. [47], the fluctuating velocity field is rescaled using the Cholesky decomposition of the Reynolds stress tensor in order to ensure reproduction of the desired Reynolds stresses. Since those synthetic eddies already violate the divergence-free condition, the fact that rescaling using the Cholesky decomposition generally violates the divergence-free condition is of little consequence. Because the THV is divergence-free, the amplitudes of the THV’s need to be calculated using a different method in order to match the desired Reynolds stresses and also preserve the divergence-free nature of the synthetic field.

Smirnov et al. [150] proposed a method, which was later expanded upon by Davidson and Bilson [29], to match the Reynolds stresses for anisotropic turbulence. The idea behind the method is to find a local reference system where the normal Reynolds stress terms are non-zero, while the off-diagonal Reynolds stress terms are zero, calculate the associated
amplitudes, and then transform the signal back to the global reference system. To be able
to apply this method in the current THV framework, the creation of the THV’s has to take
place in the local principal-axis reference system. This begins by calculating the principal-
axis Reynolds stresses and eigenvectors of the local Reynolds stress tensor. Examining the
principal-axis Reynolds stresses for a single stream of THV’s can give insight into how
to set each of the principal-axis amplitudes, \((u_{0x}^{p}, u_{0y}^{p}, u_{0z}^{p})\), where superscript \(p\) denotes
variables in the principal-axis coordinate system. Since the velocity components are at a
maximum, consider the principal-axis Reynolds stresses at the center of a single stream of
THV’s.

\[
\langle \tilde{u}\tilde{u} \rangle^p = \left\langle (u_{0x}^{p})^2 (f_{x}^{u,p})^2 \right\rangle + \left\langle (u_{0y}^{p})^2 (f_{y}^{u,p})^2 \right\rangle + \left\langle (u_{0z}^{p})^2 (f_{z}^{u,p})^2 \right\rangle \\
\langle \tilde{v}\tilde{v} \rangle^p = \left\langle (u_{0x}^{p})^2 (f_{x}^{v,p})^2 \right\rangle + \left\langle (u_{0y}^{p})^2 (f_{y}^{v,p})^2 \right\rangle + \left\langle (u_{0z}^{p})^2 (f_{z}^{v,p})^2 \right\rangle \\
\langle \tilde{w}\tilde{w} \rangle^p = \left\langle (u_{0x}^{p})^2 (f_{x}^{w,p})^2 \right\rangle + \left\langle (u_{0y}^{p})^2 (f_{y}^{w,p})^2 \right\rangle + \left\langle (u_{0z}^{p})^2 (f_{z}^{w,p})^2 \right\rangle
\]

\[
\langle \tilde{u}\tilde{v} \rangle^p = \langle \tilde{u}\tilde{w} \rangle^p = \langle \tilde{v}\tilde{w} \rangle^p = 0
\]

Let the amplitudes in the principal-axis coordinate system associated with a THV be de-
defined as a product of a constant amplitude and a random number,

\[
\begin{bmatrix}
    \hat{u}_{x}^{p} \\
    \hat{u}_{y}^{p} \\
    \hat{u}_{z}^{p}
\end{bmatrix} =
\begin{bmatrix}
    \epsilon^{x} \hat{u}_{x}^{p} \\
    \epsilon^{y} \hat{u}_{y}^{p} \\
    \epsilon^{z} \hat{u}_{z}^{p}
\end{bmatrix}
\]

(3.51)
where $\epsilon_x$, $\epsilon_y$, and $\epsilon_z$ are independent random numbers such that $\epsilon = \pm 1$, $\langle \epsilon \rangle = 0$, and $\langle \epsilon^2 \rangle = 1$. Since the amplitudes and the constant shape functions are independent, after inserting equation (3.51) and using the properties of $\epsilon$, equation (3.50) becomes

\[
\begin{bmatrix}
\langle \tilde{u} u \rangle^p \\
\langle \tilde{v} v \rangle^p \\
\langle \tilde{w} w \rangle^p
\end{bmatrix} = \begin{bmatrix}
(f^{x,p}_u)^2 (f^{x,p}_u)^2 (f^{x,p}_u)^2 \\
(f^{y,p}_v)^2 (f^{y,p}_v)^2 (f^{y,p}_v)^2 \\
(f^{z,p}_w)^2 (f^{z,p}_w)^2 (f^{z,p}_w)^2
\end{bmatrix} \begin{bmatrix}
\langle \tilde{u} u \rangle^p \\
\langle \tilde{v} v \rangle^p \\
\langle \tilde{w} w \rangle^p
\end{bmatrix} = F_{uvw}^{xyz} \begin{bmatrix}
\langle \tilde{u} u \rangle^p \\
\langle \tilde{v} v \rangle^p \\
\langle \tilde{w} w \rangle^p
\end{bmatrix}
\]

(3.52)

$F_{uvw}^{xyz}$ is constant and invertible. Thus, the amplitudes of a THV can be calculated from the given principal-axis Reynolds stresses. By combining the amplitudes calculated in equation (3.52) with equations (3.51) and (3.48),

\[
\begin{bmatrix}
u^x_p \\
u^y_p \\
u^z_p
\end{bmatrix} = \begin{bmatrix}
\epsilon_x \hat{u}_0^x f_x^x \\
\epsilon_y \hat{u}_0^y f_y^y \\
\epsilon_z \hat{u}_0^z f_z^z
\end{bmatrix}, \quad \begin{bmatrix}
v^x_p \\
v^y_p \\
v^z_p
\end{bmatrix} = \begin{bmatrix}
\epsilon_x \hat{v}_0^x f_x^y \\
\epsilon_y \hat{v}_0^y f_y^y \\
\epsilon_z \hat{v}_0^z f_z^z
\end{bmatrix}, \quad \begin{bmatrix}
w^x_p \\
w^y_p \\
w^z_p
\end{bmatrix} = \begin{bmatrix}
\epsilon_x \hat{w}_0^x f_x^w \\
\epsilon_y \hat{w}_0^y f_y^w \\
\epsilon_z \hat{w}_0^z f_z^w
\end{bmatrix}
\]

(3.53)

and then inserting the result into equation (3.43), the velocity contribution of a single THV in the local principal-axis reference system can be found as

\[
\begin{align*}
\tilde{u}^p &= u^x_p + u^y_p + u^z_p \\
\tilde{v}^p &= v^x_p + v^y_p + v^z_p \\
\tilde{w}^p &= w^x_p + w^y_p + w^z_p
\end{align*}
\]

(3.54)

Distribution Statement A. Approved for public release; distribution is unlimited.
The last step is to transform the velocities from the local principal-axis reference system to the global reference system using the transformation matrix created from the eigenvectors of the local Reynolds stress tensor, $T_p^G$.

\[
\begin{bmatrix}
\tilde{u} \\
\tilde{v} \\
\tilde{w}
\end{bmatrix} = T_p^G 
\begin{bmatrix}
\tilde{u}^p \\
\tilde{v}^p \\
\tilde{w}^p
\end{bmatrix}
\]  

(3.55)

This is the velocity contribution of a single THV in the global reference system that is combined with all the other THV’s in equation (3.49) to create the imposed inflow.

3.3.7 Modification of the Target Reynolds Stresses

As is defined in Section 3.3.6, the imposed Reynolds stress tensor at any point in space is reproduced by a single stream of THV’s moving through that point in time. To account for the fact that the imposed turbulent inflow is composed of many THV’s over a range of different sizes, the method for determining the amplitudes of each THV needs to include the influence of the THV’s currently at the inlet when a new THV is created in order to ensure that Reynolds stress matching is recovered. The target Reynolds stress modification procedure is proposed to address this issue.

Consider the inlet plane shown in Figure 3.1. The black circles represent THV’s that are currently passing through the inlet and the red circle represents a new THV that is about to be created. If the surrounding THV’s did not exist, the calculated amplitudes of the new THV ensure Reynolds stress matching at the center of the THV. Since the surrounding THV’s already exist and are influencing the flow field at the center of the new THV, the
amplitudes of the new THV need to be modified to account for the contributions of its neighbors.

![Figure 3.1](image)

Figure 3.1

An example of inlet plane: black) existing THV’s; red) new THV.

Looking at equation (3.39), the velocity components for a Hill’s vortex approach zero quickly when moving away from the surface of the vortex. Thus, only the contribution of THV’s that encircle the center of the new THV will be considered (the center of THV new falls within the boundaries of THV 1, 2, and 3). In the global coordinate system, the
Reynolds stresses at the point on the inlet where the center of the new THV will pass through are defined as

\[
\langle u'_i u'_j \rangle_{existing} = \left\langle \left( \sum_{m=1}^{M_s} (\tilde{u}_i)_m \right) \left( \sum_{n=1}^{N_s} (\tilde{u}_j)_n \right) \right\rangle \tag{3.56}
\]

\[
= \sum_{m=1}^{M_s} \sum_{n=1}^{N_s} \langle (\tilde{u}_i)_m (\tilde{u}_j)_n \rangle \tag{3.57}
\]

where \( M_s \) and \( N_s \) are both equal to the number of encircling THV’s. Since the random numbers associated with each THV are independent,

\[
\langle (\tilde{u}_i)_m (\tilde{u}_j)_n \rangle = 0 \quad \text{for} \quad m \neq n \tag{3.58}
\]

Thus, the Reynolds stress contribution from the surrounding THV’s is

\[
\langle u'_i u'_j \rangle_{existing} = \sum_{n=1}^{N_s} \langle \tilde{u}_i \tilde{u}_j \rangle_n \tag{3.59}
\]

where \( N_s \) is again the number of encircling THV’s. This contribution Reynolds stress tensor is then used to modify the given Reynolds stress tensor that is being matched.

\[
\langle u'_i u'_j \rangle_{target} = \langle u'_i u'_j \rangle_{given} - \langle u'_i u'_j \rangle_{existing} \tag{3.60}
\]

The target Reynolds stress tensor can then be used in the THV creation framework to determine the amplitudes of the new THV. As a note, if any of the eigenvalues of the target Reynolds stress tensor are negative, that means that the given Reynolds stresses are already reproduced by the present THV’s and the new THV does not need to be created at that point on the inlet.
3.3.7.1 Reynolds Stresses for a Single THV

The Reynolds stress tensors of the individual THV’s needed by equation (3.59) can be found by first considering the velocity components of a single THV in the global coordinate system.

\[
\tilde{u}_i = u_0^x f_i^x + u_0^y f_i^y + u_0^z f_i^z
\]  (3.61)

\((f_i^x, f_i^y, f_i^z)\) are the shape functions for the three Hill’s vortices associated with the \(\tilde{u}_i\) velocity component. \((u_0^x, u_0^y, u_0^z)\) are the randomized amplitudes in the global coordinate system and are defined as

\[
\begin{bmatrix}
u_0^x \\
u_0^y \\
u_0^z \\
\end{bmatrix} = T_p^G \begin{bmatrix}
\epsilon^x \hat{\tilde{u}}_{0x}^p \\
\epsilon^y \hat{\tilde{u}}_{0y}^p \\
\epsilon^z \hat{\tilde{u}}_{0z}^p \\
\end{bmatrix}
\]  (3.62)

where \(T_p^G\) is the eigenvector transformation matrix, \((\hat{\tilde{u}}_{0x}^p, \hat{\tilde{u}}_{0y}^p, \hat{\tilde{u}}_{0z}^p)\) are the constant amplitudes in the principal coordinate system, and \((\epsilon^x, \epsilon^y, \epsilon^z)\) have the following properties

\[
\epsilon^x = \pm 1 ; \quad \epsilon^y = \pm 1 ; \quad \epsilon^z = \pm 1
\]  (3.63)

\[
\langle \epsilon^x \rangle = \langle \epsilon^y \rangle = \langle \epsilon^z \rangle = 0
\]

\[
\langle (\epsilon^x)^2 \rangle = \langle (\epsilon^y)^2 \rangle = \langle (\epsilon^z)^2 \rangle = 1
\]

\[
\langle \epsilon^x \epsilon^y \rangle = \langle \epsilon^x \epsilon^z \rangle = \langle \epsilon^y \epsilon^z \rangle = 0
\]

By taking the time-average of the products of the velocity components in equation (3.61) and recognizing that the amplitudes and shape functions are independent from each other,
expressions can be found for the Reynolds stresses of an infinite stream of THV’s being convected through the same location at the inlet.

\[
\langle \tilde{u}_i \tilde{u}_j \rangle = \langle (u_0^x)^2 \rangle \langle f_i^x f_j^x \rangle + \langle (u_0^y)^2 \rangle \langle f_i^y f_j^y \rangle + \langle (u_0^z)^2 \rangle \langle f_i^z f_j^z \rangle 
\]

(3.64)

\[
+ \left( (f_i^x f_j^y) + (f_j^y f_i^x) \right) \langle u_0^x u_0^y \rangle + \left( (f_i^y f_j^z) + (f_j^z f_i^y) \right) \langle u_0^y u_0^z \rangle 
\]

\[
+ \left( (f_i^z f_j^x) + (f_j^x f_i^z) \right) \langle u_0^z u_0^x \rangle 
\]

The time-average of the products of the shape functions can be solved for analytically by assuming the time period for a single THV to flow through the inlet is

\[
T_{in} = \frac{4a}{U} 
\]

(3.65)

where \(a\) is the radius of the THV and \(U\) is the mean velocity at the center of the THV.

Through the multiplying of the global amplitudes in equation (3.62), taking the time-average of the resulting products, recognizing that the eigenvector transformation matrix and the principal coordinate system amplitudes are all constant, and finally inserting the properties of the random numbers seen in equation (3.63); the time-averaged global coordinate system amplitude correlations can be determined.

\[
\langle u_0^i u_0^j \rangle = (t_p^G)_{i1} (t_p^G)_{j1} \langle \hat{u}_0^{x_p} \rangle^2 + (t_p^G)_{i2} (t_p^G)_{j2} \langle \hat{u}_0^{y_p} \rangle^2 + (t_p^G)_{i3} (t_p^G)_{j3} \langle \hat{u}_0^{z_p} \rangle^2 
\]

(3.66)

t_p^G are the individual elements of the eigenvector transformation matrix, \(T_p^G\).

3.3.8 Generations of Triple Hill’s Vortices

To be able to recreate turbulent flow, the Reynolds stress tensor is not the only quantity that needs to be matched. Turbulent structures occur over a wide range of length scales, and
the population varies over a length scale distribution which can be taken from the turbulent kinetic energy spectrum. Thus, the number of larger structures is less than the number of medium sized structures, which is less than the number of smaller structures. This type of distribution also needs to be replicated to make sure the space is fully covered by structures in the whole range of wavenumbers. The minimum, $a_{\text{min}}$, and maximum, $a_{\text{max}}$, radii of the THV allowed are dictated by the grid resolution and the integral length scale or the domain size, respectively. As shown in Jarrin et al. [48], the overall number of imposed THV’s are calculated as follows,

$$N = C \frac{A_{\text{in}}}{a_{\text{min}}^2}$$

(3.67)

where $N$ is the total number of THV’s, $A_{\text{in}}$ is the area of the inlet plane, and $C$ is a proportionality constant. This number of THV’s will ensure that the inflow plane is fully populated with THV’s at any moment in time. Jarrin et al. [48] recommends that $C = 1$ to minimize the computational overhead required with any unnecessary THV’s. Given the limits for the smallest and the largest eddy, the idea of defining multiple generations of THV’s is introduced here. The range of radii is divided discretely into several subranges, or generations. Based on a given distribution, the number of THV’s in each generation is set. Table 3.2 shows an example of five generations of THV’s, where $a_{\text{min}} < a_4 < a_3 < a_2 < a_1 < a_{\text{max}}$ and $N = N_1 + N_2 + N_3 + N_4 + N_5$

The multiple generations of THV’s ensure that the inlet is covered with enough multiscale structures. The range of locations for these generations can also be restricted, for example, to cluster smaller THV’s near a wall for wall-bounded flows.

