Numerical experiments in geostrophic turbulence using the finite element method

Jeremy Allen Sauer

The University of Montana
The University of Montana

Permission is granted by the author to reproduce this material in its entirety, provided that this material is used for scholarly purposes and is properly cited in published works and reports.

**Please check "Yes" or "No" and provide signature**

Yes, I grant permission

No, I do not grant permission

Author's Signature:

Date: 5-31-05

Any copying for commercial purposes or financial gain may be undertaken only with the author's explicit consent.
Numerical Experiments in Geostrophic Turbulence Using the Finite Element Method

by

Jeremy Allen Sauer

B.A. Combined Physics/Computer Science, The University of Montana, 2002

presented in partial fulfillment of the requirements for the degree of Master Of Science

The University of Montana
Missoula, Montana
May, 2005

Approved by:

Co-Chairperson

Co-Chairperson

Dean, Graduate School

Date
Sauer, Jeremy A., M.S., June, 2005  

Numerical Experiments in Geostrophic Turbulence Using the Finite Element Method  

Co-Chairpersons: Dr. Jesse V. Johnson: Dept. of Computer Science,  
Dr. Andrew S. Ware: Department of Physics and Astronomy  

Turbulence is one of the most challenging problems in Computational Fluid Dynamics (CFD). The wide-range of scales involved and the exchange of energy between different scales make computation of turbulent flow numerically intensive. Geostrophic turbulent flow is particularly important in geophysical fluid dynamics where large spatial scales characterize motion on the surface of a rotating sphere and coupling between mean flow and background turbulence occurs. Shear, the difference in flow magnitude in a direction perpendicular to the mean flow, can have a strong impact on turbulent eddies, as in the Gulf stream where smaller scale flow dynamics occur within a larger scale mean zonal flow.

Many numerical methods exist for modeling geophysical fluid dynamics. Predominantly spectral or pseudospectral methods (SM,PSM) have been used to study geostrophic turbulence in the presence of mean shear. These approaches require periodic boundary conditions restricting experiments to mean shear profiles which are also periodic. This thesis seeks to test the hypothesis that the Finite Element Method (FEM) can be used to model geostrophic turbulence in the presence of a non-periodic mean shear.

Two strategies for including the effects of mean shear are explored. Firstly, in a kinematic description the mean shear is treated as an external force on the turbulent fluid. Secondly, the mean flow is allowed to evolve with the turbulent flow. When mean shear is treated as an external force the background turbulence has no feedback effect on the mean flow, and thus the mean flow profile is constant. In contrast, when mean flow is considered an integral part of the system, boundary values of the mean flow are constant and the initial profile of the mean flow is linear, but through coupling with the background flow can evolve over time. Using experiments of relatively short duration both strategies gave comparable results. Results also showed that FEM can be used to model geostrophic turbulence in the presence of mean shear profiles which cannot be studied using spectral methods.
Turbulence is one of the most challenging problems in Computational Fluid Dynamic (CFD). The wide-range of scales involved and the exchange of energy between different scales make computation of turbulent flow numerically intensive. Geostrophic turbulent flow is particularly important in geophysical fluid dynamics where large spatial scales characterize motion on the surface of a rotating sphere and coupling between mean flow and background turbulence occurs. Shear, the difference in flow magnitude in a direction perpendicular to the mean flow, can have a strong impact on turbulent eddies, as in the Gulf Stream where smaller scale flow dynamics occur within a larger scale mean zonal flow.

Many numerical methods exist for modeling geophysical fluid dynamics. Predominantly spectral or pseudospectral methods (SM,PSM) have been used to study geostrophic turbulence in the presence of mean shear. These approaches require periodic boundary conditions restricting experiments to mean shear profiles which are also periodic. This thesis seeks to test the hypothesis that the Finite Element Method (FEM) can be used to model geostrophic turbulence in the presence of a non-periodic mean shear.

Two strategies for including the effects of mean shear are explored. Firstly, in a kinematic description the mean shear is treated as an external force on the turbulent fluid. Secondly, the mean flow is allowed to evolve with the turbulent flow. When mean shear is treated as an external force the background turbulence has no feedback effect on the mean flow, and thus the mean flow profile is constant. In contrast, when mean flow is considered an integral part of the system, boundary values of the mean flow are constant and the initial profile of the mean flow is linear, but through coupling with the background flow can evolve over time. Using experiments of relatively short duration both strategies gave comparable results. Results also showed that FEM can be used to model geostrophic turbulence in the presence of mean shear profiles which cannot be studied using spectral methods.
ACKNOWLEDGMENTS

I would like to take this opportunity to express my thanks first to my parents for their encouragement, patience, and diligent efforts to ensure every opportunity was available to me. Second I would also like to thank my partner and soon to be wife for without her everyday support this would not have been possible.
# TABLE OF CONTENTS

ACKNOWLEDGMENTS ................................................................. iii

CHAPTER 1 Introduction .............................................................. 1
  Problem Overview and Solution Schemes ........................................ 1
  Current Research ................................................................. 2

CHAPTER 2 The Model ............................................................... 4
  Field Equations ................................................................. 4
    Navier-Stokes Equations ..................................................... 5
    The $\beta$-plane Approximation .......................................... 5
    Streamfunction - Vorticity Formulation .................................. 8
    Non-dimensionalization ................................................... 9
  Model Domain ................................................................. 10
  Boundary Conditions ......................................................... 11

CHAPTER 3 Numerical Methods .................................................. 14
  Pseudospectral Method ....................................................... 15
  Spectral Methods .......................................................... 15
  Collocation ................................................................. 16
  Pseudospectral vs. Spectral ............................................... 17
  Finite Element Method .................................................... 18
Galerkin Method ................................................................. 18
Mesh Generation ............................................................... 18

CHAPTER 4  FEM Implementation ......................................... 21
Femlab® ....................................................................................... 21
Five Steps to a Femlab® Model .............................................. 23
  Geometry .............................................................................. 23
  Mesh Generation ................................................................. 23
  Physics Definition .............................................................. 23
  Solving the ODE system ................................................... 24
  Postprocessing Results ..................................................... 24
Model Specific Implementation ............................................ 24
  General form PDE .............................................................. 25
  Subdomain Definition ........................................................ 25
  Boundary Definition .......................................................... 27
  Constraint Requirements .................................................. 27
External vs. Internal Mean Shear .......................................... 28
  External ............................................................................. 28
Internal Mean Shear ............................................................ 29
  Initial Conditions .............................................................. 29
  Neumann Boundaries ....................................................... 29
  Constraints ...................................................................... 30
Quantities of Interest .......................................................... 30