Distribution Statement A. Approved for public release; distribution is unlimited.
Table 3.2
Example of five generations of THV’s

<table>
<thead>
<tr>
<th>Generation</th>
<th>Range of Radii</th>
<th>Number of THV’s</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>(a_1 &lt; a &lt; a_{\text{max}})</td>
<td>(N_1)</td>
</tr>
<tr>
<td>2</td>
<td>(a_2 &lt; a &lt; a_1)</td>
<td>(N_2)</td>
</tr>
<tr>
<td>3</td>
<td>(a_3 &lt; a &lt; a_2)</td>
<td>(N_3)</td>
</tr>
<tr>
<td>4</td>
<td>(a_4 &lt; a &lt; a_3)</td>
<td>(N_4)</td>
</tr>
<tr>
<td>5</td>
<td>(a_{\text{min}} &lt; a &lt; a_4)</td>
<td>(N_5)</td>
</tr>
</tbody>
</table>

For homogeneous turbulence, each generation of THV matches a portion of the overall Reynolds stress tensor. The given Reynolds stress tensor in equation (3.60) is multiplied by a factor, \(b_m\), based on the ratio of turbulent kinetic energy (TKE) contained in that specific generation, \(TKE_m\), to the total turbulent kinetic energy, \(TKE\).

\[
b_m = \frac{TKE_m}{TKE} ; \quad \sum_{m=1}^{M} b_m = 1
\]  

(3.68)

\(M\) is the total number of THV generations and \(m\) represents the current generation. Thus, the equation for the target Reynolds stresses for a particular THV, equation (3.60), is modified as follows.

\[
\langle u'_i u'_j \rangle_{\text{target}} = b_m \langle u'_i u'_j \rangle_{\text{given}} - \langle u'_i u'_j \rangle_{\text{existing}}
\]  

(3.69)

The Reynolds stress contribution from the surrounding THV’s, \(\langle u'_i u'_j \rangle_{\text{existing}}\), is only calculated over the THV’s in the same generation as the THV being created.

\[
\langle u'_i u'_j \rangle_{\text{existing}} = \sum_{n=1}^{N_m} \langle \tilde{u}_i \tilde{u}_j \rangle_n
\]  

(3.70)

\(N_m\) is the number of encircling THV’s in the same generation. By only matching a portion of the TKE, THV’s from multiple generations can exist on top of each other. This enables...
energy to be contributed over a wider range of scales. For the homogeneous isotropic turbulence case in Section 3.4.2, each THV generation is given an equal portion of the TKE, $b_m = 1/M$.

### 3.3.9 Near-wall THV Stretching

A spherical Triple Hill’s Vortex created near a wall does not agree with the observed elongated flow structures observed in the near-wall region. A stretching operation is introduced to produce angled, elongated THV’s in the near-wall region consistent with the observations of Jeong et al. [49] and Sibilla and Beretta [145]. The physical coordinates system associated with the center of a near-wall THV is stretched according to a streamwise length scale, rotated about the spanwise coordinate axis according to a inclination angle, and rotated about the vertical coordinate axis according to a tilting angle. Figure 3.2 shows a diagram illustrating the stretched and rotated THV.

\[
\begin{bmatrix}
x'_s \\
y'_s \\
z'_s \\
\end{bmatrix} = \begin{bmatrix}
\frac{1}{2}L_x & 0 & 0 \\
0 & 1 & 0 \\
0 & 0 & 1 \\
\end{bmatrix} \begin{bmatrix}
cos(\alpha) & -\sin(\alpha) & 0 \\
\sin(\alpha) & \cos(\alpha) & 0 \\
0 & 0 & 1 \\
\end{bmatrix} \begin{bmatrix}
\cos(\beta) & 0 & -\sin(\beta) \\
0 & 1 & 0 \\
\sin(\beta) & 0 & \cos(\beta) \\
\end{bmatrix} \begin{bmatrix}
x' \\
y' \\
z' \\
\end{bmatrix}
\]

(3.71)

\((x', y', z')\) are the physical coordinates associated with the center of a THV and \((x'_s, y'_s, z'_s)\) are the stretched and rotated coordinates that are then used to calculate the velocity components of the THV. \(L_x/2a\) is the length to diameter ratio and it is multiplied by \(1/2\) in equation (3.71) because the spherical THV is stretched equally in the positive and negative streamwise direction. The inclination angle is \(\alpha\) and \(\beta\) is the tilting angle. Jeong et al.
[49] found that the inclination angle was constant and that the tilting angle varied between $-\beta_{\text{max}}$ and $\beta_{\text{max}}$ depending on the local vertical component of vorticity. Here,

$$\beta = \epsilon^\beta \beta_{\text{max}}$$  \hspace{1cm} (3.72)

where $\epsilon^\beta$ is a random number defined such that $\langle \epsilon^\beta \rangle = 0$ and $\langle \epsilon^{2\beta} \rangle = 1$. The constant stretching and rotation procedure allows for the continued satisfaction of the divergence-free condition for a THV.

![Diagram of THV](image)

**Figure 3.2**

A diagram of the stretched THV imposed in the near-wall region: a) inclination angle; b) tilting angle.

### 3.4 Results

Four test cases were considered: convection of an isolated Triple Hill’s Vortex, homogeneous isotropic turbulence, turbulent channel flow, and a turbulent mixing layer.
3.4.1 Single Triple Hill’s Vortex

The convection of a single Triple Hill’s Vortex was investigated. The THV was generated at the inlet plane and convected downstream by a uniform mean flow. The dimensions of the domain were $12a \times 6a \times 6a$ in the streamwise, vertical, and spanwise directions and it was discretized using a uniform Cartesian grid with $120 \times 60 \times 60$ grid points. Far field boundary conditions were used on the vertical and spanwise boundaries.

Contours of the velocity magnitude on planes through the center of a single convected THV are shown in Figure 3.3. The contours clearly show the spherical, and thus symmetric, nature of the THV. The $xy$- and $xz$-planes also show the stretching of the THV by the convecting flow.

![Contours of the velocity magnitude on planes through the center of a THV.](xy-plane)(xz-plane)(yz-plane)

Figure 3.3

Contours of the velocity magnitude on $xy$ (left), $xz$ (middle), and $yz$ (right) planes through the center of a THV.

Figure 3.4 depicts the generation in time of a single THV from the inlet. Each of the frames is a constant time step apart. In the first frame, the THV can be seen just beginning to emerge from the inlet. Notice that there are no spurious waves originating from the front.
of the THV. Then, moving forward in time, the next two frames show the THV as half of it is generated and then as it just leaves the inlet. In the last frame, the THV has fully released from the inlet. Notice that the THV passed through the inflow boundary cleanly, with no spurious waves originating from the back of the THV, which is an indication that the divergence-free condition is satisfied (otherwise, spurious waves may be generated in the downstream of the eddy). In the synthetic turbulence model, analyzed next, when a THV is released from the inlet, it is no longer acted upon by the THV SEM and the creation process begins again with a new THV at a new random location.

![Images of THV generation and release](image.png)

Figure 3.4

A single THV being generated at the inlet. Time is increasing with a constant time step from a) to d) and the inlet is on the left side of each frame.

3.4.2 Homogeneous Turbulent Flow

Large eddy simulation of isotropic turbulence was performed and the results were compared to experimental data collected by Comte-Bellot and Corrsin [27, 28]. Comte-Bellot and Corrsin measured the temporal decay of the turbulent kinetic energy of mesh-generated...
isotropic turbulence in a wind tunnel. For this case, a study of the spatial decay of the turbulent kinetic energy was used to validate the THV synthetic eddy method.

The numerical domain corresponds to the contracted test section of the wind tunnel downstream of the mesh. The dimensions of the domain were $120L_{\text{mesh}} \times 10L_{\text{mesh}} \times 10L_{\text{mesh}}$ in the streamwise, vertical, and spanwise directions, where $L_{\text{mesh}}$ is the size of the experimental mesh, which equals 0.0508m [27]. In both the vertical and spanwise directions, the numerical domain size was chosen to be ten experimental mesh sizes with periodic boundary conditions, instead of using the entire cross-section of the wind tunnel test section. A uniform Cartesian grid consisting of $1200 \times 100 \times 100$ grid points was used.

The increased artificial viscosity region was added for the last $20L_{\text{mesh}}$ in the streamwise direction at the outflow. A $M_\infty = 0.05$ uniform mean flow was imposed at the inflow, which is slightly larger than the experimental mean velocity (12.7 m/s); this is because the numerical algorithm is not able to handle very low Mach number flows. The non-dimensional streamwise turbulence intensity of the synthetic eddies imposed at the inlet corresponds to level reported at $42L_{\text{mesh}}$ in Comte-Bellot and Corrsin [28]. The THV’s were broken into ten generations, as given in Table 3.3, where the number of THV’s in each generation was determined using equation (3.67). The maximum radius of a THV was $1.5L_{\text{mesh}}$ and the minimum radius was based on the grid resolution (five to six grid points across the smallest THV).

Isosurfaces of Q-criterion and contours of vorticity magnitude are shown in Figure 3.5. The THV’s are generated at the inlet on the left and are convected downstream to the right. It can be seen from the isosurfaces in the top of Figure 3.5 that the shape of the generated
Table 3.3
THV generations for the homogeneous isotropic case

<table>
<thead>
<tr>
<th>Generation</th>
<th>Range of Radii</th>
<th>Number of THV’s</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$1.375 \ L_{\text{mesh}} &lt; a &lt; 1.5 \ L_{\text{mesh}}$</td>
<td>8</td>
</tr>
<tr>
<td>2</td>
<td>$1.25 \ L_{\text{mesh}} &lt; a &lt; 1.375 \ L_{\text{mesh}}$</td>
<td>12</td>
</tr>
<tr>
<td>3</td>
<td>$1.125 \ L_{\text{mesh}} &lt; a &lt; 1.25 \ L_{\text{mesh}}$</td>
<td>22</td>
</tr>
<tr>
<td>4</td>
<td>$1.0 \ L_{\text{mesh}} &lt; a &lt; 1.125 \ L_{\text{mesh}}$</td>
<td>28</td>
</tr>
<tr>
<td>5</td>
<td>$0.875 \ L_{\text{mesh}} &lt; a &lt; 1.0 \ L_{\text{mesh}}$</td>
<td>36</td>
</tr>
<tr>
<td>6</td>
<td>$0.75 \ L_{\text{mesh}} &lt; a &lt; 0.875 \ L_{\text{mesh}}$</td>
<td>48</td>
</tr>
<tr>
<td>7</td>
<td>$0.625 \ L_{\text{mesh}} &lt; a &lt; 0.75 \ L_{\text{mesh}}$</td>
<td>68</td>
</tr>
<tr>
<td>8</td>
<td>$0.5 \ L_{\text{mesh}} &lt; a &lt; 0.625 \ L_{\text{mesh}}$</td>
<td>104</td>
</tr>
<tr>
<td>9</td>
<td>$0.375 \ L_{\text{mesh}} &lt; a &lt; 0.5 \ L_{\text{mesh}}$</td>
<td>180</td>
</tr>
<tr>
<td>10</td>
<td>$0.25 \ L_{\text{mesh}} &lt; a &lt; 0.375 \ L_{\text{mesh}}$</td>
<td>400</td>
</tr>
</tbody>
</table>

THV’s change from larger and compact structures near the inflow to smaller and stretched eddies farther downstream. In the proximity to the inflow boundary, one can hardly notice any transition from artificial to realistic turbulence. Comparing the regions near the inlet and outlet, there has been a noticeable dissipation of the eddies, which is expected. The middle and bottom contours in Figure 3.5 further show this dissipation of the turbulent structures. The magnitude of the vorticity decreases moving downstream from the inlet, as expected.

Two-point spatial velocity correlations and non-dimensionalized turbulent kinetic energy spectrum of the synthetic fluctuations at the inlet plane are shown in Figure 3.6 along with experimental data collected at $42 L_{\text{mesh}}$ by Comte-Bellot and Corrsin [28]. Looking at the spatial correlations in Figure 3.6(a), very good agreement is found for both the longitudinal and transverse correlations. Other than providing insight into what the maximum
Top) Isosurfaces of Q-criterion. Contours of vorticity magnitude: middle) xy-plane through the center of the domain; bottom) xz-plane through the center of the domain.
radius of the largest sized THV generation should be, the spatial correlations are not used as a control on the creation of the THV’s. The energy spectrum in Figure 3.6(b) also shows the same good agreement with the experimental data. Like with the spatial correlations, the only influence exerted over the energy spectrum of the created THV’s is through the definition of the THV generations. While each THV generation is given an equal fraction of the total TKE, the wavenumber range inherently present in each THV generation is different. Because each THV is a coherent structure, each one contributes to the energy spectrum over the range of scales from the overall size of the THV to the smallest scale resolved by the grid.

Figure 3.6

Two-point spatial correlations (a) and turbulent kinetic energy spectrum as a function of wavenumber (b), captured at the inlet, compared with the experimental data collected by Comte-Bellot and Corrsin [28]. a) ——) transverse; - - -) longitudinal; O) experimental transverse; X) experimental longitudinal. b) O) THV SEM; ——) experimental.

Distribution Statement A. Approved for public release; distribution is unlimited.
Figure 3.7(a) shows the spatial decay of the turbulent kinetic energy for the numerical results compared with the $k^{-1.25}$ power law fit to experimental data collected by Comte-Bellot and Corrsin [27]. The THV SEM was able to reproduce the rate of turbulent kinetic energy decay seen in the experimental data. The slower decay of TKE present near the inlet is caused by the larger THV’s at the inlet containing more of the TKE. This is seen in the energy spectrum at the inlet shown in Figure 3.6(b), where the energy of the lower wavenumbers is slightly greater than the experimental results and the energy of the higher wavenumbers is slightly less than the experimental results. Even though the filtering required by the numerical method for stability was keep to a minimum, this added dissipation contributed to the slightly quicker TKE decay. The lack of a recovery region just after the inlet further reinforces that the THV SEM is divergence-free.

The downstream development of the velocity derivative skewness is shown in Figure 3.7(b) along with LES data from Jarrin et al. [47] of spatially decaying homogeneous isotropic turbulence at a similar integral length scale Reynolds number. At the inlet, the skewness of the synthetic fluctuations generated using THV’s is zero, which is expected because the THV’s are symmetric. Moving shortly downstream, the skewness rapidly increases in magnitude for the flow generated by the THV SEM, while the skewness of the original SEM flow requires a much longer development length. Even though the Gaussian and tent function synthetic eddies of the original SEM and the THV’s are all symmetric, the more physically based THV allows for the synthetic field to transition to realistic turbulence much quicker than the more abstract shapes.
Turbulent kinetic energy (a) and velocity derivative skewness (b) in the streamwise direction compared with the experimental data collected by Comte-Bellot and Corrsin [27] and LES data from Jarrin et al. [47]: —–) THV SEM; O) $k^{-1.25}$ experimental fit; - - -) original SEM.

A comparison of the turbulent kinetic energy decay and the longitudinal two-point spatial correlation at the inlet (already presented in Figure 3.7(a) and Figure 3.6(a), respectively) with the LES data from Dietzel et al. [34] is shown in Figure 3.8. Dietzel et al. [34] simulated the temporal decay of homogeneous isotropic turbulence using the following three synthetic turbulence methods to generate synthetic initial conditions: the digital filtering method proposed by Klein et al. [66] and improved by Kempf et al. [59], the diffusion method of Kempf et al. [58], and the Fourier method of Billson et al. [16] and Davidson [30]. The decay of the TKE for the THV SEM exhibits similar behavior to the divergence-free (for isotropic turbulence) Fourier method, whereas the digital filtering and diffusion methods both show the significant dissipation immediately after synthetic fluctu-
ations are imposed that is characteristic of methods that do not satisfy the divergence-free condition.

![Figure 3.8](image)

Turbulent kinetic energy in the streamwise direction (a) and longitudinal two-point spatial correlation at the inlet compared with the experimental data collected by Comte-Bellot and Corrsin [27] and LES data from Dietzel et al. [34]: black) THV SEM; O) experimental; blue) digital filtering method [66, 59]; red) diffusion method [58]; yellow) Fourier method [16, 30].

Two-point spatial velocity correlations and non-dimensionalized turbulent kinetic energy spectrum $56L_{\text{mesh}}$ downstream of the inlet plane are shown in Figure 3.9 along with experimental data collected at the corresponding downstream location by Comte-Bellot and Corrsin [28]. Again, very good agreement with the experimental results is seen for both the spatial correlations in Figure 3.9(a) and the energy spectrum in Figure 3.9(b).
The well-modeled synthetic fluctuations imposed by the THV SEM at the inlet quickly developed into realistic turbulence and correctly reproduced downstream statistics.

![Figure 3.9](image)

Two-point spatial correlations (a) and turbulent kinetic energy spectrum as a function of wavenumber (b), captured $56L_{mesh}$ downstream of the inlet, compared with the experimental data collected by Comte-Bellot and Corrsin [28]. a) —–) transverse; - - -) longitudinal; O) experimental transverse; X) experimental longitudinal. b) O) THV SEM; —–) experimental.

### 3.4.3 Turbulent Channel Flow

To further test the capability of the THV SEM to model anisotropic non-homogeneous turbulence, a turbulent channel flow case was considered. Reynolds stress tensor and mean velocity profiles from the DNS of a fully turbulent channel at $Re_\tau = 395$ collected by Moser et al. [105] were imposed at the inlet plane. The dimensions of the domain were $15\delta \times 2\delta \times 3\delta$ in streamwise, vertical, and spanwise directions, where $\delta$ is the channel...
half height. A Cartesian grid composed of $200 \times 100 \times 100$ grid points was used. The grid was uniformly spaced in the streamwise and spanwise directions, and stretched in the vertical direction in order to increase resolution at the walls. This allowed for grid spacings of $\Delta x^+ = 30$, $\Delta z^+ = 12$, $\Delta y_{min}^+ = 1$, and $\Delta y_{max}^+ = 17.5$. No-slip wall boundary conditions were used on the bottom and top boundaries, and periodic boundary conditions were applied in the spanwise direction. The increased artificial viscosity region was added for the last $3\delta$ in the streamwise direction at the outflow. 6218 THV’s were imposed at any one instant in time on to the mean flow at the inflow boundary. The maximum radius allowed was $0.6\delta$, while the minimum radius was $0.006\delta$. The THV’s were split into five generations, with the first generation being the elongated eddies described in Section 3.3.9 and the fifth generation being clustered very near the walls. This clustering of THV’s replicates the smallest turbulent structures very near the wall.

Table 3.4

<table>
<thead>
<tr>
<th>Generation</th>
<th>Range of Radii</th>
<th>Range of $y$</th>
<th>Number of THV’s</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>$0.24\delta &lt; a &lt; 0.32\delta$</td>
<td>$0.64\delta &lt;</td>
<td>y</td>
</tr>
<tr>
<td>2</td>
<td>$0.4\delta &lt; a &lt; 0.6\delta$</td>
<td>$0 &lt;</td>
<td>y</td>
</tr>
<tr>
<td>3</td>
<td>$0.1\delta &lt; a &lt; 0.2\delta$</td>
<td>$0 &lt;</td>
<td>y</td>
</tr>
<tr>
<td>4</td>
<td>$0.02\delta &lt; a &lt; 0.1\delta$</td>
<td>$0 &lt;</td>
<td>y</td>
</tr>
<tr>
<td>5</td>
<td>$0.006\delta &lt; a &lt; 0.01\delta$</td>
<td>$0.9\delta &lt;</td>
<td>y</td>
</tr>
</tbody>
</table>

Contours of streamwise velocity and vorticity magnitude at the inflow plane are shown in Figure 3.10. These contours show the synthetic inflow that was imposed by the THV.
SEM. Notice how the concept of the generations of THV’s manifests itself in the synthetic inflow with a few large eddies surrounded by ever smaller structures. Moving from the center of the channel to the walls, the size the imposed eddies decreases. The vorticity magnitude contour especially illuminates the effect of the clustered eddies near the wall, where the vorticity magnitude is larger as compared to the center of the channel. The development of the elongated turbulent structures stretching downstream from the walls can be seen from the Q-criterion isosurfaces in Figure 3.11. While the synthetic eddies are released into the domain and quickly evolve into realistic turbulent structures, there is still a small region near the inlet where the non-stretched THV’s of generations two through five need to develop.

![Contours for turbulent channel flow at the inlet plane](image)

**Figure 3.10**

Contours for turbulent channel flow at the inlet plane (from the synthetic turbulence model): a) streamwise velocity; b) vorticity magnitude.

Vertical profiles of the mean streamwise velocity and Reynolds stresses are plotted in Figure 3.12, where averaging in both time and the homogeneous spanwise directions has

**Distribution Statement A. Approved for public release; distribution is unlimited.**
been taken. As seen in Figure 3.12(a), the DNS mean streamwise velocity profile is well reproduced. When looking at the Reynolds stresses in Figure 3.12(b), there is excellent agreement between the heights of $-0.9\delta$ and $0.9\delta$. This height range corresponds to the effective region where most of the THV’s are imposed. A portion of the smaller THV’s of the fourth generation and the fifth clustered generation are imposed at heights closer to the wall, where their effect is clearly seen in the bulge of the $\langle u'u' \rangle$ profile very close to the walls, but those THV’s are not enough to reproduce the DNS profiles. With more smaller and smaller THV’s in those nearest wall regions, those Reynolds stress profiles will be better matched.

Two-point spanwise spatial velocity correlations of the synthetic fluctuations at the inlet plane and at three different heights from the wall are shown in Figure 3.13 along with the corresponding DNS data from Moser et al. [105]. There is generally good agreement at spanwise distances less than $0.4\delta$. Closer to the wall, Figure 3.13(a), the synthetic fluctuations are composed of the smaller THV’s. They are only locally correlated, so the

Distribution Statement A. Approved for public release; distribution is unlimited.
Vertical profiles at the inlet compared with the DNS data collected by Moser et al. [105] of: a) mean streamwise velocity; b) Reynolds stresses; —–) THV SEM; - - -) DNS data from Moser et al. [105].

spatial correlations quickly go to zero. Moving away from the wall, Figure 3.13(b), and towards the center of the channel, Figure 3.13(c), the largest THV’s become more present. This is seen in the higher correlations at larger spanwise distances, which might be a hint that the largest THV’s are too large in size. As a general trend across all three heights from the wall, the THV’s do not reproduce negative spatial correlations, especially at larger spanwise distances. The transverse correlation $0.2\delta$ from the wall does dip slightly below zero, which was also seen for the transverse correlation at the inlet of the homogeneous isotropic turbulence (Figure 3.6(a)), but it does not reproduce the negative correlation.

Figure 3.14 shows the non-dimensionalized spanwise turbulent kinetic energy spectra of the synthetic fluctuations at the inlet plane and at three different heights from the wall.
Two-point spanwise spatial correlations, captured at the inlet and at three different heights from the wall, compared with the DNS data collected by Moser et al. [105]. blue) transverse; red) longitudinal; - - -) DNS transverse; -.-.-) DNS longitudinal.

along with the corresponding DNS data from Moser et al. [105]. There is good agreement at all three heights in the higher wavenumbers, with better agreement when moving towards the wall. Unlike for homogeneous turbulence where each THV generation matches an equal fraction of the TKE at any point, because of the non-homogeneous nature of the wall normal direction, each individual THV matches the total TKE at any point. So, as seen in the darker large circular areas in the inlet vorticity contour of Figure 3.10(b), the generations of smaller THV’s are not created inside as many of the largest THV’s which does not allow for the same sort of fine tuning of the energy spectrum as for the homogeneous turbulence. This causes the general under-prediction of the peaks of the energy spectra. Looking back again to Figure 3.10 and considering the larger THV present towards the middle top of the contour, these larger THV’s match the Reynolds stresses away from the walls, but can contribute disproportionately to energy at the lowest wavenumbers. This is why the spectra of the synthetic fluctuations levels off at the lowest wavenumbers.
Turbulent kinetic energy spectra, captured at the inlet and at three different heights from the wall, compared with the DNS data collected by Moser et al. [105]. (blue) transverse; (red) longitudinal; (--) DNS transverse; (--) DNS longitudinal.

The development of the friction coefficient at the walls is presented in Figure 3.15. Results from the THV SEM are compared with data reported by Jarrin et al. [47] using the original SEM (OSEM), Poletto et al. [125] using a divergence-free SEM (DFSEM), and Skillen et al. [148] using a SEM with improved eddy positioning and amplitude calculation (SSEM). The THV SEM shows the same spike in friction coefficient as the DFSEM and the SSEM immediately downstream of the inlet as the synthetic fluctuations begin to interact with the wall. $C_F$ then quickly settles to within $\pm 2\%$ of the fully-developed value by $2\delta$ downstream. The THV SEM shows a significant reduction in the development distance over the DFSEM and a similar development distance with the SSEM.

Vertical profiles of the turbulent kinetic energy and Reynolds shear stress at three streamwise locations are plotted in Figure 3.16. Both the TKE and Reynolds shear stress have recovered their given values away from the wall ($-0.5\delta < y < 0.5\delta$) by one channel height downstream of the inlet ($x = 2\delta$). The decrease in both quantities near the wall is

Distribution Statement A. Approved for public release; distribution is unlimited.
Downstream development of the friction coefficient compared with the DNS data collected by Moser et al. [105] and three different synthetic eddy methods: black) THV SEM; blue) Original SEM (OSEM) of Jarrin et al. [47]; red) Divergence-Free SEM (DFSEM) of Poletto et al. [125]; yellow) SEM (SSEM) of Skillen et al. [148]; - - -) DNS of Moser et al. [105].

because the grid in the streamwise direction is too coarse to properly resolve the convection of the smaller THV’s in the fourth and fifth THV generations. Returning to Figure 3.10, the larger THV’s away from the wall do influence this nearer-wall region; which is why the TKE and Reynolds shear stress both decrease to values consistent with the Reynolds stresses at the center of the larger THV’s.

Two-point streamwise spatial velocity correlations at two different heights from the wall, calculated from the inlet, are presented in Figure 3.17. The influence of stretched THV’s defined in Section 3.3.9 can clearly be seen in the differences between the longitudinal correlation near the wall in Figure 3.17(a) and near the height where the stretched THV’s are clustered in Figure 3.17(b). Close to the wall, only the smaller spherical THV’s...
Vertical profiles at three different streamwise locations compared with the DNS data collected by Moser et al. [105] of: a) turbulent kinetic energy; b) Reynolds shear stress; blue) $x = 0$ (inlet); red) $x = 2\delta$; yellow) $x = 4\delta$; - - -) DNS data from Moser et al. [105].

are imposed. Although those small THV’s give good agreement at distances less than 0.5$\delta$ from the inlet, their spherical nature is unable to model the larger downstream coherence. This is in stark contrast to the longitudinal correlation 0.5$\delta$ away from the wall in Figure 3.17(b). The spatial correlation is in agreement much farther downstream. This is because the stretched THV’s are being created at the inlet at this general height. The stretched THV’s introduce larger correlated scales which provides a better model of the physical structures.

The downstream evolution of the two-point spanwise transverse spatial velocity correlation is shown in Figure 3.18 and compared with DNS data from Moser et al. [105] at two representative heights. Again, at the inlet, the THV’s are well correlated at small distances, but they are unable to reproduce the larger negative correlations. As the syn-
Figure 3.17

Two-point streamwise spatial correlations, captured at two different heights from the wall, compared with the DNS data collected by Moser et al. [105]. blue) transverse; red) longitudinal; - - -) DNS transverse; -.-.-) DNS longitudinal.

The synthetic fluctuations are transformed by the Navier-Stokes equations, the correct negative correlations quickly develop by $2\delta$ downstream of the inlet. This rapid development of the physical coherence of the synthetic fluctuations agrees with the rapid settling of the friction coefficient, the turbulent kinetic energy, and the Reynolds shear stress.

### 3.4.4 Turbulent Mixing Layer

The LES of a spatially developing turbulent mixing layer was undertaken and compared with the experimental data collected by Mehta [95]. In the wind tunnel experiments of Mehta [95], the mixing layer is formed from the merging of two streams at the trailing edge of a sharp splitter plate. The boundary layers on both sides of the splitter plate were
Downstream evolution of the two-point spanwise spatial correlations, captured at two different heights from the wall, compared with the DNS data collected by Moser et al. [105]. blue) $x = 0$ (inlet); red) $x = 2\delta$; yellow) $x = 4\delta$; - - -) DNS transverse.

tripped in order to induce turbulent flow. Given a high enough Reynolds number and long enough development length, the turbulent mixing layer will yield a self-similar flow [160].

In this work, the inlet of the numerical domain was taken to be immediately downstream of the trailing edge of splitter plate (the splitter plate was not simulated). Mean velocity and Reynolds stress profiles from the DNS of a flat plate turbulent boundary layer at $Re_\theta = 1100$ conducted by Jiménez et al. [51] were imposed on the upper half of the inlet plane. This corresponds to the high speed turbulent boundary layer flow coming of the top of the splitter plate in the experiments of Mehta [95]. Those same DNS mean velocity and Reynolds stress profiles were mirrored about the streamwise coordinate direction, scaled in magnitude, and imposed on the lower half of the inlet plane to mimic the low speed turbulent boundary layer flow coming of the bottom of the splitter plate. The dotted lines in

Figure 3.18
Figures 3.20 and 3.21 correspond to the inlet profiles constructed from the DNS boundary layer data. The freestream velocity ratio, \( U_{\text{high}}/U_{\text{low}} \), was equal to 2 and the ratio of the Reynolds stresses, \( \langle u'_i u'_j \rangle_{\text{high}}/\langle u'_i u'_j \rangle_{\text{low}} \), was 4. Due to the contrived nature of the overall inlet profiles and the strong dependence of the initial development of the mixing layer on the inlet conditions [94], only the results at the inlet plane (the synthetic turbulence) and in the far downstream self-similar region were quantitatively examined.

The dimensions of the domain were \( 48\delta_{BL} \times 12\delta_{BL} \times 6\delta_{BL} \) in the streamwise, vertical, and spanwise directions, where \( \delta_{BL} \) is the height of the boundary layers, which equals 0.86 cm [95]. A uniform Cartesian grid consisting of \( 720 \times 180 \times 96 \) grid points was used. The increased artificial viscosity region was added for the last \( 12\delta_{BL} \) in the streamwise direction at the outflow. Periodic boundary conditions were applied in the spanwise direction and farfield boundary conditions were applied in the vertical direction. The freestream Mach number of the high speed boundary layer was equal to 0.1. 900 THV’s were imposed at any one instant in time on to the mean flow at the inflow boundary. The maximum radius allowed was \( 0.3\delta_{BL} \), while the minimum radius was \( 0.075\delta_{BL} \). The THV’s were broken into four generations, as given in Table 3.5, and imposed between a vertical height of \( -1.1\delta_{BL} \) and \( 1.1\delta_{BL} \).

Figure 3.19 presents isosurfaces of Q-criterion. As was seen in the homogeneous turbulence and the turbulent channel flow, the synthetic fluctuations quickly develop into realistic turbulent structures without any significant dissipation because there is some level of production from the mean shear. Moving downstream, the isosurfaces qualitatively shows the expected linear growth in vertical mixing layer thickness [13, 95].

Distribution Statement A. Approved for public release; distribution is unlimited.
Figure 3.19

Isosurfaces of Q-criterion for the turbulent mixing layer.
Vertical profiles of the mean streamwise velocity and the non-zero Reynolds stresses at the inlet plane are shown in Figures 3.20 and 3.21, respectively, along with the target profiles based on the DNS data of Jiménez et al. [51]. The target mean streamwise velocity profile was exactly reproduced including the zero velocity point at $y = 0$ corresponding to the stagnation point at the trailing edge of the splitter plate. Excellent agreement in the Reynolds stresses is also seen away from the centerline ($y = 0$). As with the turbulent channel flow, increased grid resolution and smaller THV’s near the centerline would allow for better reproduction of the Reynolds stresses in that area. The effect of restricting the location of the THV’s between a vertical height of $-1.1\delta_{BL}$ and $1.1\delta_{BL}$ can clearly been seen in the $\langle v'v' \rangle$ and $\langle w'w' \rangle$ profiles. Above the boundary layer height, the Reynolds stresses quickly go to zero because no THV’s are being imposed in that region.

Figures 3.22 and 3.23 present vertical profiles of the mean streamwise velocity and non-zero Reynolds stresses, respectively, sampled far downstream of the inlet ($x = 34\delta_{BL}$)
Vertical profile of the mean streamwise velocity at the inlet. blue) THV SEM; - - -) target value based on DNS data from Jiménez et al. [51]. and compared with the experimental data collected by Mehta [95]. The non-dimensional vertical coordinate is defined by the similarity parameter $\xi$.

$$\xi = \frac{y - y_0}{\delta}$$  \hspace{1cm} (3.73)

$y_0$ is the vertical location of the centerline of the mixing layer at the specific streamwise location and $\delta$ is the mixing layer thickness at that same streamwise location. The velocity and Reynolds stresses are non-dimensionalized using the difference in the freestream velocities, $\Delta U_\infty = U_{\text{high}} - U_{\text{low}}$. Both the mean velocity, Figure 3.22, and the Reynolds stresses, Figure 3.23, show very good agreement with the self-similar experimental results of Mehta [95].
Figure 3.21

Vertical profiles of the non-zero Reynolds stresses at the inlet. Blue) THV SEM; - - - target values based on DNS data from Jiménez et al. [51].
Vertical profile of the mean streamwise velocity far downstream of the inlet \((x = 34\delta_{BL})\) compared with the experimental data collected by Mehta [95]. \(U_c\) is the convection velocity \((U_{\text{high}} + U_{\text{low}})/2\). blue) THV SEM; O) experimental; - - -) error function, \(erf(\xi)/2\).