CHAPTER 5  Numerical Experiments ........................................ 33
Initial Conditions ................................................................. 33
Tolerance Sensitivity ........................................................... 34
<table>
<thead>
<tr>
<th>Topic</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>CHAPTER 6 Discussion and Future Directions</strong></td>
<td>63</td>
</tr>
<tr>
<td>Discussion</td>
<td>63</td>
</tr>
<tr>
<td>Future Directions</td>
<td>64</td>
</tr>
<tr>
<td><strong>BIBLIOGRAPHY</strong></td>
<td>65</td>
</tr>
<tr>
<td>Absolute vs. Relative Tolerance</td>
<td>36</td>
</tr>
<tr>
<td>Experiments</td>
<td>36</td>
</tr>
<tr>
<td>Degree of Freedom Sensitivity</td>
<td>37</td>
</tr>
<tr>
<td>Element Type</td>
<td>37</td>
</tr>
<tr>
<td>Element Size</td>
<td>47</td>
</tr>
<tr>
<td>Application to Mean Shear</td>
<td>47</td>
</tr>
<tr>
<td>Experiments</td>
<td>47</td>
</tr>
</tbody>
</table>
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Normalized domain and boundary.</td>
<td>11</td>
</tr>
<tr>
<td>2.2</td>
<td>Boundary labels.</td>
<td>12</td>
</tr>
<tr>
<td>3.1</td>
<td>Uniformly spaced 2-d grid, $\Delta x = \Delta y$.</td>
<td>17</td>
</tr>
<tr>
<td>3.2</td>
<td>Triangular mesh, maximum edge length = $L$.</td>
<td>19</td>
</tr>
<tr>
<td>3.3</td>
<td>Triangular mesh, maximum edge length = $\frac{L}{2}$.</td>
<td>20</td>
</tr>
<tr>
<td>4.1</td>
<td>Area of integration for $E_k(k)$.</td>
<td>31</td>
</tr>
<tr>
<td>5.1</td>
<td>Initial streamfunction and energy distribution.</td>
<td>35</td>
</tr>
<tr>
<td>5.2</td>
<td>$TE(t)$ and $EN(t)$ for increasingly restrictive tolerance settings. $U_{ms} = 0.0$, $R_e = 10^3$, $E_k = 10^{-4}$.</td>
<td>38</td>
</tr>
<tr>
<td>5.3</td>
<td>$TE(t)$ and $EN(t)$ for increasingly restrictive tolerance settings. $U_{ms} = 1.0$, $R_e = 10^3$, $E_k = 10^{-4}$.</td>
<td>39</td>
</tr>
<tr>
<td>5.4</td>
<td>$TE(t)$ and $EN(t)$ for increasingly restrictive tolerance settings. $U_{ms} = 10.0$, $R_e = 10^3$, $E_k = 10^{-4}$.</td>
<td>40</td>
</tr>
<tr>
<td>5.5</td>
<td>General triangular element.</td>
<td>42</td>
</tr>
<tr>
<td>5.6</td>
<td>Lagrange cubic element, with 3 vertex nodes (circles), 6 line nodes (squares), and one interior node (triangle).</td>
<td>43</td>
</tr>
<tr>
<td>5.7</td>
<td>Hermite cubic element with 6 vertex nodes (circles), 6 line nodes (squares), and one interior node (triangle).</td>
<td>44</td>
</tr>
</tbody>
</table>
5.8 Argyris quintic element with 18 vertex nodes (circles) ............................................................ 45
5.9 Normalized Laplacian error norm over increasing mesh resolution ...................................................... 46
5.10 $TE(t)$ and $EN(t)$ over increasing mesh resolution. $U_{ms} = 0.0$, $R_e = 10^3$ ............................................. 48
5.11 External: $\psi(x,y)$ and $E_k(k)$. Time = 0.05 normalized units ................................................................. 50
5.12 External: $\omega(x,y)$ and $\zeta(k)$. Time = 0.05 normalized units ...................................................... 51
5.13 External: $\psi(x,y)$ and $E_k(k)$. Time = 0.20 normalized units ................................................................. 52
5.14 External: $\omega(x,y)$ and $\zeta(k)$. Time = 0.20 normalized units ................................................................. 53
5.15 External: $\psi(x,y)$ and $E_k(k)$. Time = 1.0 normalized units ................................................................. 54
5.16 External: $\omega(x,y)$ and $\zeta(k)$. Time = 1.0 normalized units ................................................................. 55
5.17 Internal: $\psi(x,y)$ and $E_k(k)$. Time = 0.05 normalized units ................................................................. 56
5.18 Internal: $\omega(x,y)$ and $\zeta(k)$. Time = 0.05 normalized units ................................................................. 57
5.19 Internal: $\psi(x,y)$ and $E_k(k)$. Time = 0.20 normalized units ................................................................. 59
5.20 Internal: $\omega(x,y)$ and $\zeta(k)$. Time = 0.20 normalized units ................................................................. 60
5.21 Internal: $\psi(x,y)$ and $E_k(k)$. Time = 1.0 normalized units ................................................................. 61
5.22 Internal: $\omega(x,y)$ and $\zeta(k)$. Time = 1.0 normalized units ................................................................. 62
CHAPTER 1 Introduction

Problem Overview and Solution Schemes

The intent of this paper is twofold and interdisciplinary in nature. First, of interest to the discipline of geophysical fluid dynamics, is the effect of mean shear on incompressible turbulent flow occurring on the surface of a rotating sphere. Mean shear is the difference in flow magnitude, in a direction perpendicular to the mean flow. The numerical simulations in [13] use the pseudospectral method (PSM). The governing partial differential equations (PDEs) are well known in the field of geophysical fluid dynamics [2, 9, 11]. Numerical solutions to the model in this study are also relevant to the field of magnetohydrodynamics (MHD) where the magnetic field influence has a similar physical effect to that of rotation in geophysical flows. Implementation of a finite element method (FEM) approach to solving the PDEs in this study will allow modeling of non-periodic mean shear profiles which could not be addressed using the spectral method (SM) or PSM approaches.

Second, of interest to the discipline of Computer Science, is the analysis of the efficiency with which the FEM solves this particularly challenging problem. While the SM and PSM are known to support global accuracy, finite elements support better local accuracy (see [6]). SM or PSM also typically use an explicit timestepping
scheme, as opposed to the usually implicit or semi-implicit schemes of the FEM [4]. The implicit or semi-implicit schemes tend to be more stable and allow for larger time-steps than a fully explicit scheme. Numerical stability is a property of numerical algorithms describing the ability of an algorithm to handle errors in input data. In a stable method, input errors are damped out of the system as computations proceed. In an unstable method, errors are retained and possibly magnified throughout the series of calculations.

As in spectral approaches or finite difference methods (FDM), the resolution of the discretized space has an impact on the stability of the FEM. With the FEM, increasing the number of elements in the mesh is similar to increasing the number of grid points in a PSM or FDM scheme. The choice of shape functions determines element type and correspondingly the degrees of freedom (DOFs) of the FEM. Experiments studying the effect of element type and mesh refinement on stability will be presented. Finally, this study implements the FEM using the application package Femlab®. Some advantages and disadvantages of this choice will be discussed.

**Current Research**

A purely spectral method uses trigonometric basis functions which form an orthogonal set to generate a system of ordinary differential equations (ODEs). The name pseudospectral implies that Fast Fourier Transforms (FFTs) are used only to calculate spatial derivatives [4]. Thus the contributions of one or more terms of the governing PDE is computed in real space. In [13], the nonlinear convolution term is solved in real space, and the remaining contributions are all added in Fourier space. Thus at each timestep, both a FFT and an inverse FFT must be performed. While
pseudospectral methods have been shown to be efficient in solving geophysical fluid flow problems [4], there are some limitations imposed by their very nature. In particular, the boundaries of the domain must be periodic. This is due to the fact that the basis functions are trigonometric. Therefore, studies using SM or PSM are restricted to mean shear profiles which are periodic in one dimension [12, 13].

The FEM treats all contributions in real space and does not require periodic boundary conditions. Thus the implementation in this study will allow research of mean shear profiles unapproachable with the spectral methods. To date, research using the FEM to simulate geostrophic turbulence is sparse. This study can help to explore the validity of this less common approach for modeling geostrophic turbulence.
CHAPTER 2  The Model

In this chapter, the model will be described in general. First the field equations and assumptions involved in the derivation will be defined and discussed. Second, the model domain will be defined. Finally, the boundary conditions for each method are derived.

Field Equations

The equations governing the motion of an incompressible fluid have been well known since the mid 1800s. The incompressible Navier-Stokes equations were derived separately by Claude Louis Marie Henri Navier, in France, circa 1821 and later by George Gabriel Stokes in England, circa 1845. These equations have been the subject of rigorous study and a large portion of work in the field of fluid dynamics ever since. In fact these equations are the topic of one of the six "Millennium Problems" posed by the Clay Mathematics Institute. Each of these problems rewards a million dollar bounty for finding the solution. While our study is based in principle on this famous set of equations, there are some modifications and assumptions which require elaboration.
Navier-Stokes Equations

Equations 2.1 and 2.2 are known as the Navier-Stokes equations for viscous incompressible fluids. Here \( u \) is the velocity field, \( \rho \) is the density, \( p \) is the pressure, \( \nu \) is the viscosity, and \( F \) is a vector valued forcing function. Equation 2.1 is a direct result of conservation of mass. It is often referred to as the incompressibility or divergence-free condition. Equation 2.2 is known as the Navier-Stokes equation and is derived from conservation of momentum and energy. Most introductory graduate level texts on fluid dynamics contains a complete derivation of these equations for example [5].

\[
\nabla \cdot \mathbf{u} = 0 \quad (2.1)
\]

\[
\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} + \mathbf{F} \quad (2.2)
\]

The \( \beta \)-plane Approximation

One major characteristic of geostrophic flow is that this flow occurs on the surface of a rotating sphere. The Coriolis force is a fictitious force arising from the choice of a rotating framework of reference (see [2].) The Earth can be considered a sphere rotating about its North-South pole axis. Consider a reference frame rotating with the Earth, where the \( x \)-axis points eastward or zonally, the \( y \)-axis points northward or meridionally, and the \( z \)-axis points vertically. We define the two dimensional velocity field \( \mathbf{u} \), with zonal component \( u \) and meridional component \( v \). \( \Omega \) is the Earth's rotation vector and has the form,

\[
\Omega = (\Omega \cos \varphi) \hat{y} + (\Omega \sin \varphi) \hat{z} \quad (2.3)
\]
where $\varphi$ is the latitude measured northward from the equator. Note that the component of the Earth's rotation vector along the local vertical changes with latitude.