3.5 Distorted Triple Hill’s Vortex

The Triple Hill’s Vortex is a symmetric eddy that, as was seen in the homogeneous turbulence case in Section 3.4.2, produces a field of synthetic fluctuations with zero skewness. To introduce non-zero skewness into the final synthetic turbulence field, each of the three Hill’s spherical vortices are first distorted as follows before being combined together to form the Distorted Triple Hill’s Vortex. The only difference between the definition of the Distorted Triple Hill’s Vortex and the undistorted Triple Hill’s Vortex defined in Section 3.3.2 is the added distortion of the base component Hill’s vortices.
Figure 3.23

Vertical profiles of the non-zero Reynolds stresses far downstream of the inlet ($x = 34\delta_{BL}$) compared with the experimental data collected by Mehta [95]. (blue) THV SEM; (O) experimental.
3.5.1 Distortion of a Hill’s Vortex

The distortion of the Hill’s spherical vortex is performed on the streamfunction and the velocity components are then determined from equation (3.32) which will guarantee the divergence-free condition. Rosales and Meneveau [132][133] used a minimal Lagrangian map (also known as a naive Lagrangian map in the terminology introduced by Bec and Frisch [11] or Frisch et al. [37]) to introduce skewness in a synthetic isotropic turbulent field with Gaussian distribution and a prescribed energy spectrum. The minimal Lagrangian map is generalized here as

\[ \mathcal{L}_y : x \mapsto x'; \quad x'(x, t) = x + \mathcal{F}(x, t) \]  

(3.74)

where \( x = (x, y, z) \), \( x' = (x', y', z') \), \( \mathcal{F} = (\mathcal{F}_x, \mathcal{F}_y, \mathcal{F}_z) \) (in particular, for the minimal Lagrangian map, \( \mathcal{F}(x) = V_0(x) t \)); \( t \) is a parameter (associated with time), and \( V_0 = (u_0, v_0, w_0) \) is the initial velocity of the Lagrangian particle which should be constant along the path. In polar coordinates, the generalized map is defined as \( x'(r, z) = x + \mathcal{F}(r, z) \), where \( x = (r, z) \), \( x' = (r', z') \) and \( \mathcal{F} = (\mathcal{F}_r, \mathcal{F}_z) \). By an appropriate choice of functions \( \mathcal{F}_r \) and \( \mathcal{F}_z \) the spherical vortex may be deformed to introduce skewness. The following functions were utilized,

\[ z'(r, z) = z + \beta_z (z - z_0) \exp \left( -\frac{(z - z_0)^2}{2\sigma_z^2} \right) \]  

(3.75)

and

\[ r'(r, z) = r - \beta_r r \exp \left( -\frac{r^2}{2\sigma_r^2} \right) \exp \left( -\frac{(z + z_0)^2}{2\sigma_z^2} \right) \]  

(3.76)

where \( \beta_r, \beta_z, \sigma_r, \sigma_z \), and \( z_0 \) are parameters that depend on the size of the vortex and are associated with the “intensity” of the distortion.

Distribution Statement A. Approved for public release; distribution is unlimited.
Figure 3.24 shows the streamlines associated with a single Hill’s vortex with its axis in the vertical direction. Pictured in Figure 3.24(a) is an undistorted Hill’s spherical vortex, while Figure 3.24(b) depicts a distorted Hill’s vortex after the mapping given by equations (3.75) and (3.76) was applied to the streamfunction.

![Figure 3.24](image-url)

Contour of the streamlines of a single Hill’s vortex: a) without distortion; b) with distortion.

### 3.5.2 Results

Two test cases were considered: convection of an isolated Triple Hill’s Vortex and homogeneous isotropic turbulence.

#### 3.5.2.1 Single Triple Hill’s Vortex

The convection of a single Distorted THV was investigated. The domain and flow conditions are identical to convection of a single undistorted THV case in Section 3.4.1.
As a note, the contours from Section 3.4.1 for an undistorted THV are reproduced here for ease of comparison.

Contours of the velocity magnitude of a single undistorted (top) and distorted (bottom) THV are shown in Figure 3.25. In comparing the top and bottom row, the distortion has modified the shape of the THV while preserving the amplitude. This was as expected. The clearest connection back to the theoretical distortion of a single Hill’s vortex in Figure 3.24 can be seen in the yz-plane contours. The spherical shape of the undistorted THV in the left column is distorted to become the mushroom-like shape of the distorted THV in the right column, just like the change in shape of the streamlines between Figure 3.24(a) and Figure 3.24(b). The distortion can still clearly be seen in the xy- and xz-plane contours, but the circular to mushroom-like shape change is less apparent because of the stretching by the convecting flow.

Figure 3.26 depicts the generation in time of a single undistorted (top) and distorted (bottom) THV from the inlet. Each of the frames is a constant time step apart. Notice that distorted THV exhibits the same clean release from the inlet plane and lack of spurious waves surrounding the eddy that was seen with the undistorted THV. This indicates that the distortion of the Hill’s vortices is consistent with the divergence-free condition.

3.5.2.2 Homogeneous Turbulent Flow

Large eddy simulation of the spatial decay of isotropic turbulence is considered. The problem set up is similar to Section 3.4.2 with the goal of investigating whether skewness is introduced at the inlet through the use of distorted THV’s. The simpler (but much
Figure 3.25

Contours of the velocity magnitude on xy (left), xz (middle), and yz (right) planes through the center of an undistorted THV (top) and a distorted THV (bottom).

A (less dynamic) proportionally controlled amplitude matching method (see Appendix D) is utilized instead of the Modification of the Target Reynolds Stresses method presented in Section 3.3.7.

The probability distribution function (PDF) calculated along the homogeneous $y$ and $z$ directions at the inlet are displayed in Figure 3.27. In the same figure, the Gaussian function, which is representative to the normal distribution, is also plotted to show the skewness of the imposed synthetic turbulence field. The asymmetry seen in the PDF’s is a consequence of the distortion of the Hill’s vortices. The skewness of the longitudinal velocity gradient at the inlet was calculated to be $-0.4$. This is in contrast to the zero skewness seen at the inlet in Figure 3.7(b).

Distribution Statement A. Approved for public release; distribution is unlimited.
Figure 3.26

A single undistorted THV (top) and a single distorted THV (bottom) being generated at the inlet. Time is increasing with a constant time step from a) to d) and the inlet is on the left side of each frame.

Figure 3.27

Probability density function at the inlet calculated along the homogeneous: a) $y$-direction; b) $z$-direction; O) numerical; ——) Gaussian.
3.6 Conclusions and Future Work

The Triple Hill’s Vortex was proposed as a combination of three Hill’s vortices with their axes perpendicular to each other. The proposed new synthetic eddy was applied in the framework of the synthetic eddy method in order to generate synthetic turbulent inflow velocity fields that satisfied the divergence-free condition and matched given Reynolds stress profiles.

Simulation of homogeneous isotropic turbulence, turbulent channel flow, and a turbulent mixing layer were all able to reproduce given Reynolds stress tensors. The transition from artificial to realistic turbulence in the proximity to the inflow boundary was found to be small in all test cases that were considered. Excellent agreement between the turbulent kinetic energy spectrum and the two-point spatial correlations were found for the homogeneous case both at the inlet and far downstream. The spatial decay of the turbulent kinetic energy for isotropic homogeneous turbulence was shown to be in agreement with experimental data of isotropic turbulence in a wind tunnel and the synthetic fluctuations quickly developed skewness downstream of the inlet. For the channel flow, Reynolds stress profiles taken from DNS data were able to be reproduced away from the wall, but the distribution of THV’s was not sufficient to provide good agreement very close to the walls. The recovery of the friction coefficient was shown to be as just quick as for one of the latest synthetic eddy methods. Although good agreement with the DNS data was found at the inlet for the spanwise two-point correlations, recovery of the correct correlations occurred quickly downstream. For the turbulent mixing layer, the synthetic fluctuations transformed into
realistic turbulence quickly enough to capture the self-similar flow region far downstream of the inlet.

Finally, the distorted Triple Hill’s Vortex was proposed as a combination of three distorted Hill’s vortices. The distortion of the streamfunction of the Hill’s spherical vortex was realized by means of a coordinate mapping in order to introduce skewness. The distorted THV was then applied within the framework of a simplified THV SEM for the simulation of spatially decaying isotropic turbulence. In this test, skewness introduced by the distortion of the Triple Hill’s Vortices was observed in the probability density functions calculated at the inlet.

For the future, the THV SEM needs to be expanded to allow for the use of a wider range of arbitrary turbulent statistics as inputs. This is also true of most synthetic eddy methods in general. The near-wall stretching, uniform division of the total TKE across the THV generations, and distortion of the component Hill’s vortices all showed that rudimentary indirect control over the spatial correlations, energy spectra, and higher order statistics can be exerted. Through a systematic approach, general relationships might be able to be formulated to connect arbitrary energy spectra to how the target TKE for each THV generation is allocated, connect arbitrary spatial correlations to the stretching of the THV’s, and connect arbitrary higher order statistics (for example, skewness or flatness) to the multiple parameters present in the distortion method. The intrinsic coherence of a synthetic eddy, where all of the resolved scales smaller than the size of the eddy are contained within the eddy, introduces an additional layer of complexity that could hinder the formulation of the general relationship described above. Combining the synthetic eddies with an other class

Distribution Statement A. Approved for public release; distribution is unlimited.
of synthetic turbulence method, for example filtering the field of synthetic eddies in the spirit of the Digital Filtering Methods or generating the synthetic eddies in Fourier space, may offer an alternate development path towards increasing the direct control exerted over the desired synthetic fluctuations.
CHAPTER 4
CONTROL FORCED CONCURRENT PRECURSOR METHOD

This chapter deals with the development of a new turbulent inflow generation method in the class of precursor methods. The fluctuations imposed with this concurrent precursor method are determined through the solution of the Navier-Stokes equations, whereas in the previous chapter, the fluctuations imposed at the inlet represented an artificial model of realistic turbulence. Through the addition of controlled forcing planes, some degree of control is exerted over the imposed Reynolds stresses.

Before the Control Forced Concurrent Precursor Method (CFCPM) is described, the governing equations and numerical algorithm are discussed. Results from the simulation of high and low Reynolds number turbulent boundary flows are then presented, followed by conclusions and an outline for the future development of the method.

4.1 Governing Equations

The governing equations employed are the filtered continuity and momentum equations for incompressible flow. Included in the momentum equations are the immersed boundary method, controlled forcing, and concurrent precursor method source terms.

\[
\frac{\partial \tilde{u}_i}{\partial x_i} = 0
\]  

113

Distribution Statement A. Approved for public release; distribution is unlimited.
\[
\frac{\partial \tilde{u}_i}{\partial t} + \tilde{u}_j \frac{\partial \tilde{u}_i}{\partial x_j} = -\frac{\partial \tilde{p}^*}{\partial x_i} - \frac{\partial \tilde{\tau}_{ij}}{\partial x_j} + \frac{1}{Re} \frac{\partial^2 \tilde{u}_i}{\partial x_j^2} + \begin{cases} 
(F_M)_i + (F_F)_i & \text{; precursor} \\
(F_{CPM})_i + (F_{IBM})_i & \text{; main} 
\end{cases}
\]

Spatial filtering at scale \( \tilde{\Delta} \) is represented by tilde, \( \tilde{u}_i \) are the components of the velocity field corresponding to the streamwise \( x_1 \)-direction, spanwise \( x_2 \)-direction, and vertical \( x_3 \)-direction, respectively, and \( \tilde{p}^* \) is the effective pressure divided by the reference density. The SGS stress is given as \( \tau_{ij} = \tilde{u}_i \tilde{u}_j - \tilde{u}_i \tilde{u}_j \). The immersed boundary method force, \( (F_{IBM})_i \), is necessary because of the grid restrictions imposed by the pseudo-spectral method. There are three addition forcing terms associated with the CFCPM: the mean and fluctuating flow controlled forcing terms, \( (F_M)_i \) and \( (F_F)_i \), which maintain prescribed mean flow and turbulence levels in the precursor domain and the concurrent precursor method force, \( (F_{CPM})_i \), which forces the main flow to match the precursor flow in the precursor forcing region. By adding the forcing terms to the momentum equations, before the solution of the Poisson equation for pressure, the forced flow is automatically divergence-free.

### 4.1.1 Subgrid Scale Model

The SGS stress is modeled using the Lagrangian scale-dependent dynamic model developed by Bou-Zeid et al. \[19\]. Using an eddy-viscosity model, the SGS stress tensor is defined as

\[
\tau_{ij} = -2 \left( C_{s,\Delta} \right)^2 |\tilde{S}| \tilde{S}_{ij}
\]

where \( \Delta \) is the grid scale, \( C_{s,\Delta} \) is the Smagorinsky coefficient, \( \tilde{S}_{ij} \) is the resolved strain-rate tensor, and \( |\tilde{S}| \) is the strain-rate magnitude. The unknown Smagorinsky coefficient

Distribution Statement A. Approved for public release; distribution is unlimited.
is calculated using a relationship based on the Lagrangian-averaged SGS stresses at two different test filter scales.

The method for calculating the local grid-filter scale Smagorinsky coefficient, $C_{s,\Delta}$, begins with the assumption postulated by Porté-Agel et al. [122] that the coefficient has a power-law dependence as a function of scale.

$$C_{s,\alpha\Delta}^2 = C_{s,\Delta}^2 \alpha^\phi$$

Thus, the ratio of coefficients evaluated at filter levels $\alpha\Delta$ and $\Delta$ are equal to the ratio of coefficients evaluated at filter levels $\alpha^2\Delta$ and $\alpha\Delta$.

$$\frac{C_{s,4\Delta}^2}{C_{s,2\Delta}^2} = \frac{C_{s,2\Delta}^2}{C_{s,\Delta}^2}$$

The grid-filter level Smagorinsky coefficient can then be defined as a ratio of the $2\Delta$ test-filter level Smagorinsky coefficient and the coefficient $\beta$

$$C_{s,\Delta}^2 = \frac{C_{s,2\Delta}^2}{\beta}$$

where $\beta$ accounts for possible scale dependence and is defined as the ratio of the $4\Delta$ test-filter level Smagorinsky coefficient and the $2\Delta$ test-filter level Smagorinsky coefficient a

$$\beta = \frac{C_{s,4\Delta}^2}{C_{s,2\Delta}^2}$$

While $\beta$ can vary between zero and infinity, Bou-Zeid et al. [19] imposed a minimum limit of 0.125 to avoid numerical instabilities associated with $\beta$ tending towards zero. The coefficients at the $2\Delta$ and $4\Delta$ test-filter levels are determined using the same framework of the standard dynamic Smagorinsky proposed by Germano et al. [39] and modified by

Distribution Statement A. Approved for public release; distribution is unlimited.
Lilly [79]. The mean-square error between the resolved stress tensor and the difference between the SGS stress tensors at the test-filter level and the grid-filter level is minimized by the following

\[ C_{s,2\Delta}^2 = \frac{\langle L_{ij} M_{ij} \rangle}{\langle M_{ij} M_{ij} \rangle}; \quad C_{s,4\Delta}^2 = \frac{\langle Q_{ij} N_{ij} \rangle}{\langle N_{ij} N_{ij} \rangle} \] (4.8)

where the angle brackets, \( \langle \rangle \), denote spatial averaging in any homogeneous directions in order to limit numerical instabilities [39, 121]. The resolved stress tensors between the grid-filter level \( \Delta \) and 2\( \Delta \) test-filter level, \( L_{ij} \), and between the grid-filter level \( \Delta \) and 4\( \Delta \) test-filter level, \( Q_{ij} \), are defined using Germano’s identity and can be calculated using the resolved velocity field [40].

\[ L_{ij} = \overline{\tilde{u}_i \tilde{u}_j} - \overline{\tilde{u}_i \tilde{u}_j} \; ; \quad Q_{ij} = \hat{\overline{\tilde{u}_i \tilde{u}_j}} - \hat{\overline{\tilde{u}_i \tilde{u}_j}} \] (4.9)

As a note, a tilde (\( \tilde{\cdot} \)) represents filtering at the grid level \( \Delta \), a bar (\( \overline{\cdot} \)) represents filtering at the 2\( \Delta \) test-filter level, and a caret (\( \hat{\cdot} \)) represents filtering at the 4\( \Delta \) test-filter level. The differences between the SGS stress tensor at 2\( \Delta \) test-filter level and the grid-filter level \( \Delta \), \( M_{ij} \), and between the SGS stress tensor at 4\( \Delta \) test-filter level and the grid-filter level \( \Delta \), \( N_{ij} \), are defined as follows

\[ M_{ij} = 2\Delta^2 \left( |\overline{S}| \overline{S}_{ij} - 4\beta |\overline{S}| \overline{S}_{ij} \right) \; ; \quad N_{ij} = 2\Delta^2 \left( |\hat{S}| \hat{\overline{S}}_{ij} - 16\beta |\hat{S}| \hat{\overline{S}}_{ij} \right) \] (4.10)

where both test-filter levels and the grid-filter level SGS stress tensors are modeled using the original Smagorinsky model [149]. In the Lagrangian scale-dependent dynamic model, \( \beta \) in equation (4.10) is assumed to equal one, since the value will be corrected with the evaluation of equation (4.7).
Meneveau et al. [98] modified the dynamic model to account for general inhomogeneous flows by replacing the homogeneous spatial averaging with Lagrangian averaging in time along fluid-particle pathlines.

\[ C_{s,2\Delta}^2 = \frac{J_{LM}}{J_{MM}} ; \quad C_{s,4\Delta}^2 = \frac{J_{QN}}{J_{NN}} \]  

\[ (4.11) \]

The weighted backward time integrals,

\[ J_{LM}(x) = \int_{-\infty}^{t} L_{ij} M_{ij} (z(t'), t') W(t - t') dt' \]  

\[ J_{MM}(x) = \int_{-\infty}^{t} M_{ij} M_{ij} (z(t'), t') W(t - t') dt' \]  

\[ J_{QN}(x) = \int_{-\infty}^{t} Q_{ij} N_{ij} (z(t'), t') W(t - t') dt' \]  

\[ J_{NN}(x) = \int_{-\infty}^{t} N_{ij} N_{ij} (z(t'), t') W(t - t') dt' \]  

where \( z(t') \) are the previous positions of the fluid particles and \( W(t - t') \) is a weighting, can be replaced with forward relaxation transport equations by choosing a weighting function with an exponential form. These relaxation transport equations can then be discretized using first-order approximations in time and space to give the following expressions to update the integral terms from time-step \( n \) to \( n + 1 \).

\[ J_{LM}(x) = H [\epsilon_{2\Delta} (L_{ij} M_{ij})^{n+1} (x) + (1 - \epsilon_{2\Delta}) J_{LM}^n (x - \bar{u}^n \Delta t)] \]  

\[ J_{MM}(x) = \epsilon_{2\Delta} (M_{ij} M_{ij})^{n+1} (x) + (1 - \epsilon_{2\Delta}) J_{MM}^n (x - \bar{u}^n \Delta t) \]  

\[ J_{QN}(x) = H [\epsilon_{4\Delta} (Q_{ij} N_{ij})^{n+1} (x) + (1 - \epsilon_{4\Delta}) J_{QN}^n (x - \bar{u}^n \Delta t)] \]  

\[ J_{NN}(x) = \epsilon_{4\Delta} (N_{ij} N_{ij})^{n+1} (x) + (1 - \epsilon_{4\Delta}) J_{NN}^n (x - \bar{u}^n \Delta t) \]  

\[ (4.16) \quad (4.17) \quad (4.18) \quad (4.19) \]
x is the location of a grid point, \( \bar{u} \) is the resolve velocity at the grid point, \( \Delta t \) is the time step, \( H[\phi] \) is a ramp function used to ensure that errors introduced by discretizing the transport equations do not result in negative values of \( C_{s,\Delta}^2 \),

\[
H[\phi] = \begin{cases} 
\phi & ; \phi \geq 0 \\
0 & ; \phi < 0 
\end{cases} 
\tag{4.20}
\]

and time constants for the exponential weighting (\( \epsilon_{2\Delta} \) and \( \epsilon_{4\Delta} \)) are defined as

\[
\epsilon_{2\Delta} = \frac{\Delta t}{T_{2\Delta}^n} ; \quad T_{2\Delta}^n = 1.5\Delta \left( J_{LM}^n J_{MM}^n \right)^{-\frac{1}{8}} 
\tag{4.21}
\]

\[
\epsilon_{4\Delta} = \frac{\Delta t}{T_{4\Delta}^n} ; \quad T_{4\Delta}^n = 1.5\Delta \left( J_{QN}^n J_{NN}^n J_{LM}^n \right)^{-\frac{1}{8}} 
\tag{4.22}
\]

In summary, the Smagorinsky coefficient in equation (4.3) is calculated at every grid point and at every time step using the Lagrangian averaged quantities given by the following expression.

\[
C_{s,\Delta}^2 = \frac{C_{s,2\Delta}^2}{\max (\beta, 0.125)} = \frac{\frac{J_{LM}^n}{J_{MM}^n}}{\max \left( \frac{J_{QN}^n J_{MM}^n}{J_{NN}^n J_{LM}^n}, 0.125 \right)} 
\tag{4.23}
\]

### 4.1.2 Wall Model

Monin-Obukhov similarity theory is used to relate wall stress to the velocity at the first grid point away from a wall. Monin and Obukhov [102] used empirical data of wind velocities under various temperature stratifications to propose a correction to the logarithmic boundary layer model in order to account for nonuniform temperature distributions. While the similarity theory was developed for averaged quantities, Moeng [100] imposed the wall stress in a strictly local sense. Bou-Zeid et al. [19] then showed that by using local filtered

Distribution Statement A. Approved for public release; distribution is unlimited.
velocities, the local similarity formulation produces an average stress that is very close to the averaged similarity formulation. The wall shear stress components are defined as follows:

\[ \tau_{13}\big|_{z=0} = -u^2 \frac{\tilde{u}_i}{V_f} = -\left( \kappa V_f \ln \left( \frac{z}{z_0} \right) - \Psi_M \right)^2 \frac{\tilde{u}_i}{V_f} ; \quad i = 1, 2 \]  
\[ V_f = \sqrt{\left( \tilde{u}_1\big|_{z=\Delta z} \right)^2 + \left( \tilde{u}_2\big|_{z=\Delta z} \right)^2} \]  

where \( \tau_{13}\big|_{z=0} \) and \( \tau_{23}\big|_{z=0} \) are the instantaneous local wall stress components, \( u_r \) is the friction velocity, \( z_0 \) is the effective roughness length, \( \kappa \) is the von Kármán constant (taken to be \( \kappa = 0.4 \)), \( \Psi_M \) is the stability correction function for momentum, and \( V_f \) is the local filtered horizontal velocity magnitude at the first grid point. It should be noted that since the boundary layer flows considered in Section 4.4 do not consider temperature effects, the stability correction function is taken to equal to zero, \( \Psi_M = 0 \), thus returning the model to the traditional logarithmic law-of-the-wall.

### 4.1.3 Immersed Boundary Method

The presence of a body in the boundary layer flow is modeled using the direct forcing immersed boundary method (IBM) introduced by Mohd-Yusof [101]. In the direct forcing approach, a discrete force is added to the discretized momentum equations to drive the numerical solution towards the desired velocity of the body.

\[ \frac{u_i^{n+1} - u^n}{\Delta t} = RHS_i + (F_{IBM})_i \]  

Distribution Statement A. Approved for public release; distribution is unlimited.
The imposed force can then be calculated by specifying the velocity of the body, \((u_i)_{body}\).

\[
(F_{IBM})_i = \begin{cases} 
\frac{(u_i)_{body} - u^n_i}{\Delta t} - RHS_i &; \text{body points} \\
0 &; \text{fluid points}
\end{cases}
\] (4.27)

By substituting the imposed force back into the discretized momentum equations, it is shown that the desired velocity of the body is satisfied at each discrete point.

\[
u^n_{i+1} = (u_i)_{body}
\] (4.28)

Because the geometry of the body coincides with the grid points of the numerical mesh, use of one of the more sophisticated immersed boundary methods is not required. Kim and Choi [62] provide a review of the immersed boundary methods developed for more complex geometries and fluid-structure interaction problems.

Since this LES does not resolve the flow at the surface of the body, the flow at the walls of the box is modeled using the same Monin-Obukhov similarity theory as is used on the bottom boundary of the domain. Tseng et al. [161] has shown that this type of wall model produces satisfactory results in their study of three dimensional flow around square cylinders.

### 4.2 Numerical Algorithm

The numerical tool is a pseudo-spectral LES code that solves the filtered Navier-Stokes equations through the use of a pseudo-spectral horizontal discretization and a centered finite difference vertical discretization. The code was first developed by Albertson [3] and then improved and used extensively (Albertson and Parlange [4], Tseng et. al [161].
Calaf et al. [23], Calaf et al. [24], and Sescu and Meneveau [142]). The continuity equation is enforced through the solution of the Poisson equation resulting from taking the divergence of the momentum equation. An immersed boundary method is used to simulate bluff bodies and the new Control Forced Concurrent Precursor Method is used to impose turbulent inflow conditions.

### 4.2.1 Spatial Discretization

The horizontal directions employ a uniform discretization to accommodate the use of a pseudo-spectral approach [25]. The Fourier representation of a given variable is

\[
\phi(x, y, z) = \sum_{k_x} \sum_{k_y} \hat{\phi}(k_x, k_y, z)e^{i(k_x x + k_y y)}
\]

(4.29)

where \( \hat{\phi} \) is the complex Fourier amplitude associated with \( \phi \), and \( k_x \) and \( k_y \) are the wavenumbers in the horizontal directions. Through taking the derivatives of equation (4.29)

\[
\frac{\partial \phi(x, y, z)}{\partial x} = \sum_{k_x} \sum_{k_y} \left[ \hat{\phi}(k_x, k_y, z)(ik_x) \right] e^{i(k_x x + k_y y)}
\]

(4.30)

\[
\frac{\partial \phi(x, y, z)}{\partial y} = \sum_{k_x} \sum_{k_y} \left[ \hat{\phi}(k_x, k_y, z)(ik_y) \right] e^{i(k_x x + k_y y)}
\]

(4.31)

and noticing that \( \hat{\phi}(ik_x) \) and \( \hat{\phi}(ik_y) \) are the complex Fourier amplitudes associated with the \( \partial \phi/\partial x \) and \( \partial \phi/\partial y \), a procedure for calculating the horizontal derivatives can be formed.

At each horizontal plane,

1. Take the two-dimensional Fourier transform of \( \phi(x, y, z) \) to find \( \hat{\phi} \).
2. Multiply \( \hat{\phi} \) by \( ik_x \) or \( ik_y \)
3. Set \( \hat{\phi} = 0 \) for the zeroth wavenumber and the Nyquist wavenumber
4. Take the two-dimensional inverse Fourier transform of \( \hat{\phi} \) to find \( \partial \phi/\partial x \) or \( \partial \phi/\partial y \)

Distribution Statement A. Approved for public release; distribution is unlimited.
It is important to set the zeroth wavenumber and the Nyquist wavenumber complex amplitudes to zero because, at those wavenumbers, \( \hat{\phi} \) is real-valued. If those amplitudes are included, the resultant derivative after the inverse Fourier transform will be complex.

The corresponding aliasing errors that arise from the pseudo-spectral approach of calculating products between variables in physical space before taking the Fourier transform, as opposed to transforming each variable first and then performing a convolution in the Fourier domain, are corrected according to the \( 3/2 \) rule \[25\]. That means that the Fourier transforms are performed using \( 3N/2 \) points and then truncated to \( N \) modes.

In the vertical direction, a staggered uniform discretization is used where the vertical velocity is stored at points half a vertical grid spacing below the other variables. Derivatives in the vertical direction are approximated using a second order accurate central difference scheme and are stored at points halfway in between the location of the original variable.

\[
\frac{\partial \phi}{\partial z} = \frac{\phi(x, y, z + \frac{\Delta z}{2}) - \phi(x, y, z - \frac{\Delta z}{2})}{\Delta z} \tag{4.32}
\]

4.2.2 Time Marching

The time marching is performed using a fully-explicit second-order Adams-Bashforth scheme \[20\].

\[
\phi^{n+1} = \phi^n + \Delta t \left( \frac{3}{2} RHS_{\phi}^n - \frac{1}{2} RHS_{\phi}^{n-1} \right) \tag{4.33}
\]

\( \phi \) represents the velocity components.
4.2.3 Boundary Conditions

Under the hypothesis of horizontal homogeneity of turbulence in the boundary layer, periodic boundary conditions are imposed along both horizontal directions. However, the proposed concurrent precursor simulation provides inflow boundary conditions that are introduced at the end of the domain, while keeping the periodicity condition in the streamwise direction.

The vertical gradients of velocity and the vertical component of velocity must vanish at the top boundary which is located well above the boundary layer top.

Because of the vertically staggered discretization, the horizontal velocities do not receive formal no-slip boundary conditions. The horizontal velocities at the first point away from the wall ($z = \Delta z/2$) are set through the velocity gradients in the vertical direction calculated using the Monin-Obukhov similarity theory. The vertical velocity at the wall is set to zero.

4.3 Proposed Precursor Method

The Control Forced Concurrent Precursor Method is a combination of a traditional periodic concurrent precursor method with controlled forcing methods to allow for the imposition of given mean flow profiles and anisotropic Reynolds stress tensors. A region of the precursor flow field, outlined in red in Figure 4.1, is transferred to main domain through a forcing region, the red area in Figure 4.1, that penalizes the difference between the two flows. By applying the controlled forcing before the solution of the Poisson equation for pressure, the forced flow is automatically divergence-free.
4.3.1 Concurrent Precursor Method

The concurrent precursor method introduced by Stevens et al. [156] and modified by Munters et al. [108] was specifically developed to provide inflow conditions for periodic domains, although it is not limited to periodic domains. A precursor domain is considered with identical dimensions and discretization as the main domain. The two simulations are carried out at the same flow conditions and are synchronized in time. Body forces, added to each of the momentum equations over a region of the main domain, force the main flow towards the precursor flow. They are defined as follows

\[ (F_{CPM})_i(x, y, z, t) = \sigma(x) \left[ u^{pre}_i(x, y, z, t) - u^{main}_i(x, y, z, t) \right] \]  

(4.34)

\[ \sigma(x) = \begin{cases} 
\sigma_{max} \left( \frac{x - x_s}{x_{pl} - x_s} \right)^n & ; x_s \leq x \leq x_{pl} \\
\sigma_{max} & ; x_{pl} < x \leq L_x 
\end{cases} \]  

(4.35)
where $\sigma$ is the precursor forcing strength, $x_s$ is the streamwise location of the start of the precursor forcing region and $x_{pl}$ is the streamwise location of the end of the increasing precursor forcing strength region. $x_{pl}$ and the exponent $n$ determine the smoothness of the transition and $\sigma_{max}$ is the maximum precursor forcing strength. The parameters used in this work are as follows:

$$
x_s = 0.8L_x \\
x_{pl} = 0.99L_x \\
n = 5 \\
\sigma_{max} = \frac{0.7}{\Delta t}
$$

where $L_x$ is the length of the flow domain and $\Delta t$ is the time step. The precursor forcing region can be located anywhere in the main domain where “inflow” conditions are desired.

The precursor and main domains are not required to have the same dimensions, grid resolutions, and/or flow conditions. These differences between the two domains introduce various complexities that must then be addressed (e.g., proper interpolation needs to be in place). Streamwise periodicity in the main domain can be introduced by using a precursor domain with a smaller streamwise length [111]. In the case that the precursor and main domains do not share the same discretization, interpolation would be required to transfer the precursor flow to the main domain. For flow conditions that differ between the precursor and main domains, the precursor flow would also need to be rescaled to match the flow conditions in the main domain before it is transferred.
4.3.2 Fluctuating Flow Controlled Forcing

The framework of the fluctuating flow controlled forcing is rooted in the original method Spille-Kohoff and Kaltenbach [155] introduced to accelerate the development of wall-bounded turbulence by increasing the production of Reynolds shear stress. This increase in production was achieved by adding body forces to the wall-normal momentum equation on planes normal to the flow that amplified the existing wall-normal velocity fluctuations. The amplitudes of the body forces were defined using a proportional-integral (PI) controller based on the error between given and calculated Reynolds shear stress profiles.\(^1\)

The new fluctuating flow controlled forcing extends the original method introduced by Spille-Kohoff and Kaltenbach [155] to two and three dimensions and moves the calculation of the forces into the local principal-axis coordinate system in order to match 2D and 3D anisotropic Reynolds stress tensors. The fluctuating flow controlled forcing method only modifies fluctuations that already exist. It does not generate its own fluctuations. These existing velocity fluctuations are modified through body forces, \((F_F)_i\), added to the momentum equations on planes normal to the streamwise direction. The body forces are defined in the local principal-axis coordinate system as follows

\[
(F_F)_i^p(x_f, y, z, t) = r_i^p(x_f, y, z, t) \left[ u_i^p(x_f, y, z, t) - \langle u_i^p \rangle(x_f, y, z, t) \right]
\]  

(4.38)

where \(x_f\) are the streamwise locations of the fluctuating flow controlled forcing planes, \(r_i^p\) are the amplitudes of the forces, \(u_i^p\) are the instantaneous velocities at time \(t\), and \(\langle u_i^p \rangle\) are the mean velocities at time \(t\). The superscript \(^p\) denotes variables in the local principal-axis coordinate system.