The total time derivative referred to in some texts as the material derivative is $\frac{du}{dt} = \frac{\partial u}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u}$. The absolute acceleration in the rotating frame, minus the centrifugal component is,

$$\frac{du}{dt} + 2\Omega \times \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} + \mathbf{F}.$$

In two dimensions only the local vertical component of the Earth's rotation vector contributes to the dynamics. In terms of its components, equation 2.2 for geostrophic flow is,

\begin{align}
\frac{du}{dt} - f v &= -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \frac{\partial^2 u}{\partial x^2} + F_x \\
\frac{dv}{dt} + f u &= -\frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \frac{\partial^2 v}{\partial y^2} + F_y,
\end{align}

where $f = 2\Omega \sin \varphi$ is the Coriolis parameter.

To simplify the model we employ the $\beta$-plane approximation. This approximation is appropriate for large-scale, mid-latitude flows on the Earth's surface [2, 9]. It allows the modeling of geostrophic flow in a two dimensional domain as opposed to three dimensions, significantly reducing the computational complexity of the problem. For large scale waves, which span several degrees of latitude we must consider the effective change in the meridional contribution of the Coriolis parameter. Beginning with a reference latitude $\varphi_0$, and adding a small departure in the northward direction of the form $\frac{\varphi}{R_E}$, where $R_E$ is the radius of the Earth ($\approx 6371$ km). We can expand
the Coriolis parameter using a Taylor series and retaining the first two terms:

\[ f = 2\Omega \sin \varphi_0 + \frac{2\Omega y}{R_E} \cos \varphi_0 + \ldots, \]

or equivalently by using a small angle approximation, and the sum of angles identity,

\[
2\Omega \sin \left( \varphi_0 + \frac{y}{R_E} \right) = 2\Omega \left[ \sin \varphi_0 \cos \left( \frac{y}{R_E} \right) + \cos \varphi_0 \sin \left( \frac{y}{R_E} \right) \right]
\]

\[
\approx 2\Omega \sin \varphi_0 + \frac{2\Omega y}{R_E} \cos \varphi_0.
\]

Standard notation is to set \( f_0 = \sin \varphi_0 \) as the reference Coriolis parameter and \( \beta_0 = \frac{2\Omega}{R_E} \cos \varphi_0 \) as the \( \beta \) parameter. In this notation, the horizontal motion given by equations 2.4 and 2.5 is

\[
\frac{du}{dt} - (f_0 + \beta_0 y) v = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \frac{\partial^2 u}{\partial x^2} + F_x, \quad (2.6)
\]

\[
\frac{dv}{dt} + (f_0 + \beta_0 y) u = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \frac{\partial^2 v}{\partial y^2} + F_y. \quad (2.7)
\]

Again this approximation is valid only at mid-latitudes where the \( \beta_0 y \) term is small relative to the leading \( f_0 \) term. In terms of the meridional length scale \( L \), that is,

\[
\beta = \frac{\beta_0 L}{f_0} \ll 1. \quad (2.8)
\]

A more rigorous development of the \( \beta \)-plane approximation can be found in [9].
Streamfunction - Vorticity Formulation

The vorticity is the curl of the velocity field and is an indication of the rotational nature of the flow. In two dimensions this is a scalar quantity and is given by

\[ \omega = \mathbf{\hat{z}} \cdot \nabla \times \mathbf{u} = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \quad (2.9) \]

This study is restricted to barotropic flows by making the assumption that we can write the pressure as a function of the density alone \( p(\rho) \). Barotropic flows are those in which the pressure is determined strictly by the density. This is in contrast to baroclinic flows where the pressure is determined from both density and temperature. In barotropic flows contours of equal pressure are parallel to contours of equal density so that,

\[ \nabla p \times \nabla \rho = 0, \quad (2.10) \]

and since the curl of a gradient is always equal to zero,

\[ \nabla \times \left( \frac{1}{\rho} \nabla p \right) = \frac{1}{\rho} (\nabla \times \nabla p) - \frac{1}{\rho^2} \nabla p \times \nabla \rho \]

\[ = \mathbf{0}. \]

By following this assumption when taking the curl of equation 2.2, using equations 2.6 and 2.7 for the acceleration in the rotating frame, and noting that the pressure is determined by the incompressibility condition, our evolution equation becomes

\[ \frac{\partial \omega}{\partial t} - \beta_0 u + u \cdot \nabla \omega = \nu \nabla^2 \omega + \mathbf{\hat{z}} \cdot \nabla \times \mathbf{F}. \quad (2.11) \]
Finally, since the flow is divergence free we can introduce the streamfunction $\Psi$ defined as

$$ (u, v) = \left( \frac{\partial \Psi}{\partial y}, -\frac{\partial \Psi}{\partial x} \right). \quad (2.12) $$

Notice that $u = \nabla \times \Psi \hat{z}$. Let $F_\omega = \hat{z} \cdot \nabla \times F$. The two dimensional velocity equation is

$$ \frac{\partial}{\partial t} \omega + \left[ (\nabla \times \Psi \hat{z}) \cdot \nabla \right] \omega + \beta \frac{\partial}{\partial x} \Psi = \nu \nabla^2 \omega + F_\omega, \quad (2.13) $$

where,

$$ -\nabla^2 \Psi = \omega. \quad (2.14) $$

**Non-dimensionalization**

In order to simplify the treatment of units, let the characteristic scales of motion be given by a velocity scale $U$, a length scale $L$, and a time scale $\frac{U}{f}$ which are uniform over our problem domain. Introduce the following dimensionless quantities

$$ \omega = \frac{U}{L} \tilde{\omega}, \quad \Psi = U L \tilde{\Psi}, \quad \nabla = \frac{1}{L} \tilde{\nabla}, \quad t = \frac{U}{f} \tilde{t}. $$

Using these in equation 2.13, a dimensionless form of the evolution equation suitable for numeric simulation can be obtained. For simplicity now drop the forcing function $F_\omega$ and assume that when reintroduced its form is appropriately scaled.

$$ \frac{U^2}{L^2} \frac{\partial \tilde{\omega}}{\partial \tilde{t}} + \frac{U^2}{L^2} \left[ \left( \nabla \times \tilde{\Psi} \hat{z} \right) \cdot \tilde{\nabla} \right] \tilde{\omega} + \beta U \frac{\partial \tilde{\Psi}}{\partial \tilde{x}} = \frac{\nu U}{L^3} \tilde{\nabla}^2 \tilde{\omega} \quad (2.15) $$
This is equivalent to (hereafter, the tilde notation will be dropped on the dimensionless quantities)

$$\nabla^2 \omega = \frac{UL}{\nu} \frac{\partial}{\partial t} \omega + \frac{UL}{\nu} [(\nabla \times \Psi \mathbf{z}) \cdot \nabla] \omega + \frac{\beta L^3}{\nu} \frac{\partial}{\partial x} \Psi$$

(2.16)

In this model the Reynolds number is defined as the ratio of inertial forces to viscous forces, \( R_e = \frac{UL}{\nu} \), and the Ekman number is defined as the ratio of viscous forces to the force arising from the Coriolis effect, \( E_k = \frac{\nu}{\beta L^3} \) [2]. We can write a two parameter dimensionless evolution equation as,

$$\nabla^2 \omega = R_e \frac{\partial}{\partial t} \omega + R_e [(\nabla \times \Psi \mathbf{z}) \cdot \nabla] \omega + \frac{1}{E_k} \frac{\partial}{\partial x} \Psi$$

(2.17)

For large values of \( R_e \), the magnitude of viscous effects is small relative to the advection forces. In other words damping due to viscous interactions occurs at rates such that, even at the smallest spatial scales (recall the viscous term \( \nu \nabla^2 \omega \)), the solution is not dominated by the viscosity. As we decrease \( R_e \) viscous effects play a larger role in the overall dynamics and energy is damped out of the system at a faster rate. The Ekman number determines the strength of the viscous effects relative to the Coriolis effects. As we decrease \( E_k \) the meridional variation in the local vertical component of the Earth’s rotation vector plays a larger role in the dynamics.