\(^1\)For purposes of this discussion, the wall-normal direction is the \(z\)-direction and the wall-normal velocity is the \(w\) velocity.
coordinate system. The amplitudes of the forces are defined using a PI controller based on
the error between the given and calculated principal-axis Reynolds stress profiles.

\[
\begin{align*}
    r_i^p(x_f, y, z, t) &= \alpha_i e_i^p(x_f, y, z, t) + \beta_i \int_0^t e_i^p(x_f, y, z, t') \, dt' \\
    e_i^p(x_f, y, z, t) &= \langle u'_i u'_i \rangle_{\text{given}}(x_f, y, z, t) - \langle u'_i u'_i \rangle^p(x_f, y, z, t)
\end{align*}
\] (4.39) (4.40)

\(\alpha_i\) and \(\beta_i\) are chosen such that the error decreases sufficiently fast without introducing
numerical instabilities. In this work, \(\alpha_i = 1\) and \(\beta_i = 100\). \(\langle u'_i u'_i \rangle_{\text{given}}\) and \(\langle u'_i u'_i \rangle^p\) are the
given and current principal-axis Reynolds stress profiles. Using the transformation matrix
created from the eigenvectors of the local Reynolds stress tensor, \(T_p^G\), the body forces
are transformed from the local principal-axis coordinate system to the global coordinate
system before being applied to the momentum equations.

\[
(F_F)_i = (T_p^G)_{ij} (F_F)_j^p
\] (4.41)

The following criteria for the application of the body forces ensure that the more en-
ergetic fluctuations are amplified consistent with the sign of cross-velocity correlations,
while unrealistically large fluctuations are not amplified [56, 155].

\[
|u'_i| < 0.4 U_\infty
\] (4.42)

\[
|u'_i u'_3| > 0.0015 U_\infty^2
\]

If these criteria are not met,

\[
(F_F)_i(x_f, y, z, t) = 0
\] (4.43)
Keating et al. [56] suggested using an exponential weighted moving average to calculate the current time-averaged quantities,

$$\langle \phi \rangle (t + \Delta t) = \phi (t) \frac{\Delta t}{T_{avg}} + \left(1 - \frac{\Delta t}{T_{avg}}\right) \langle \phi \rangle (t)$$  (4.44)

where $\phi$ is any quantity needing time-averaging, $\Delta t$ is the time step, and $T_{avg}$ is the averaging time period. In this work, $T_{avg}$ is set to two flow-throughs.

For a 2D Reynolds stress tensor, a force is not applied to the fluctuating velocity component where the associated Reynolds stresses are not known. If only the normal Reynolds stresses are known, the proposed method reduces to applying the original controlled forcing method to each velocity component independently. This method is also not restricted to a certain type of flow or numerical method. In principle, the controlled forcing method allows for time-varying Reynolds stresses. The force applied at a point is only based on the current and given Reynolds stresses. For flows that involve faster varying Reynolds stresses, care needs to be taken on how the current Reynolds stresses are determined such that a proper average can be calculated.

4.3.3 Mean Flow Controlled Forcing

The imposition of given mean flow profiles is handled in the same manner as Schlüter et al. [137]. Body forces, $(F_M)_i$, are added to the momentum equations in the precursor domain on planes normal to the streamwise direction in order to drive the mean flow towards a given mean profile. For a constant density flow, these body forces take the form of the following proportional controller

$$(F_M)_i (x_m, y, z, t) = \gamma_i \left[ \langle u_i \rangle_{\text{given}} (x_m, y, z, t) - \langle u_i \rangle (x_m, y, z, t) \right]$$  (4.45)
where \( x_m \) are the streamwise locations of the mean flow forcing planes, \( \gamma_i \) are the mean flow forcing strength factors, \( \langle u_i \rangle_{\text{given}} \) are the given mean velocity profiles, and \( \langle u_i \rangle \) are the current mean velocity profiles at time \( t \) calculated using the exponential weighted moving averaging shown in Section 4.3.2. The mean flow forcing strength factor should be large enough that the body forces promptly respond to changing flow conditions while also not being too large as to introduce numerical instabilities. In this work, \( \gamma_i = 0.7 \). In the same manner as the fluctuating flow controlled forcing, the given mean velocities are allowed to be time-varying with the same caveat when calculating the current mean velocity for faster varying flows.

### 4.4 Results

Two validation cases are considered: a high and low Reynolds number turbulent boundary layer. The high Reynolds number case utilizes 3D Reynolds stress tensor profiles, while the low Reynolds number case only prescribes 2D tensor profiles. The objectives of these numerical simulations are to impose a experimentally measured turbulent boundary layer and then validate the LES results with the experimental results. For both cases, the proposed fluctuating flow controlled forcing method is compared with simulations without controlled forcing and with only the original controlled forcing of Spille-Kohoff and Kaltenbach [155]. Additionally, the ability of the proposed method to match the given Reynolds stresses is evaluated by running the precursor simulation at three different grid resolutions.
4.4.1 High Reynolds Number Turbulent Boundary Layer

A high Reynolds number boundary layer flow developing over a 0.2 m wall-mounted cube is considered at $Re_\theta = 3.0 \times 10^5$. This was the subject of the wind tunnel experiment of Castro and Robins [21]. The streamwise, spanwise, and vertical dimensions of domain are $1.92 \times 0.8 \times 2.7$ m, where the height of both the domain and the cube are equal to the height of the wind tunnel test section and the experimental cube. The precursor and main domains are shown in Figure 4.2, where the blue line represents the controlled forcing plane. The precursor and main domain are both periodic and use the same grid resolution. The three uniform discretizations considered are shown in Table 4.1. The three fluctuating forcing methods (none, original, and proposed) are compared using the fine grid.

The physical dimensions of the domain used for comparison to the experimental results of Castro and Robins [21]. The blue line represents the single controlled forcing plane.

A turbulent boundary layer of thickness 2.0 m with a freestream velocity of 2.02 m/s is imposed in the precursor simulation by a single controlled forcing plane using mean

Distribution Statement A. Approved for public release; distribution is unlimited.
streamwise velocity and complete Reynolds stress tensor profiles given by Castro and Robins [21]. For the original forcing method, only $\langle w'w' \rangle$ is used as a target. The initial fluctuations are provided by a random field of white noise scaled to match the Reynolds stress tensor profiles. The simulations comparing the forcing methods were all run until the first method converged to the desired Reynolds stresses, within 250 flow-throughs; then statistics were collected for 30 flow-throughs.

Table 4.1

Grid point dimensions of the high Reynolds number turbulent boundary layer domain.

<table>
<thead>
<tr>
<th>Case</th>
<th>$N_x \times N_y \times N_z$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>64 $\times$ 32 $\times$ 64</td>
</tr>
<tr>
<td>Medium</td>
<td>128 $\times$ 48 $\times$ 128</td>
</tr>
<tr>
<td>Fine</td>
<td>192 $\times$ 64 $\times$ 324</td>
</tr>
</tbody>
</table>

The mean streamwise velocity and non-zero Reynolds stress profiles sampled at the location of controlled forcing plane are plotted in Figure 4.3. These profiles were collected from the precursor simulation using the proposed forcing method for three different grid resolutions and compared with the experimental data from Castro and Robins [21]. The proposed forcing method was able to successfully match the target Reynolds stress profiles for all three grid resolutions. This is not surprising because the controlled forcing method operates on each discrete grid point on the controlled forcing plane independently. The forces associated with a point on the forcing plane are only controlled by the local velocity fluctuations and target values at that point.

Distribution Statement A. Approved for public release; distribution is unlimited.
Figure 4.3

Turbulent statistics along the vertical direction from the precursor domain, using the proposed forcing method and three grid resolutions, sampled at the controlled forcing plane at conditions matching the momentum thickness Reynolds number of Castro and Robbins [21]. - - -) coarse; - - -) medium; —) fine; X) experimental data.

Contour plots of the instantaneous streamwise velocity in the precursor and main domain for the three different forcing cases are presented in Figures 4.4 and 4.5. For all three forcing cases, the flow structures immediately upstream of the outlet and downstream of the inlet are extremely similar when comparing their respective main and precursor domains. Also notice how the wake created by the cube in the main domain is smoothly transformed by the precursor forcing region into the precursor flow field. When comparing the different controlled forcing methods present in the precursor simulations, the level of turbulence present in the precursor simulations increases moving from the “no forcing” case on the left to the proposed forcing case on the right. Through the addition of controlled forces on more of the fluctuating velocity components, the development of the turbulent boundary layer is accelerated. It is worth noting that the controlled forcing plane (the black line on the precursor contours) only affects the intensity of turbulent structures as it passes through...
it, and does not change the shape of the structures. In the main domain contours, the box wakes are all similar in shape below the height of the box.

![Figure 4.4](image)

**Figure 4.4**

Contour of the instantaneous streamwise velocity normalized by the freestream velocity on an xz plane through the center of the precursor domain, at conditions matching the momentum thickness Reynolds number in Castro and Robins [21]. The black line represents the controlled forcing plane. left) “no forcing”; middle) “original forcing”; right) proposed forcing

The mean streamwise velocity and non-zero Reynolds stress profiles from the precursor simulation are plotted in Figure 4.6, alongside the experimental data from Castro and Robins[21] at the location of controlled forcing plane. In the computational time it took for the proposed forcing method to grow the desired turbulent boundary layer and match all of the Reynolds stress profiles, the boundary layers in the “no forcing” and “original forcing” cases were still developing and hence had not matched the target Reynolds stresses yet. Without forcing, the turbulence produced at the bottom wall is left to naturally grow into the domain. This is evidenced by slightly larger Reynolds stresses close to the wall.
Contour of the instantaneous streamwise velocity normalized by the freestream velocity on an xz plane through the center of the main domain, at conditions matching the momentum thickness Reynolds number in Castro and Robins [21]. left) “no forcing”; middle) “original forcing”; right) proposed forcing

quickly decreasing to near zero moving towards the freestream, and the lack of turbulent structures seen away from the wall in the left contour of Figure 4.4. Because the original forcing method actively targeted the wall-normal Reynolds stress, it does show an increased level. The Reynolds shear stress also shows an increase, which is consistent with the primary function of the original controlled forcing method. These increases agrees with what is seen in the middle contour of Figure 4.4; turbulent structures are present, but they have a lower magnitude than the structures seen in the right contour of Figure 4.4. If the “no forcing” and “orginal forcing” simulations were allowed to continue integrating the governing equations, it is expected that the desired turbulent boundary layer would eventually develop. The comparisons in figure Figure 4.6 shows that the proposed forcing
method reduces the development time of the turbulent boundary layer as compared to the “no forcing” and “original forcing” cases, thus reducing the overall computational cost.

Vertical profiles of the mean streamwise velocity and turbulence intensity from the main simulation are plotted in Figures 4.7 and 4.8 alongside the experimental data from Castro and Robins at three streamwise locations in the wake, $x/h_{box} = \{0, 1, 2\}$. Good agreement was found for the mean streamwise velocity for all three methods. In terms of the turbulence intensity, the proposed forcing method showed the best agreement. The other two forcing methods both underpredicted the turbulence intensity above the height of the box, which is consistent with the low intensity turbulence in their precursor simulations. The effect of the “original forcing” method is seen in the increased turbulence intensity as compared to the “no forcing” case. It is worth noting that below the height and at one box.
height downstream of the cube all three cases give reasonable intensities, which supports
the similar wake shape seen in the contours of Figure 4.5. This suggests that immediately
behind the cube, the presence of the cube and the correct mean flow are more significant
than the correct levels of freestream turbulence.

\[
\begin{align*}
x/h_{\text{box}} = 0 & \quad x/h_{\text{box}} = 1 & \quad x/h_{\text{box}} = 2 \\
\end{align*}
\]

\[
\begin{align*}
\langle u \rangle / U_\infty \\
\end{align*}
\]

Mean streamwise velocity along the vertical direction sampled at streamwise locations
downstream of the cube. - - - - ) no forcing; - - - ) original forcing; — - ) proposed forcing;
X) experimental data for the high Reynolds number case of Castro and Robbins [21].

4.4.2 Low Reynolds Number Turbulent Boundary Layer

A low Reynolds number turbulent boundary layer flow developing over a 0.0098 m
backward-facing step is considered at \( Re_\theta = 6.1 \times 10^2 \). Jovic and Driver [53] carried out
this experiment in a wind tunnel as a companion to the DNS validation of Le et al. [76].
The streamwise, spanwise, and vertical dimensions of the domain are \( 41 h_{\text{step}} \times 12 h_{\text{step}} \times
6 h_{\text{step}} \), where the height of the step is equal to the height of the experimental step and the
Streamwise turbulence intensity along the vertical direction sampled at streamwise locations downstream of the cube. -.-.-) no forcing; - - -) original forcing; —–) proposed forcing; X) experimental data for the high Reynolds number case of Castro and Robbins [21].

The size of the domain is comparable to the size of the numerical domain in Le et al. [76]. The precursor and main domains are shown in Figure 4.9, where the blue line represents the controlled forcing plane. The precursor and main domain are both periodic and use the same grid resolution. The three uniform discretizations considered are shown in Table 4.2. The three forcing methods (none, original, and proposed) are compared using the fine grid.

A turbulent boundary layer of thickness $0.0115$ m with a freestream velocity of $7.72$ m/s is imposed in the precursor simulation by a single controlled forcing plane using mean streamwise velocity and 2D Reynolds stress tensor profiles given by Jovic and Driver [53]. Because correlations are only known for the streamwise and vertical velocities, no force was applied to the spanwise momentum equation for the proposed forcing method. As in Section 4.4.1, only $\langle w'w' \rangle$ is used as a target for the original forcing method. The initial
Table 4.2

Grid point dimensions of the low Reynolds number turbulent boundary layer domain.

<table>
<thead>
<tr>
<th>Case</th>
<th>$N_x \times N_y \times N_z$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>$128 \times 32 \times 96$</td>
</tr>
<tr>
<td>Medium</td>
<td>$256 \times 64 \times 144$</td>
</tr>
<tr>
<td>Fine</td>
<td>$384 \times 96 \times 192$</td>
</tr>
</tbody>
</table>

fluctuations are provided by a random field of white noise scaled to match the Reynolds stress tensor profiles. The simulations comparing the forcing methods were all run until the first method converged to the desired Reynolds stresses, within 144 flow-throughs; then statistics were collected for 30 flow-throughs.

The mean streamwise velocity and non-zero Reynolds stress profiles sampled at the location of controlled forcing plane are plotted in Figure 4.10. These profiles were collected from the precursor simulation using the proposed forcing method for three different grid resolutions and compared with the experimental data from Jovic and Driver [53]. As was seen in the high Reynolds number turbulent boundary layer case, the proposed forcing method was able to successfully match the target Reynolds stress profiles for all three grid resolutions.

Contour plots of the instantaneous streamwise velocity in the precursor and main domain for the three different forcing cases are presented in Figures 4.11 and 4.12. The areas of grey represent where the immersed boundary method is applied to model the presence of the step. The region of zero velocity immediately upstream from the outlet of the main domains is a product of the concurrent precursor forcing, clearly showing how the precur-
Figure 4.9

The physical dimensions of the domain used for comparison to the experimental results of Jovic and Driver [53]. The blue line represents the single controlled forcing plane.

Figure 4.10

Turbulent statistics along the vertical direction from the precursor domain, using the proposed forcing method and three grid resolutions, sampled at the controlled forcing plane at conditions matching the momentum thickness Reynolds number of Jovic and Driver [53]. -.-.-) coarse; - - -) medium; —–) fine; X) experimental data

Distribution Statement A. Approved for public release; distribution is unlimited.
sor forcing region successfully transferred the precursor flow field into the main domain. The highly turbulent wake of the step is smoothly transformed into the precursor domain flow. Looking at the precursor contour for the “original forcing” case in Figure 4.11, the region of much larger magnitude velocity near the wall immediately after the forcing plane is characteristic of the controlled forces still trying to converge. Whereas immediately downstream of the proposed forcing plane, there is only a small increase in velocity to counteract the natural decay as the fluctuations move throughout the rest of the domain.

Figure 4.11

Contour of the instantaneous streamwise velocity normalized by the freestream velocity on an xz plane through the center of the precursor domain, at conditions matching the momentum thickness Reynolds number in Jovic and Driver [53]. The black line represents the controlled forcing plane. top) “no forcing”; middle) “original forcing”; bottom) proposed forcing

The mean streamwise velocity and given Reynolds stress profiles from the precursor simulation are plotted in Figure 4.13, alongside the experimental data from Jovic and
Contour of the instantaneous streamwise velocity normalized by the freestream velocity on an xz plane through the center of the main domain, at conditions matching the momentum thickness Reynolds number in Jovic and Driver [53]. The black line represents the controlled forcing plane. top) “no forcing”; middle) “original forcing”; bottom) proposed forcing

Driver [53] at the location of the controlled forcing plane. Once again, excellent agreement was found for all of profiles for the proposed forcing case. Unlike the high Reynolds number boundary layer, the smaller boundary layer height allowed for the “original forcing” method to nearly converge on the wall-normal Reynolds stress by the time the proposed forcing reached convergence. This in turn produces a noticeable effect in the Reynolds shear stress, but only a small effect on the streamwise Reynolds stress at that point in time. The “no forcing” case also shows significant development in the wall-normal and Reynolds shear stress. The proposed forcing method is again seen to reduce the development time of the turbulent boundary layer as compared to the “no forcing” and “original forcing” cases.
Turbulent statistics along the vertical direction from the precursor domain sampled at the controlled forcing plane at conditions matching the momentum thickness Reynolds number of Jovic and Driver [53]. "no forcing"; "original forcing"; proposed forcing; X) experimental data

The mean streamwise velocity, two-dimensional turbulent kinetic energy, and Reynolds shear stress profiles along the vertical direction are plotted in Figures 4.14, 4.15, and 4.16 alongside the experimental data from Jovic and Driver at four streamwise locations downstream of the step, \( x/h_{\text{step}} = \{6, 10, 15, 19\} \). The profiles at \( 6h_{\text{step}} \) are located at the edge of recirculation bubble in the experimental flow. As seen in the mean velocity profile, the numerical simulations overpredicted the length of this bubble. This was expected because the Monin-Obukhov similarity theory used to model the flow at the wall was not developed for use in regions of recirculating flow. The wall model predicts the size of the recirculation bubble close enough to still allow comparisons of the turbulent statistics farther away from it. The farther away the profiles are from the recirculating region the better the solution calculated with the proposed forcing method matches with the experimental data. The “no forcing” and “original forcing” cases both show lower two-dimensional turbulent kinetic energy and Reynolds shear stress around one step height, with the difference between the...
proposed forcing case and the other two cases growing smaller moving downstream of the step. For this case, the imposition of the correct levels of boundary layer turbulence only gives a minor improvement.

![Image](image.png)

Figure 4.14

Mean streamwise velocity along the vertical direction sampled at streamwise locations downstream of the step. -.-.-) “no forcing”; - - -) “original forcing”; —–) proposed forcing; X) experimental data for the low Reynolds number case of Jovic and Driver [53].

4.5 Conclusions and Future Work

An extension of the original controlled forcing method was proposed and added into an existing concurrent precursor simulation method along with a mean flow forcing method. By calculating the controlled forces in the principal-axis coordinate system and applying them to each of the momentum equations, after a transformation, the Control Forced Concurrent Precursor Method is able to match a full anisotropic Reynolds stress tensor. Using high and low Reynolds number turbulent boundary layer flows as a test cases, the CFCPM matched the given mean velocity and both 3D and 2D Reynolds stress tensor profiles in

Distribution Statement A. Approved for public release; distribution is unlimited.
Figure 4.15

2D turbulent kinetic energy along the vertical direction sampled at streamwise locations downstream of the step. -.-.-) “no forcing”; - - -) “original forcing”; —–) proposed forcing; X) experimental data for the low Reynolds number case of Jovic and Driver [53].

Figure 4.16

Reynolds shear stress along the vertical direction sampled at streamwise locations downstream of the step. -.-.-) “no forcing”; - - -) “original forcing”; —–) proposed forcing; X) experimental data for the low Reynolds number case of Jovic and Driver [53].
the precursor simulation. The main domain flows showed good agreement with experimental wind-tunnel results for flows around a wall-mounted cube and over a backward-facing step. Simulations without forcing and with only the original controlled forcing did not reproduce the desired Reynolds stresses in the precursor simulations within the time period it took for the proposed controlled forcing to reproduce them. This shows that the proposed controlled forcing reduced the development times of the two turbulent boundary layers. The proposed controlled forcing also showed a modest improvement in agreement with the experimental results over the other two forcing cases in the main domain for the high Reynolds number turbulent boundary layer case, but only a slight improvement in low Reynolds number case.

The major development area for the Control Forced Concurrent Precursor Method is extending the controlled forcing to the reproduction of turbulent scalar flux profiles. The turbulent scalar fluxes are single point correlations between the fluctuating velocity components and the fluctuating scalar; this is analogous to the Reynolds stresses which are single point correlations between the different fluctuating velocity components. Given turbulent scalar flux profiles as targets, controlled forces can be added to the appropriate governing equations in a correlated manner. A force controlled by a mean scalar profile could also be added. The specific motivation for this development path is the study of the effect of large wind farms on the Atmospheric Boundary Layer (ABL). Experimental measurements and numerical simulation have previously indicated that wakes generated inside large wind farms can substantially impact the exchanges of heat and humidity within the ABL. Previous LES studies of wind farms have employed the concurrent precursor method without

145
Distribution Statement A. Approved for public release; distribution is unlimited.
controlled forcing [156, 108, 2], which required long simulations times for the ABL to become fully developed in the precursor domains. By applying the controlled forcing to not only the velocity components, but to also the temperature and the humidity, the overall simulation time required for studying various wind farm configurations in the same atmospheric conditions can be reduced.
REFERENCES


Distribution Statement A. Approved for public release; distribution is unlimited.


Distribution Statement A. Approved for public release; distribution is unlimited.


Distribution Statement A. Approved for public release; distribution is unlimited.


Distribution Statement A. Approved for public release; distribution is unlimited.


Distribution Statement A. Approved for public release; distribution is unlimited.


Distribution Statement A. Approved for public release; distribution is unlimited.


Distribution Statement A. Approved for public release; distribution is unlimited.


APPENDIX A

VALIDATION OF THE COHERENT STRUCTURE MODEL - TURBULENT MIXING LAYER
In order to validate the implementation of the Coherent Structure Model (and also an implementation of the traditional Smagorinsky model [149]), the simulation of a temporally developing weakly compressible turbulent mixing layer was undertaken. The results from these simulations were then compared with the LES data from Vreman et al. [163]. Four different cases are considered: DNS, LES without an SGS model, LES with the traditional Smagorinsky SGS model, and LES with the CSM.

The flow conditions, numerical domain, and discretizations are all identical to those in Vreman et al. [163]. The convective Mach number is 0.2 and the Reynolds number based on the upper-stream velocity and half of the initial vorticity thickness is 50. The dimensions of the cubic domain are $L \times L \times L$, where $L$ is equal to four times the wavelength of the most unstable mode according to linear stability theory [134, 163]. Both the DNS grid and the LES grid employ uniform Cartesian grids consisting of $192 \times 192 \times 192$ and $32 \times 32 \times 32$ grid points, respectively. Periodic boundary conditions are applied in the streamwise and spanwise directions, while farfield boundaries are used in the vertical direction. The initial non-dimensional mean velocity profile is given by the hyperbolic tangent function, $U/U_\infty = \tanh(y/\delta(0))$, where $\delta(0)$ is the initial vorticity thickness. The initial mean pressure distribution is uniform and the initial temperature profile is given by the Crocco-Busemann relation. While the initial perturbation field in Vreman et al. [163] is formed from the superposition of eigenfunctions of unstable waves provided by linear stability theory [134], the initial perturbation field in this work is created from two- and three-dimensional modes based on the simpler Kelvin-Helmhotz instability solution for an inviscid mixing layer [114]. Three two-dimensional perturbation modes with streamwise
wave numbers of 1, 2, and 4 and six three-dimensional perturbation modes with streamwise and spanwise wavenumber pairings of \((1, 1), (1, -1), (2, 2), (2, -2), (4, 4),\) and \((4, -4)\) were imposed. The non-dimensional amplitude of the 2D modes was 0.05 and 0.15 for the 3D modes.

Isosurfaces of Q-criterion for the four cases at the same non-dimensional time are shown in Figure A.1. As expected, the traditional Smagorinsky model is overly dissipative and destroys all but the largest flow features. The flow features in the CSM case qualitatively appear closer in nature to the DNS case than the case without an SGS model. Within the resolution of the LES grid, the CSM case shows smaller turbulent structures than are present in the No SGS case.

Figure A.2 presents the temporal evolution of the momentum thickness of the mixing layer for the No SGS, Smagorinsky, and CSM cases along with the LES data from Vreman et al. [163]. Excellent agreement is seen for all cases, especially after a non-dimensional time of 40. The smaller discrepancies early on are caused by using the simpler Kelvin-Helmholtz based initial perturbations instead of the linear stability theory based perturbations.

The vertical profiles of the streamwise and spanwise averaged turbulence intensities and Reynolds shear stress for the CSM case at the non-dimensional time of 70 are plotted in Figure A.3 along with the traditional Dynamic Smagorinsky LES results from Vreman et al. [163]. Very good agreement is seen between the two different dynamic methods. The CSM was able to not only reasonably reproduced the peak intensities and Reynolds shear stress, but also the spreading of the mixing layer. This matching of the vertical spread of
Figure A.1

Isosurfaces of Q-criterion for the DNS and different SGS cases at the same non-dimensional time.
Figure A.2

Temporal evolution of the momentum thickness of the mixing layer. The symbols all represent data from from Vreman et al. [163]. blue) no SGS model; red) Smagorinksy SGS model; yellow) Coherent Structure Model; O) no SGS model; Δ) Smagorinsky SGS model; +) Dynamic Smagorinsky SGS model.

The turbulent statistics is consistent with matching seen in the growth of the momentum thickness.

As was seen in previous studies [67, 68, 112, 15], this implementation of the Coherent Structure Model performed just as well as the Dynamic Smagorinsky Model and will be utilized to model the subgrid stress in all subsequent large eddy simulations involving the Triple Hill’s Vortex Synthetic Eddy Method.
Vertical profiles of the turbulence intensities, $\sqrt{\langle \rho u'_i u'_i \rangle}$, and Reynolds shear stress, $-\langle \rho u'_i v'_i \rangle$, for the temporal mixing layer at $tU_{\infty}/\delta_{\omega(0)} = 70$. Yellow) Coherent Structure Model; +) Dynamic Smagorinsky SGS model from Vreman et al. [163].

Figure A.3

Distribution Statement A. Approved for public release; distribution is unlimited.
APPENDIX B

DIVERGENCE-FREE ORIENTATION OF A HILL’S VORTEX
B.1 Divergence of a Hill’s Vortex oriented in the \( x \)-direction

Consider an arbitrary orthogonal coordinate system \((x,y,z)\) with a Hill’s vortex of radius \( a \) and constant amplitude \( u_0^x \) located at the origin oriented and in the \( x \)-direction. From equations (3.38), (3.39), and (3.40), the velocity components of this Hill’s vortex are

\[
\begin{align*}
    u^x(x, y, z) &= \begin{cases} 
    \frac{3}{2} u_0^x \left( 1 - \frac{x^2 + 2y^2 + 2z^2}{a^2} \right) & ; \quad x^2 + y^2 + z^2 \leq a^2 \\
    \frac{3}{2} u_0^x \left[ \frac{a^2}{x^2 + y^2 + z^2} \left( \frac{2y^2 - y^2 - z^2}{2a^2} \right) - 1 \right] & ; \quad x^2 + y^2 + z^2 > a^2
    \end{cases} \\
    v^x(x, y, z) &= \begin{cases} 
    \frac{3}{2} u_0^x \frac{xy}{a^2} & ; \quad x^2 + y^2 + z^2 \leq a^2 \\
    \frac{3}{2} u_0^x \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{5}{2}} \left( \frac{2x^2 - y^2 - z^2}{2a^2} \right) & ; \quad x^2 + y^2 + z^2 > a^2
    \end{cases} \\
    w^x(x, y, z) &= \begin{cases} 
    \frac{3}{2} u_0^x \frac{xz}{a^2} & ; \quad x^2 + y^2 + z^2 \leq a^2 \\
    \frac{3}{2} u_0^x \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{5}{2}} & ; \quad x^2 + y^2 + z^2 > a^2
    \end{cases}
\end{align*}
\]

Taking the divergence of velocity inside the Hill’s vortex \((x^2 + y^2 + z^2 \leq a^2)\),

\[
(\nabla \cdot \mathbf{u}^x)_{inside} = \frac{\partial u^x}{\partial x} + \frac{\partial v^x}{\partial y} + \frac{\partial w^x}{\partial z} = 0
\]

Distribution Statement A. Approved for public release; distribution is unlimited.
Taking the divergence of velocity outside the Hill’s vortex \((x^2 + y^2 + z^2 > a^2)\),

\[
(\nabla \cdot u^x)_{outside} = \frac{\partial u^x}{\partial x} + \frac{\partial v^x}{\partial y} + \frac{\partial w^x}{\partial z}
\]

\[
= u_0^x \left[ -\frac{5}{2} \left( \frac{a^2}{x^2+y^2+z^2} \right)^{\frac{3}{2}} \left( \frac{2x}{a^2} \right) \left( \frac{a^2}{x^2+y^2+z^2} \right)^2 \left( \frac{2x^2-y^2-z^2}{2a^2} \right) \right.

+ \left( \frac{a^2}{x^2+y^2+z^2} \right) \left( \frac{4x}{2a^2} \right) \right]

+ u_0^x \left\{ \frac{3x}{2a^2} \left( \frac{a^2}{x^2+y^2+z^2} \right)^{\frac{5}{2}} \right.

+ \frac{3xy}{2a^2} \left[ -\frac{5}{2} \left( \frac{a^2}{x^2+y^2+z^2} \right)^{\frac{3}{2}} \left( \frac{2y}{a^2} \right) \left( \frac{a^2}{x^2+y^2+z^2} \right)^2 \right] \right\}

+ u_0^x \left\{ \frac{3x}{2a^2} \left( \frac{a^2}{x^2+y^2+z^2} \right)^{\frac{5}{2}} \right.

+ \frac{3xz}{2a^2} \left[ -\frac{5}{2} \left( \frac{a^2}{x^2+y^2+z^2} \right)^{\frac{3}{2}} \left( \frac{2z}{a^2} \right) \left( \frac{a^2}{x^2+y^2+z^2} \right)^2 \right] \right\}

= u_0^x \left[ -\frac{5x}{a^2} \left( \frac{a^2}{x^2+y^2+z^2} \right)^{\frac{7}{2}} \left( \frac{2x^2-y^2-z^2}{2a^2} \right) + \frac{2x}{a^2} \left( \frac{a^2}{x^2+y^2+z^2} \right)^{\frac{5}{2}} \right]

+ u_0^x \left\{ \frac{3x}{2a^2} \left( \frac{a^2}{x^2+y^2+z^2} \right)^{\frac{5}{2}} - \frac{15xy}{2a^4} \left[ \left( \frac{a^2}{x^2+y^2+z^2} \right)^{\frac{7}{2}} \right] \right\}

+ u_0^x \left\{ \frac{3x}{2a^2} \left( \frac{a^2}{x^2+y^2+z^2} \right)^{\frac{5}{2}} - \frac{15xz}{2a^4} \left[ \left( \frac{a^2}{x^2+y^2+z^2} \right)^{\frac{7}{2}} \right] \right\}

= \frac{u_0^x}{2a^4} \left( \frac{a^2}{x^2+y^2+z^2} \right)^{\frac{7}{2}} \left\{ \left[ -5 \left( 2x^2-y^2-z^2 \right) + 4 \left( x^2+y^2+z^2 \right) \right]

+ \left[ 3 \left( x^2+y^2+z^2 \right) - 15y^2 \right] + \left[ 3 \left( x^2+y^2+z^2 \right) - 15z^2 \right] \right\}

= \frac{u_0^x}{2a^4} \left( \frac{a^2}{x^2+y^2+z^2} \right)^{\frac{7}{2}} \left[ \left( -10 + 4 + 3 + 3 \right) x^2 + \left( 5 + 4 + 3 + 3 - 15 \right) y^2 \right.

+ \left( 5 + 4 + 3 + 3 - 15 \right) z^2 \right]

= \frac{u_0^x}{2a^4} \left( \frac{a^2}{x^2+y^2+z^2} \right)^{\frac{7}{2}} \left[ \left( 0 \right) x^2 + \left( 0 \right) y^2 + \left( 0 \right) z^2 \right]

(\nabla \cdot u^x)_{outside} = 0

Distribution Statement A. Approved for public release; distribution is unlimited.
Since the velocities inside and outside the vortex are both divergence-free,

\[
\begin{align*}
(\nabla \cdot \mathbf{u}^x)_{\text{inside}} &= 0 \\
(\nabla \cdot \mathbf{u}^x)_{\text{outside}} &= 0
\end{align*}
\Rightarrow \nabla \cdot \mathbf{u}^x = 0 \quad (B.4)
\]

the Hill’s vortex oriented in the \(x\)-direction is divergence-free.