**Model Domain**

Our two parameter, dimensionless evolution equation allows us to restrict the domain of this model to a square with side length \( L \). By allowing for varying values of Reynolds and Ekman numbers, we can implement a flexible model which allows for a variety of length and mean velocity scales. Figure 2.1, shows the normalized model domain \( \Omega \) and boundaries \( d\Omega \).
In this study the value of $E_k$ is varied from $10^{-4}$ to $10^{-6}$. The value of $R_e$ in this study is varied from a lower limit of $R_e = 10^3$ to a maximum of $R_e = 10^5$. Ideally, we would like to achieve a Reynolds number $\approx 10^6$ or $10^7$. However, numerically speaking this is unfeasible. The difficulty arises from the fact that large values of $R_e$ result in a stiff ODE system (see chapter 3). Stiff systems of ODEs are those in which the solution has time scale dependencies which vary greatly. For increasingly large values of $R_e$ it becomes increasingly difficult to solve the ODE system. Similarly for decreasing values of $E_k$ the ODE system becomes increasingly stiff.

**Boundary Conditions**

Figure 2.2 shows the labeling scheme for the following discussion of boundary conditions. Boundary conditions for our model are specified as either Neumann boundaries or periodic boundaries. For boundaries specified as Neumann, or *natural* boundary conditions, this means we specify the of the value of the outward normal derivatives.
of $\omega$ and $\psi$ on each boundary. For periodic boundary conditions we simply ensure that the values of $\omega$ and $\psi$ on opposing boundaries are equal, and that contributions to the future approximations on opposing boundaries are appropriately weighted.

For the PSM, periodic boundary conditions in both the $x$ and $y$ directions are used. That is boundary 1 maps to boundary 4, and boundary 2 maps to boundary 3. For the FEM we enforce periodic boundary conditions in the $x$ direction, mapping boundary 1 to boundary 4. In the $y$ direction the FEM maintains fixed values of both of the normal derivatives of $\Psi$ and $\omega$. That is on boundaries 2 and 3 the normal derivatives of the stream function and vorticity have a constant value. The following table shows the settings on each of the boundaries. Other models may require Dirichlet, or essential boundary conditions which require the specification of the value of the solution itself on each boundary.
<table>
<thead>
<tr>
<th>Boundary</th>
<th>$\omega$</th>
<th>$\Psi$</th>
<th>$\frac{\partial \omega}{\partial x}$</th>
<th>$\frac{\partial \omega}{\partial y}$</th>
<th>$\frac{\partial \Psi}{\partial x}$</th>
<th>$\frac{\partial \Psi}{\partial y}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>periodic</td>
<td>periodic</td>
<td>periodic</td>
<td>*</td>
<td>periodic</td>
<td>*</td>
</tr>
<tr>
<td>2</td>
<td>0</td>
<td>0</td>
<td>*</td>
<td>0</td>
<td>*</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>0</td>
<td>0</td>
<td>*</td>
<td>0</td>
<td>*</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>periodic</td>
<td>periodic</td>
<td>periodic</td>
<td>*</td>
<td>periodic</td>
<td>*</td>
</tr>
</tbody>
</table>
CHAPTER 3 Numerical Methods

In this chapter we will discuss the key characteristics which separate the PSM and FEM. The PSM, and FEM can both be classified as *series-expansion* methods. This general classification is due to the common goal of each method to approximate the spatial dependence of the solution with the linear combination of a finite number of predetermined expansion functions [4]. This discretization process transforms the governing PDE into a system of ODEs. Consider the following general equation in two dimensions where $F$ is an operator and $\phi$ the unknown function of interest,

$$\frac{\partial \phi}{\partial t} + F(\phi) = 0 \quad (3.1)$$

Both the PSM and FEM find approximate solutions to our governing equations, which have the following form.

$$\Phi(x, y, t) = \sum_{k=1}^{N} a_k(t) \varphi_k(x, y) \quad (3.2)$$

$\varphi_1, \ldots, \varphi_N$, are the expansion functions. The linear combination of these expansion functions comprise a system of ODEs which can be solved for the unknown coefficients $a_k$. 
Defining the residual as follows,

\[ R(\Phi) = \frac{\partial \Phi}{\partial t} + F(\Phi), \tag{3.3} \]

we have a quantity for which the best choice of expansion coefficients \( a_1, \ldots, a_N \), leads to the minimal deviation from the true solution or \( R(\phi) \). This process of finding the "right" expansion coefficients is called "minimizing the residual".

The PSM uses the collocation strategy for minimizing the residual, which requires the residual to be zero at a discrete set of points in the problem domain. The FEM can use the collocation strategy, but more often uses the Galerkin method, which requires the integral of the product of the residual and the expansion functions (applied for weighting) to be zero. In other words, collocation is a pointwise constraint on the solution, and the Galerkin method is a weighted integral, domain averaging constraint. These two methods are discussed in more depth in subsequent sections.

**Pseudospectral Method**

The PSM and SM are closely related because both use Fourier series to form an orthogonal set of expansion functions. In discussing the PSM it is beneficial to discuss the SM and the Discrete Fourier Transform (DFT).

**Spectral Methods**

Discretization of the governing equations on a wavenumber domain allows spectral methods to take advantage of the Discrete Fourier Transform (DFT). Wave phenomena are difficult to quantify in a real space regime. Fourier space is a much more
tractable way to study wave behavior. The DFT uses a linear combination of cosine and sine functions to decompose sampled data into its component Fourier modes. The resulting coefficients of the Fourier series reveal which modes are present and in what magnitudes [6].

**Fourier Transforms**

Given a periodic real valued function \( f \) uniformly sampled at \( N \) points, the DFT resolves \( f \) into its wavenumber components for \( k = -(N/2), \ldots, (N/2) \). The maximum resolvable mode, \( k = (N/2) \) for a given sampling rate is called the Nyquist frequency. The DFT requires \( O(N^2) \) time to perform the matrix-vector multiplication in order to calculate the coefficients of the Fourier series. The Fast Fourier Transform (FFT) uses a divide and conquer approach to calculate the DFT of an \( N \) point sequence by breaking it into two DFTs of length \( N/2 \), then further into four DFTs of length \( N/4 \), and so on. This results in the total cost of performing an FFT on an \( N \) point sequence being \( O(N \log_2 N) \).

**Collocation**

The collocation approximation requires that the residual be zero at a discrete set of points in the problem domain. Figure 3.1 shows a uniformly spaced grid in two dimensions. At each of these gridpoints the following must hold,

\[
R(\Phi(x_i, y_j)) = 0
\]  

(3.4)

for all \( i, j = 1, \ldots, M \) where \( M = 2N \).
Figure 3.1 Uniformly spaced 2-d grid, $\Delta x = \Delta y$.

**Pseudospectral vs. Spectral**

Because the PSM uses the collocation approximation, the series expansion functions are only used to calculate spatial derivatives. This requires two FFTs at each timestep as opposed to the single FFT required by the SM. The second extra FFT takes the Fourier series back to real space as follows,

$$
\Psi(x, y) = \sum_{i=-kx_{\text{max}}}^{kx_{\text{max}}} \sum_{j=0}^{ky_{\text{max}}} \left[ \Psi^*_R(i,j) \cdot \cos \left( \frac{2\pi ix}{L} + \frac{2\pi ju}{L} \right) \right. \\
\left. + \Psi^*_S(i,j) \cdot \sin \left( \frac{2\pi ix}{L} + \frac{2\pi ju}{L} \right) \right]
$$

(3.5)

The extra work required in the transformation back to real space is well worth the resources saved from calculating the contribution of the nonlinear convolution term.
The \( \mathbf{u} \cdot \nabla \omega \) term in equation 2.11 is the nonlinear convolution term. In Fourier space this calculation requires \( O(N^4) \) operations, but in real space can be calculated using standard finite differences \( O(N^2) \) operations.

**Finite Element Method**

The application package Femlab\textsuperscript{\textregistered} was used to implement the FEM for this study. Many of the following subsections are relevant to most or all finite element implementations, however some are specific to the particular application package used here.

Regardless of the implementation, the goal of the FEM is to approximate the solution with expansions function which are nonzero only over some localized portion of the domain. The FEM discretizes a real space domain using geometric shapes called elements. In two dimensions elements are most often triangles, but can be quadrilaterals. In three dimensions elements are tetrahedrons or cubes.

**Galerkin Method**

The Galerkin method of minimizing the residual requires the residual to be orthogonal to each of the expansion functions. That is,

\[
\int_S R(\Phi(x_i, y_j)) \varphi_k(x_i, y_j) \, dA = 0
\]

for all \( i, j = 1, \ldots, M \).

**Mesh Generation**

Femlab\textsuperscript{\textregistered} uses the Delaunay mesh generation algorithm. Delaunay triangulation in two dimensions on a set of \( N \) points requires that the circle containing the vertices
of any single triangle is empty of all other points in the set. The divide-and-conquer versions of Delaunay triangulation have a worst case complexity of $O(N \log N)$. In this model the mesh is generated from a single constraint parameter of the maximum edge length. This parameter is specified as a ratio of the length of the longest dimension. Figure 3.2 shows the mesh generated from a specified maximum edge length ratio of $h_{\text{max}} = 1$. The mesh contains four elements, all of which are boundary elements. There are five unique element vertices. Figure 3.3 shows the mesh generated from a specified maximum edge length ratio of $h_{\text{max}} = 1/2$. The mesh contains sixteen elements, eight of which are boundary elements. There are thirteen unique element vertices.
Figure 3.3  Triangular mesh, maximum edge length = $\frac{L}{2}$. 
CHAPTER 4  FEM Implementation

In this chapter the implementation of the FEM is discussed. Topics include a brief introduction of the software Femlab®, the 5 steps to creating a model of this type in Femlab®, and finally the specific steps taken to implement this study using Femlab®.

Femlab®

Femlab® is a powerful, interactive environment for modeling problems based on PDE's, using the FEM. Femlab® is the product of Comsol Inc. Comsol has regional offices in the United States and many European countries. The original version of Femlab® was released in 1999 [7]. The version used in this study was Femlab®3.0a.

The original release of Femlab® required the software package MATLAB distributed by Mathworks Inc. as many of the numerical algorithms were accessed via MATLAB. Later releases, (3.0 and on) allow for using Femlab® as a stand alone package or via the MATLAB scripting language. Both methods access identical routines written specifically for Femlab® in C++[7].

Femlab® provides two environments for working with PDE based models. First, users can specify model characteristics via a user friendly, graphical user interface (GUI). This is advantageous to users who are unfamiliar with programming techniques and prefer not to learn complicated function syntax and scripting semantics.
Second, users can implement a model in the MATLAB scripting language, which calls the necessary Femlab® routines used in any particular model. The motivation for using Femlab® in this study was the fact that one can use the graphical user interface to define a model, and subsequently save the model as a scripted sequence of MATLAB commands. The idea being that the time to formulate the model in Femlab® as opposed to creating the model from scratch in MATLAB, or some other low-level language would be drastically reduced. Furthermore, the ability to use MATLAB scripts to create the model would allow for hands free execution of multiple parameterized simulations.

Femlab® affords users the ability to implement a PDE based model via a standardized environment. Femlab® provides “application modes” which allow users in specific fields to specify model characteristics using relevant physical quantities with the underlying mathematical equations predefined for each application mode. Application modes include Chemical Engineering, Heat Transfer, Electromagnetics, and Structural Mechanics, along with many others. These fields specific applications modes are provided in addition to more generic modes which allow the specifications of custom PDEs. The specification of our model using the streamfunction-vorticity formulation does not allow us to use the related Navier-Stokes application mode. We instead use the time dependent, general form PDE formulation, in two dimensions on two dependent variables. This allows the specification of custom PDEs not available in field specific application modes.
Five Steps to a Femlab® Model

There are five steps to implementing a model using Femlab®. In order they are; drawing a geometry of interest, creating a mesh of elements across the geometry, defining the governing physics over the subdomain and boundaries, solving the ODE system, and postprocessing the results. In this section each step is discussed in turn.

**Geometry**

Specifying the geometry of interest in Femlab®can be accomplished via the GUI or a set of predefined routines. In two dimensions the user can draw points, lines, rectangles, circles, Bezier curves etc. The simple square geometry in this study does not require many of the advance techniques for geometry definition supplied by Femlab®, such as importing of Matlab contours or assorted CAD formats.

**Mesh Generation**

The automated mesh generation in Femlab®is one of its main advantages to this study. As mentioned in chapter 3, the primary parameter for generating a mesh is the maximum element edge length. Femlab®also allows for easy mesh refinement. More complicated features that were not applied to this study include refinement of the mesh in specific sections of the geometry designated by the user, or adaptive mesh refinement determined as necessary by Femlab®’s adaptive solver.

**Physics Definition**

This step involves the definition of the custom PDEs which govern the behavior of the model. Separate definitions on the subdomain (or interior) and boundary
sections of the geometry are required. For this study the subdomain consists of all of the elements not located on the boundary.

**Solving the ODE system**

Femlab® provides several solvers for different model types including time dependent, linear stationary, nonlinear stationary, linear and nonlinear parameterized stationary, eigenvalue, and adaptive solvers for use on stationary problems with adaptive mesh refinement. For various application modes a default solver is designated. For this study, the time dependent solver is used.

**Postprocessing Results**

Femlab® provides many features for postprocessing of data. Available plots include surface, contour, arrow, streamline, and others. Femlab® can also create animations of any of the basic plot types. The user can also specify various expressions to be integrated over the subdomain and/or boundaries. These integral values can then be visualized using Femlab®'s postprocessing plots types.

**Model Specific Implementation**

To describe our implementation it is helpful to briefly discuss how to formulate a custom PDE based model in Femlab®. Begin with the domain of interest. Recall the unit square domain $\Omega$ with boundary $d\Omega$. Femlab®’s general form allows the specification of governing equations as a system of PDEs in space and time. This is referred to in literature as the **strong** form. In the following sections the definition of the model without the presence of mean shear is given. This is followed by a section
which outlines the modifications necessary to impose the presence of both an external and internal mean shear.

**General form PDE**

Femlab®'s general form PDE for two dependent variables in two dimensions is,

$$\mathbf{d}_a \frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot \mathbf{\Gamma} = \mathbf{F} \quad \text{on} \quad \Omega. \quad (4.1)$$

Here \( \mathbf{u} \) is the solution vector, \( \mathbf{\Gamma} \) is the flux vector, \( \mathbf{F} \) is the source term vector, \( \mathbf{d}_a \) is the mass coefficient matrix, and \( \Omega \) is the subdomain. For this study Neumann boundary conditions are specified as,

$$-\mathbf{n} \cdot \mathbf{\Gamma} = \mathbf{G} \quad \text{on} \quad d\Omega \quad (4.2)$$

where \( \mathbf{n} \) is the outward normal on the boundary \( d\Omega \).