**B.2 Divergence of a Hill’s Vortex oriented in the \(y\)-direction**

Consider an arbitrary orthogonal coordinate system \((x, y, z)\) with a Hill’s vortex of radius \(a\) and constant amplitude \(u_0^y\) located at the origin oriented and in the \(y\)-direction. From equations (3.38), (3.39), and (3.41), the velocity components of this Hill’s vortex are

\[
\begin{align*}
u^y(x, y, z) &= \begin{cases} 
\frac{3}{2} u_0^y \frac{x^2}{a^2} & ; \quad x^2 + y^2 + z^2 \leq a^2 \\
\frac{3}{2} u_0^y \frac{x^2}{a^2} \left(\frac{x^2}{x^2 + y^2 + z^2}\right)^{\frac{5}{2}} & ; \quad x^2 + y^2 + z^2 > a^2
\end{cases} \\
v^y(x, y, z) &= \begin{cases} 
\frac{3}{2} u_0^y \left(1 - \frac{2x^2 + 2y^2 + 2z^2}{a^2}\right) & ; \quad x^2 + y^2 + z^2 \leq a^2 \\
u_0^y \left[\left(\frac{a^2}{x^2 + y^2 + z^2}\right)^{\frac{5}{2}} \left(\frac{2y^2 - x^2 - z^2}{2a^4}\right) - 1\right] & ; \quad x^2 + y^2 + z^2 > a^2
\end{cases} \\
w^y(x, y, z) &= \begin{cases} 
\frac{3}{2} u_0^y \frac{y}{a^2} & ; \quad x^2 + y^2 + z^2 \leq a^2 \\
w_0^y \frac{y}{a^2} \left(\frac{a^2}{x^2 + y^2 + z^2}\right)^{\frac{5}{2}} & ; \quad x^2 + y^2 + z^2 > a^2
\end{cases}
\end{align*}
\]

(B.5)

Taking the divergence of velocity inside the Hill’s vortex \((x^2 + y^2 + z^2 \leq a^2)\),

\[
(\nabla \cdot \mathbf{u}^y)_{\text{inside}} = \frac{\partial u^y}{\partial x} + \frac{\partial v^y}{\partial y} + \frac{\partial w^y}{\partial z} = 0 \quad (B.6)
\]

Distribution Statement A. Approved for public release; distribution is unlimited.
Taking the divergence of velocity outside the Hill’s vortex \((x^2 + y^2 + z^2 > a^2)\),

\[
\begin{align*}
(\nabla \cdot \bm{u}^y)_{\text{outside}} &= \frac{\partial u_y}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial w_y}{\partial z} \\
&= u_0^y \left\{ \frac{3y}{2a^2} \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{3}{2}} \right. \\
&\quad + \frac{3xy}{2a^2} \left[ -\frac{5}{2} \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{3}{2}} \left( \frac{2x}{a^2} \right) \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^2 \right] \\
&\quad + u_0^y \left[ -\frac{5}{2} \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{3}{2}} \left( \frac{2y}{a^2} \right) \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^2 \left( \frac{2y^2 - x^2 - z^2}{2a^2} \right) \right] \\
&\quad + \left. \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{3}{2}} \left( \frac{4y}{2a^2} \right) \right] \\
&\quad + u_0^y \left\{ \frac{3y}{2a^2} \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{3}{2}} \\
&\quad + \frac{3yz}{2a^2} \left[ -\frac{5}{2} \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{3}{2}} \left( \frac{2z}{a^2} \right) \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^2 \right] \\
&\quad + u_0^y \left[ -\frac{5y}{a^2} \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{3}{2}} \left( \frac{2y^2 - x^2 - z^2}{2a^2} \right) + \frac{2y}{a^2} \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{3}{2}} \right] \\
&\quad + u_0^y \left\{ \frac{3y}{2a^2} \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{3}{2}} \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{3}{2}} \right. \\
&\quad - \frac{15y^2}{2a^4} \left[ \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{3}{2}} \right] \right. \\
&\quad + \frac{5y}{a^2} \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{3}{2}} \left( \frac{2y - x - z}{2a^2} \right) + \frac{2y}{a^2} \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{3}{2}} \right] \\
&\quad + \left. \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{3}{2}} \left( \frac{4y}{2a^2} \right) \right] \\
&\quad = u_0^y \frac{y}{2a^4} \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{5}{2}} \left[ \left[ 3 \left( x^2 + y^2 + z^2 \right) - 15x \right] \\
&\quad + \left[ -5 \left( 2y^2 - x^2 - z^2 \right) + 4 \left( x^2 + y^2 + z^2 \right) \right] + \left[ 3 \left( x^2 + y^2 + z^2 \right) - 15z^2 \right] \right] \\
&\quad = u_0^y \frac{y}{2a^4} \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{5}{2}} \left[ \left( 5 + 4 + 3 + 3 - 15 \right) x^2 + \left( -10 + 4 + 3 + 3 \right) y^2 \right. \\
&\quad + \left( 5 + 4 + 3 + 3 - 15 \right) z^2 \right] \\
&\quad = u_0^y \frac{y}{2a^4} \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{5}{2}} \left[ \left( 0 \right) x^2 + \left( 0 \right) y^2 + \left( 0 \right) z^2 \right] \\

(\nabla \cdot \bm{u}^y)_{\text{outside}} &= 0
\end{align*}
\]

Distribution Statement A. Approved for public release; distribution is unlimited.
Since the velocities inside and outside the vortex are both divergence-free,
\[
(\nabla \cdot \mathbf{u}^y)_{\text{inside}} = 0 \quad \Rightarrow \quad \nabla \cdot \mathbf{u}^y = 0 \tag{B.8}
\]
the Hill’s vortex oriented in the \( y \)-direction is divergence-free.

### B.3 Divergence of a Hill’s Vortex oriented in the \( z \)-direction

Consider an arbitrary orthogonal coordinate system \((x, y, z)\) with a Hill’s vortex of radius \( a \) and constant amplitude \( u_0^z \) located at the origin oriented and in the \( z \)-direction. From equations (3.38), (3.39), and (3.42), the velocity components of this Hill’s vortex are

\[
\begin{align*}
    u^z(x, y, z) &= \begin{cases} \frac{3}{2} u_0^z \frac{x^2}{a^2} & ; \ x^2 + y^2 + z^2 \leq a^2 \\ \frac{3}{2} u_0^z \frac{x^2}{a^2} \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{3}{2}} & ; \ x^2 + y^2 + z^2 > a^2 \end{cases} \\
    v^z(x, y, z) &= \begin{cases} \frac{3}{2} u_0^z \frac{y^2}{a^2} & ; \ x^2 + y^2 + z^2 \leq a^2 \\ \frac{3}{2} u_0^z \frac{y^2}{a^2} \left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{3}{2}} & ; \ x^2 + y^2 + z^2 > a^2 \end{cases} \\
    w^z(x, y, z) &= \begin{cases} \frac{3}{2} u_0^z \left( 1 - \frac{2x^2 + 2y^2 + z^2}{a^2} \right) & ; \ x^2 + y^2 + z^2 \leq a^2 \\ u_0^z \left[ \frac{\left( \frac{a^2}{x^2 + y^2 + z^2} \right)^{\frac{1}{2}}} {\left( \frac{2x^2 + 2y^2 + z^2}{2a^2} \right)} - 1 \right] & ; \ x^2 + y^2 + z^2 > a^2 \end{cases}
\end{align*}
\tag{B.9}
\]

Taking the divergence of velocity inside the Hill’s vortex \((x^2 + y^2 + z^2 \leq a^2)\),

\[
(\nabla \cdot \mathbf{u}^z)_{\text{inside}} = \frac{\partial u^z}{\partial x} + \frac{\partial v^z}{\partial y} + \frac{\partial w^z}{\partial z} \tag{B.10}
\]

\[
= \frac{3}{2} u_0^z \frac{z}{a^2} + \frac{3}{2} u_0^z \frac{z}{a^2} + \frac{3}{2} u_0^z \left( - \frac{2z}{a^2} \right)
= 3u_0^z \frac{z}{a^2} - 3u_0^z \frac{z}{a^2}
= 0
\]

Distribution Statement A. Approved for public release; distribution is unlimited.
Taking the divergence of velocity outside the Hill’s vortex \((x^2 + y^2 + z^2 > a^2)\),

\[
(\nabla \cdot \mathbf{u}^z)_{\text{outside}} = \frac{\partial u^z}{\partial x} + \frac{\partial v^z}{\partial y} + \frac{\partial w^z}{\partial z} = 0
\]
Since the velocities inside and outside the vortex are both divergence-free,

\[
\begin{align*}
(\nabla \cdot \mathbf{u}^z)_{\text{inside}} &= 0 \\
(\nabla \cdot \mathbf{u}^z)_{\text{outside}} &= 0
\end{align*}
\]

\[\implies \nabla \cdot \mathbf{u}^z = 0 \quad \text{(B.12)}\]

the Hill’s vortex oriented in the \(z\)-direction is divergence-free.
APPENDIX C

DIVERGENCE-FREE CONVECTION OF A HILL’S VORTEX
Consider an arbitrary orthogonal coordinate system \((x, y, z)\) with a Hill’s vortex of radius \(a\) and constant amplitude \(u_0^x\) located at the origin oriented and in the \(x\)-direction. A spatially-varying mean flow \((U(x, y, z), V(x, y, z), W(x, y, z))\) is convecting the vortex over time \(t\). From equations (3.38), (3.39), and (3.40), the velocity components inside of this Hill’s vortex are

\[
\begin{align*}
    u^x(x', y', z') & = \frac{3}{2} u_0^x \left( 1 - \frac{x'^2 + 2y'^2 + 2z'^2}{a^2} \right) \\
    v^x(x', y', z') & = \frac{3}{2} u_0^x \frac{x'y'}{a^2} \\
    w^x(x', y', z') & = \frac{3}{2} u_0^x \frac{x'z'}{a^2}
\end{align*}
\]  

where the new coordinates accounting for the convection are

\[
\begin{align*}
    x' & = x - U(x, y, z)t \\
    y' & = y - V(x, y, z)t \\
    z' & = z - W(x, y, z)t
\end{align*}
\]  

Taking the divergence of velocity inside the Hill’s vortex,

\[
(\nabla \cdot \mathbf{u}^x)_{\text{inside}} = \frac{\partial u^x}{\partial x} + \frac{\partial v^x}{\partial y} + \frac{\partial w^x}{\partial z}
\]

\[
= - \frac{3u_0^x}{2a^2} \left( 2x' \frac{\partial x'}{\partial x} + 4y' \frac{\partial y'}{\partial x} + 4z' \frac{\partial z'}{\partial x} \right) + \frac{3u_0^y}{2a^2} \left( y' \frac{\partial x'}{\partial y} + x' \frac{\partial y'}{\partial y} \right) + \frac{3u_0^z}{2a^2} \left( z' \frac{\partial x'}{\partial z} + x' \frac{\partial z'}{\partial z} \right)
\]
Inserting the coordinate derivatives and rearranging,

\[ (\nabla \cdot \mathbf{u})_{\text{inside}} = -\frac{3u_0}{2a^2} \left[ 2x' \left( 1 - \frac{\partial U}{\partial x} t \right) - 4y' \frac{\partial V}{\partial x} t + 4z' \frac{\partial W}{\partial x} t \right] \]

\[ + \frac{3u_0}{2a^2} \left[ -y' \frac{\partial U}{\partial y} t + x' \left( 1 - \frac{\partial V}{\partial y} t \right) \right] \]

\[ + \frac{3u_0}{2a^2} \left[ -z' \frac{\partial U}{\partial z} t + x' \left( 1 - \frac{\partial W}{\partial z} t \right) \right] \]

\[ = \frac{3u_0}{2a^2} \left\{ (-2x' + x' + x') + \left[ (2x' \frac{\partial U}{\partial x} - y' \frac{\partial U}{\partial y} - z' \frac{\partial U}{\partial z}) \right] \right\} \]

\[ + \left( 4y' \frac{\partial V}{\partial x} - x' \frac{\partial V}{\partial y} \right) + \left( 4z' \frac{\partial W}{\partial x} - x' \frac{\partial W}{\partial z} \right) \]

\[ (\nabla \cdot \mathbf{u})_{\text{inside}} = \frac{3u_0}{2a^2} \left[ \left( 2x' \frac{\partial U}{\partial x} - y' \frac{\partial U}{\partial y} - z' \frac{\partial U}{\partial z} \right) + \left( 4y' \frac{\partial V}{\partial x} - x' \frac{\partial V}{\partial y} \right) \right] \]

\[ + \left( 4z' \frac{\partial W}{\partial x} - x' \frac{\partial W}{\partial z} \right) \]  

If the mean flow components do not vary in space,

\[ (\nabla \cdot \mathbf{u})_{\text{inside}} = 0 \]  

(C.6)

the divergence-free condition is satisfied. Analyses on vortices oriented in the \( y \)- and \( z \)-directions lead to a similar conclusion.

Distribution Statement A. Approved for public release; distribution is unlimited.
APPENDIX D

PROPORTIONAL CONTROL OF THE AMPLITUDES OF THE TRIPLE HILL’S VORTEX
This serves as an alternative to the method presented in Section 3.3.7 for accounting for the influence of the THV’s currently at the inlet when a new THV is created. The amplitudes of all of the THV’s are multiplied by a scale factor controlled using a proportional controller based on the error between the given principal-axis Reynolds stresses and imposed principal-axis Reynolds stress. This scaling method was shown in Haywood et al. [42] to require multiple time periods before the scale factors converged on values that recovered the matching of a given Reynolds stress tensor. It is because of this development time that the Modification of the Target Reynolds Stresses method in Section 3.3.7 replaced the proportional controller.

The imposed Reynolds stress tensor at any point in space is reproduced by a single stream of THV’s moving through that point in time. To account for that fact that the imposed turbulent inflow is a combination of many THV’s over a range of different sizes, the amplitudes of each THV need to be scaled to ensure that Reynolds stress matching is recovered. So, the definition of the amplitudes in equation (3.51) is modified as follows. Let the amplitudes in the principal-axis coordinate system associated with a THV now be defined as a product of a constant amplitude, a random number, and a constant scale factor.

\[
\begin{bmatrix}
u_{0x}^p \\
u_{0y}^p \\
u_{0z}^p \\
\end{bmatrix} = \begin{bmatrix}
s_x \epsilon_x \tilde{u}_{0x}^p \\
s_y \epsilon_y \tilde{u}_{0y}^p \\
s_z \epsilon_z \tilde{u}_{0z}^p \\
\end{bmatrix}
\]  

(D.1)
where $s^x$, $s^y$, and $s^z$ are the constant scale factors. The values of these scale factors are controlled using a proportional controller,

$$
\begin{align*}
    s^x_{new} &= s^x_{old} + K^x \left( \sqrt{\langle u'w' \rangle_p^{p\text{ given}}} - \sqrt{\langle u'w' \rangle_p^{p\text{ imposed}}} \right) \\
    s^y_{new} &= s^y_{old} + K^y \left( \sqrt{\langle v'v' \rangle_p^{p\text{ given}}} - \sqrt{\langle v'v' \rangle_p^{p\text{ imposed}}} \right) \\
    s^z_{new} &= s^z_{old} + K^z \left( \sqrt{\langle w'w' \rangle_p^{p\text{ given}}} - \sqrt{\langle w'w' \rangle_p^{p\text{ imposed}}} \right)
\end{align*}
$$

where $K^x$, $K^y$, and $K^z$ are damping constants. The scale factors are updated until the imposed Reynolds stress tensor components, $\left( \langle u'w' \rangle_p^{p\text{ imposed}}, \langle v'v' \rangle_p^{p\text{ imposed}}, \langle w'w' \rangle_p^{p\text{ imposed}} \right)$, match the imposed components, $\left( \langle u'w' \rangle_p^{p\text{ given}}, \langle v'v' \rangle_p^{p\text{ given}}, \langle w'w' \rangle_p^{p\text{ given}} \right)$, in the principal-axis directions of the given flow. The Reynolds stresses of the imposed flow are also averaged along the homogeneous directions on the inlet plane before the principal-axis stresses are determined. For homogeneous turbulent flows, each point on the inlet plane will have the same scale factors; while for turbulent channel flows, the scale factors will vary in the direction perpendicular to the walls.