**Subdomain Definition**

To cast equation 2.17 into Femlab®'s general form rewrite as,

$$R_e \frac{\partial \omega}{\partial t} - \nabla^2 \omega = -R_e \left[ (\nabla \times \Psi \hat{z}) \cdot \nabla \right] \omega - \frac{1}{E_k} \frac{\partial}{\partial x} \Psi. \quad (4.3)$$

Recall that \( -\nabla^2 \Psi = \omega \) and note that \( -\nabla^2 \omega \) is \( \nabla \cdot -\nabla \omega \). Thus for this study we define the solution vector as,

$$\mathbf{u} = \begin{bmatrix} \omega \\ \Psi \end{bmatrix}. \quad (4.4)$$
Define the flux vector as,

$$\Gamma = \begin{bmatrix} \frac{\partial \omega}{\partial x} & \frac{\partial \omega}{\partial y} \\ \frac{\partial \psi}{\partial x} & \frac{\partial \psi}{\partial y} \end{bmatrix}. \quad (4.5)$$

Notice that the convolution term is

$$[(\nabla \times \psi \hat{z}) \cdot \nabla] \omega = - \left( \frac{\partial \psi}{\partial x} \frac{\partial \omega}{\partial y} - \frac{\partial \psi}{\partial y} \frac{\partial \omega}{\partial x} \right).$$

Define the load vector as,

$$\mathbf{F} = \begin{bmatrix} -R_e \cdot \left( \frac{\partial \psi}{\partial x} \frac{\partial \omega}{\partial y} - \frac{\partial \psi}{\partial y} \frac{\partial \omega}{\partial x} \right) - \frac{1}{E_k} \frac{\partial \psi}{\partial x} \\ \omega \end{bmatrix}. \quad (4.6)$$

Finally define the mass coefficient matrix,

$$\mathbf{d}_a = \begin{bmatrix} R_e & 0 \\ 0 & 0 \end{bmatrix}. \quad (4.7)$$

Then the entire system over each element is,

$$\begin{bmatrix} R_e & 0 \\ 0 & 0 \end{bmatrix} \cdot \begin{bmatrix} \frac{\partial \omega}{\partial t} \\ \frac{\partial \psi}{\partial t} \end{bmatrix} + \nabla \cdot \begin{bmatrix} -\frac{\partial \omega}{\partial x} & -\frac{\partial \omega}{\partial y} \\ -\frac{\partial \psi}{\partial x} & -\frac{\partial \psi}{\partial y} \end{bmatrix} = \begin{bmatrix} -R_e \cdot \left( \frac{\partial \psi}{\partial x} \frac{\partial \omega}{\partial y} - \frac{\partial \psi}{\partial y} \frac{\partial \omega}{\partial x} \right) - \frac{1}{E_k} \frac{\partial \psi}{\partial x} \\ \omega \end{bmatrix}. \quad (4.8)$$

Due to the fact that only the vorticity has a time dependent term, this system is classified as a differential algebraic equation (DAE). The fact that it takes one derivation on the first equation and one derivation on the second equation to find the time dependence of $\Psi$ further classifies this system as an index 2 DAE.
Boundary Definition

Strictly Neumann boundary conditions in Femlab®'s general form are defined as follows

\[-\mathbf{n} \cdot \Gamma = G\]  \hspace{2cm} (4.9)

Since we have already defined the flux vector \(\Gamma\), we simply need to specify the value of the outward normal component of the gradient of both the vorticity and stream-function on each boundary (see figure 2.2). For boundaries 1 and 4 at \(x = 0\) and \(x = 1\), we enforce periodic boundary conditions. In Femlab® periodic boundary conditions imply zero flux across the boundaries as these boundaries are not considered boundaries at all, but rather as a single set of DOFs which reflect contributions to one another on opposite ends of the domain.

For boundaries 2 and 3 we must specify \(G\) in equation 4.9. Without the presence of a mean shear a no-slip condition on the zonal component of the velocity is enforced. Recall \(u = \frac{\partial \psi}{\partial y}\). The normal derivatives of both the streamfunction and vorticity are set zero along the boundaries,

\[\frac{\partial \psi}{\partial y} = 0,\]

and

\[\frac{\partial \omega}{\partial y} = 0,\]

Constraint Requirements

Femlab® requires some constraint to be set in the model domain. In personal communication with Femlab® customer support we were instructed to ensure that at some places in the model we constrain the solution to some value. Therefore, the value of both the streamfunction and the vorticity is constrained to some constant value
(usually zero) along boundaries 2 and 3. This is the main reason why according to Femlab® we can't model periodic boundaries in both the $x$ and $y$ directions. This would have allowed comparison of the solutions obtained by the PSM and FEM without the presence of mean shear.

### External vs. Internal Mean Shear

The preceding section defines the model without the presence of mean shear. In this section the modifications required to impose the effects of first an external mean shear, and second an internal mean shear are outlined. An externally imposed mean shear does not allow the mean shear to evolve in time via feedback from the background flow. An internal mean shear is considered part of the flow and can evolve over time along with the background flow as imposed by the governing equations.

#### External

The presence of an externally imposed mean shear can be added to the prior model definition by adding it’s contribution to the source vector. That is, the contribution of the mean shear is added to the right hand side of equation 2.11. Consider a linear zonal mean shear, $u_{ms} = (U_{ms}y)\hat{y}$, where $U_{ms}$ is the maximum mean shear amplitude. In terms of the stream function we need to define a function which has $\nabla \Psi = (U_{ms}y)\hat{y}$. This is,

$$\Psi_{ms}(x,y) = y^2 \frac{U_{ms}}{2}. \quad (4.10)$$

The contribution to the dynamics of the model is added by incorporating an appropriately scaled term to the vorticity component of the source vector. Recall the nonlinear convolution term in equation 2.11, $u \cdot \nabla \omega$. Applying this to the linear mean
zonal flow we find,
\[ u_{ms} \cdot \nabla \omega = U_{ms} y \frac{\partial \omega}{\partial x}. \]

Scaling appropriately for non-dimensionalization, the load vector is
\[ F = \begin{bmatrix}
-Re \left( \frac{\partial \psi}{\partial x} \frac{\partial \omega}{\partial y} - \frac{\partial \psi}{\partial y} \frac{\partial \omega}{\partial x} \right) - \frac{1}{E_k} \frac{\partial \psi}{\partial x} - Re \cdot U_{ms} \cdot y \frac{\partial \omega}{\partial x} \\
\omega
\end{bmatrix}. \]

**Internal Mean Shear**

The presence of an internally imposed mean shear can be added to the prior model definition by first adding appropriate contributions to the initial conditions of the streamfunction and vorticity. Second an adjustment of the Neumann boundary conditions on boundaries 2 and 3 is required. Finally adjustment of the constraint values for the streamfunction and vorticity on boundaries 2 and 3 is necessary.

**Initial Conditions**

Given an initial background flow \( \Psi_0 \) and associated vorticity \( \omega_0 \), add the zonal mean shear to obtain the initial conditions for the entire flow. That is, for \( u_{ms} = (U_{ms} y) \dot{y} \) we have
\[ \Psi_{init} = \Psi_0 + \left( U_{ms} \cdot \frac{y^2}{2} \right) \quad (4.11) \]
and
\[ \omega_{init} = \omega_0 - U_{ms}. \quad (4.12) \]

**Neumann Boundaries**

With this mean shear profile notice that on boundary 2 where \( y = 0 \), \( u_{ms} = 0 \) so the value of the normal derivative of both the stream function and vorticity is zero.
However on boundary 3 where \( y = 1 \), \( u_{ms} = U_{ms} \hat{y} \) so the normal derivative of the streamfunction is \( U_{ms} \) and the normal derivative of the vorticity is zero.

**Constraints**

On boundary 2 the value of both the streamfunction and vorticity are constrained to zero as is appropriate for \( y = 0 \). On boundary 3 the value of the streamfunction is set to \( \Psi(x, 1) = \frac{U_{ms}}{2} \), and the value of the vorticity to \( \omega(x, 1) = -U_{ms} \).

**Quantities of Interest**

In this section, various quantities of interest are defined and discussed. In the limit of \( Re \rightarrow \infty \), and no forcing, the first two quantities are conserved. Thus they can provide insight into the accuracy of the numerical model. The second two quantities provide information on the dominating characteristics of a run.

- **Total Energy**

  The total kinetic energy of the system at time \( t \) in real space is,

  \[
  TE(t) = \int_{\Omega} \frac{1}{2} (\nabla \Psi)^2 \, dA. \tag{4.13}
  \]

  For runs over increasing values of \( Re \) we expect this quantity to be more readily conserved. Another way to look at the energy is in Fourier space. For a given time \( t \) the distribution of energy amongst the component modes in Fourier space is of importance. First, the FEM solution for \( \psi(x, y) \) is interpolated onto a regularly gridded mesh of size \( (2 \times k_{max} \times 2 \times k_{max}) \) where \( k_{max} \) correspond to the Nyquist frequency, in the \( x \) and \( y \) directions. The two dimensional FFT is performed and normalized to \( 1/(k_{max})^2 \). Given the streamfunction over two
Dimensional Fourier space $\psi(k_x, k_y)$, the energy in the $k^{th}$ mode is calculated as the cumulative sum over all $(k_x, k_y)$ where $(k - 1) < \sqrt{k_x^2 + k_y^2} \leq k$. As an integral equation that is,

$$E_k(k) = \int_d \frac{1}{2} (\nabla \Psi)^2 \, dK.$$  \hspace{1cm} (4.14)

Here the area of integration $d$, is the grey annulus shown in figure 4.1.
• **Enstrophy**

The enstrophy of the system is a measure of the transport of vorticity in and out of the system, and is defined by,

\[
EN(t) = \int_{\Omega} \omega^2 \, dA. \tag{4.15}
\]

Again, for runs over increasing values of \( R_e \) this quantity is expected to be more readily conserved. Also, similar to the energy in \( k \)-space the same strategy is used to calculate the enstrophy in \( k \)-space. Here the vorticity is interpolated, then the two dimensional FFT is performed. Only the integrand changes as,

\[
\zeta(k) = \int_{d} \omega^2 \, dK. \tag{4.16}
\]

• **Averaged Meridional Velocities**

By integrating the squared meridional velocities over the entire domain the amount of energy transported to and from the mean meridional flow is quantified.

\[
MV(t) = \int_{\Omega} |v| \cdot |v| \, dA \tag{4.17}
\]

• **Averaged Zonal Velocities**

Similarly, by integrating the squared zonal velocities over the entire domain the amount of energy transported to and from the mean zonal flow is quantified.

\[
MU(t) = \int_{\Omega} |u| \cdot |u| \, dA \tag{4.18}
\]
CHAPTER 5 Numerical Experiments

In this chapter the numerical experiments are presented. All of the experiments in this chapter are performed on a single set of initial conditions for the streamfunction and vorticity. The tests of the Femlab® implementation of the model include tolerance sensitivity, and two types of degree of freedom sensitivity. This chapter concludes with application to the two types of linear mean shear implementation.

Initial Conditions

All of the experiments in this chapter use the same sets of initial conditions for the background flow component of the streamfunction and vorticity. These initial conditions were chosen to match the various constraints imposed by the governing equations. First initial conditions for both the streamfunction and vorticity must satisfy \( \omega = -\nabla^2 \Psi \). Using combinations of cosine and sine functions for the initial conditions of \( \Psi \), analytic values for \( \omega \) with this characteristic are easily obtained. Second both the values of the streamfunction and vorticity must vanish at boundaries 2 and 3 (see figure 2.2). Again, using particular trigonometric functions this is trivial. Finally, the normal derivatives of both the stream function and vorticity must be zero at boundaries 2 and 3. To meet this constraint the product of four sine functions in the \( y \)-direction are used. This ensures that the first three derivatives of \( \psi \) with respect to \( y \) are zero at boundaries 2 and 3. Let the initial conditions of the streamfunction
be $\Psi_0$ and the vorticity be $\omega_0$. Consider the following,

$$\Psi_0(x, y) = c(x)f(y),$$

then

$$\omega_0(x, y) = -\nabla^2\Psi = -\left(\frac{\partial^2 c}{\partial x^2} f + c\frac{\partial^2 f}{\partial y^2}\right).$$

For this study initial conditions have $c(x) = \cos(16\pi x)$ and $f(y) = \prod_{k_y=1}^{k_y=4} \sin(2\pi k_y y)$. Using basic calculus it is straightforward to show that each of the requirements listed above is met. Notice that any function $c(x)$ which is twice differentiable with respect to $x$ could be used. This particular $c(x)$ is chosen for simplicity. Figure 5.1 shows the initial conditions for both the stream function and vorticity, along with the corresponding energy and enstrophy distributions in $k$-space.

**Tolerance Sensitivity**

In order to check sensitivity of the FEM model to tolerance settings the initial conditions were allowed to evolve both in and out of the presence of a linear mean shear. The ODE solver accepts as input both absolute and relative tolerance settings. The absolute and relative tolerance settings determine the limit for the error estimated by the solver at each timestep [7]. Subsequently, if the solver finds that the estimated error exceeds either or both of these limits, the solver decreases the size of the timestep and recomputes the solution until it is assured it has an error estimate within the specified limits. In this section the sensitivity of the total energy and enstrophy to the tolerance settings is examined.
Figure 5.1 Initial streamfunction and energy distribution.
Absolute vs. Relative Tolerance

The local error $e_i$, or error in the solution at each degree of freedom $u_i$, of the ODE system must obey the following,

$$e_i \leq \max ((\text{rtol} \cdot |u_i|), \text{atol}) .$$  \hspace{1cm} (5.1)

The absolute tolerance, "atol" is the limit imposed on the estimated error of the solution in the same units as the solution. The relative tolerance, "rtol" is the limit imposed on the estimated error as a ratio of the error to the solution itself. In simple terms the absolute tolerance is used to limit local error anywhere where the absolute tolerance is smaller than the product of the relative tolerance and the absolute value of the solution. In other words, when the solution at a given degree of freedom is close to zero the absolute tolerance is used to constrain the error. Otherwise the relative tolerance is used.

Experiments

The ODE system solver used in following tests is Femlab®'s time dependent solver. This solver is based on DASPK [10], and uses a Newton-like method. At each timestep a linear system solver is employed by the time dependent solver. The linear system solver used here is the direct solver UMFPACK [3]. UMFPACK is appropriate for stiff, symmetric or unsymmetric systems of linear equations. This is the default linear system solver for the two dimensional, time dependent, general form PDE in Femlab®. Figures 5.2 - 5.4 are the result of simulations run on a single set of initial conditions. The duration of the runs is 1 normalized time unit. Each figure contains four plots (two for total energy vs. time and two for enstrophy vs. time), for specified values of mean shear amplitude, $U_{ms}$ and Reynolds number, $Re$. The Ekman number was
held constant at $10^{-4}$ normalized units. The tests here use Lagrange cubic elements which are discussed in the next section.

Clearly, figures 5.2-5.4 show that increasingly restrictive tolerances converge to a solution for these relatively short duration runs. One would expect that for runs of longer duration tighter tolerances would be required to obtain consistent solutions.

**Degree of Freedom Sensitivity**

In most numerical methods the number of degrees of freedom (DOF) of the ODE system effects the stability and accuracy of the method employed. The FEM is no exception. There are two ways to increase the number of DOFs, one can simply increase the number of elements in the problem domain, or one can choose a more complex element type.

**Element Type**

Lagrange quadratic elements are the default element type in Femlab® for the two dimensional, general form, time dependent PDE application mode. It is also possible to use one of several higher order element types including Lagrange (cubic-quintic), Hermite (cubic-quintic), or Argyris quintic [7]. For the governing equations in this study where the viscous term contains a second order space derivative (i.e. $\nu \nabla^2 \omega$), expansion functions of order 3 or higher are used. It is possible to use integration by parts in the weak form to reduce the order of the derivative which would allow use of lower order elements, however this option was not explored in this study.

Figure 5.5 shows a depiction of the general two dimensional triangular element. Different elements types use different polynomials to approximate the value of a function over a triangular element. However the relationship between an elements local
Figure 5.2 $TE(t)$ and $EN(t)$ for increasingly restrictive tolerance settings. $U_{ms} = 0.0, R_e = 10^3, E_k = 10^{-4}$. 
Figure 5.3 $TE(t)$ and $EN(t)$ for increasingly restrictive tolerance settings. $U_{ms} = 1.0$, $Re = 10^3$, $E_k = 10^{-4}$. 
Figure 5.4 $TE(t)$ and $EN(t)$ for increasingly restrictive tolerance settings. $U_{rms} = 10.0$, $R_e = 10^3$, $E_k = 10^{-4}$. 
coordinates \((\xi, \eta)\) and the global coordinates \((x, y)\) can be uniquely defined using the four parameters, \(a, b, c, \theta\).

**Lagrange Elements**

Lagrange \(k^{th}\) order elements, for positive integers \(k \in [1, 2, 3, 4, 5]\), use piecewise polynomials of degree (at most) \(k\) in the local coordinates to describe a function \(\phi\) over an element. To describe such a function it suffices to give its values at the Lagrange points of order \(k\). These are the points whose local coordinates are integer multiples of \(1/k\) [7]. There are \(k + 1\) points along each element edge to interpolate the \(k^{th}\) order polynomial, with remaining nodes being interior or “bubble nodes” [8]. Figure 5.6 shows the Lagrange cubic element, \((k = 3)\). The degrees of freedom for this element are the values of the function at each point.

**Hermite Elements**

Hermite elements use the same points as Lagrange elements. The difference lies in the DOFs used [7]. For Hermite elements there is a DOF for the value of the function \(\phi\) at every point except those adjacent to a vertex. There are also DOFs for the value of the first derivative with respect to the global coordinates (i.e. \(\phi_x\) and \(\phi_y\)) at the vertices. These elements provide continuous first derivatives across elements only at the vertices. At other points shared by two elements the derivatives are not continuous. Figure 5.7 shows the Hermite cubic element, \((k = 3)\).

**Argyris Elements**

Argyris quintic elements use as DOFs the values of the function at the vertices, along with the first and second derivatives with respect to the global coordinates. That is on each vertex the DOFs are \(\phi, \phi_x, \phi_y, \phi_{xx}, \phi_{xy}, \phi_{yy}\). There are also
Figure 5.5 General triangular element.
Lagrange 3rd order Element

Figure 5.6  Lagrange cubic element, with 3 vertex nodes (circles), 6 line nodes (squares), and one interior node (triangle).
Hermite 3\textsuperscript{rd} order Element

Figure 5.7 Hermite cubic element with 6 vertex nodes (circles), 6 line nodes (squares), and one interior node (triangle).
Figure 5.8  Argyris quintic element with 18 vertex nodes (circles).

DOF's for the value of the normal derivative, $\phi_n$ at the midpoint of each triangle edge. In fact, along each edge the interpolant reduces to a one dimensional cubic and $\phi_n$ is known everywhere on the edge [8]. These elements ensure first order continuity between elements everywhere, and second order continuity at the element vertices.

Figure 5.8 shows the argyris quintic element.

**Experiments**

The experiments in this section are meant to illustrate the differences between using different element types for this model. Figure 5.9 shows the difference between the analytical value of the vorticity supplied as initial conditions, and the value of the
Figure 5.9 Normalized Laplacian error norm over increasing mesh resolution.

Laplacian of the stream function initial conditions, calculated by different element types.

Clearly, higher order elements approximate the second spatial derivative of the stream function with greater accuracy. This plot shows that cubic elements have an error of \( \approx 10\% \), quartic elements have an error of \( \approx 5\% \), and quintic elements have an error of \( \approx 1\% \).
Element Size

The resolution of the discretized domain has a major impact on the stability and accuracy of the approach. By defining a maximum element size we can look at the effect of mesh resolution on the problem at hand. A single set of initial conditions were allowed to evolve on meshes of increasing resolution. In these tests, the tolerance settings were held constant at $atol = 1e-2, rtol = 1e-1$ in order illustrate increasing accuracy over increasingly complex elements.

Application to Mean Shear

In this section the application of the FEM to the mean shear is discussed. Experiments are defined and results are presented.

In order to conclusively quantify the effectiveness of the FEM in modeling the effects of a linear mean shear on background turbulence, it is necessary to subject many different sets of initial conditions to the effect of mean shear for much longer times than the experiments here. Unfortunately due to time restrictions a task such as this is outside the scope of this paper. However, we can look at a few preliminary cases to help judge the validity of our implementation.

Experiments

For the experiments in this section, a single set of initial conditions (as defined for the previous sections), was allowed to evolve for 1 non-dimensionalized time unit. The mesh was consistently set to have the maximum element edge length $h_{max} = 1/32$. The value of $Re$ and $Ek$ are fixed at $10^3$ and $10^{-4}$ respectively. Both the external and
Figure 5.10 $TE(t)$ and $EN(t)$ over increasing mesh resolution. $U_{ms} = 0.0$, $Re = 10^3$. 
internal implementation were run for mean shear amplitude values of $U_{ms} = 0.0, 1.0,$ and 10.0.

**External**

Figures 5.11-5.12 show the time evolution of the streamfunction for external mean shear amplitudes of $U_{ms} = 0.0, 1.0,$ and 10.0. Along with $\Psi$, the energy as a function of Fourier space is plotted for each snapshot in time. Recall that in figure 5.1 all three situations begin with the same initial conditions.

In figure 5.11, after only a short time there is a redistribution of energy from the small spatial scales (higher wavenumber $k$) to larger spatial scales (lower wavenumber $k$). This is a well documented phenomena known as condensation, or the inverse cascade effect [11]. Clearly by figures 5.13 and 5.15 the mean shear has had a significant effect on the evolution of the streamfunction. For large mean shear amplitude there is a noticeable stretching of structures in the $x$ direction. These are a good indication that the model is acting appropriately based on the governing equations.

**Internal**

Recall from chapter 4, that in order to impose an internal mean shear the contributions are added to both the streamfunction and vorticity initial conditions and boundary conditions are adjusted where appropriate. Figures 5.17-5.22 mimic the set of figures for the external case. Notice that these figures look nearly identical to the corresponding figures for external mean shear. This is because the short duration for evolution has not yet allowed feedback effects to take place. Also, with such low values of $R_e$ energy is damped from small scales very quickly. Again there is condensation of energy from the smaller scales into larger scales.
Figure 5.11  External: $\psi(x,y)$ and $E_k(k)$. Time = 0.05 normalized units.
Figure 5.12 External: $\omega(x, y)$ and $\zeta(k)$. Time = 0.05 normalized units.
Figure 5.13  External: $\psi(x, y)$ and $E_k(k)$. Time = 0.20 normalized units.
Figure 5.14 External: $\omega(x,y)$ and $\zeta(k)$. Time = 0.20 normalized units.
Figure 5.15 External: $\psi(x, y)$ and $E_k(k)$. Time = 1.0 normalized units.
Figure 5.16  External: $\omega(x,y)$ and $\zeta(k)$. Time = 1.0 normalized units.
Figure 5.17 Internal: $\psi(x, y)$ and $E_k(k)$. Time = 0.05 normalized units.
Figure 5.18 Internal: $\omega(x,y)$ and $\zeta(k)$. Time = 0.05 normalized units.
For figures 5.19 and 5.21 there are only slight differences from the corresponding external case. The two majors reasons for not seeing more differences are that again these runs were performed with the values of $R_e$ and $E_k$ set such the system was not very stiff. In other words most of the small scale dynamics are damped out of the system due to the low value of $R_e$. Likewise the effect of the Coriolis forces are less than would be used in a production run.
Figure 5.19 Internal: $\psi(x, y)$ and $E_k(k)$. Time $= 0.20$ normalized units.
Figure 5.20  Internal: $\omega(x,y)$ and $\zeta(k)$. Time = 0.20 normalized units.
Figure 5.21 Internal: $\psi(x, y)$ and $E_k(k)$. Time = 1.0 normalized units.
Figure 5.22 Internal: $\omega(x, y)$ and $\zeta(k)$. Time = 1.0 normalized units.
CHAPTER 6  Discussion and Future Directions

Discussion

The governing equations in modeling geostrophic turbulence in two dimensions using the $\beta$-plane approximation and streamfunction-vorticity formulation have been introduced. The important differences between the PSM, (used in previous research) and the FEM (used here), have been outlined. Experiments showing the validity of the FEM formulation for low values of $Re$ and high values of $Ek$ have been conducted. It is yet to be shown that for a wide range of initial conditions the FEM is a suitable alternative to the PSM in general. However, this study has shown that the FEM can be used to model mean shear profiles which cannot be simulated using spectral approaches.

For low Reynolds number values (i.e $Re \leq 10^3$) the model works well. However this is most likely due to that fact that small scale dynamics are dominated by the viscous effects and damped out of the system. This damping causes reduced stiffness of the ODE system, and subsequently much less time is required by the ODE solver.

For higher Reynolds numbers the stiffness of the system becomes a major factor due to the lengthy integration times required. Another cause of lengthy run times is when a more resolved mesh is used, or when more complex elements are used. The added complexity in the numerical quadrature performed by Femlab\textsuperscript{\textregistered} requires
even more computational resources. For example, integrals of the Argyris element can be performed analytically in closed form to machine precision [8]. The fact that Femlab® uses 10-point quadrature to evaluate these integrals at each timestep, causes runs to be extremely slow relative to the time required for the PSM which does not require these evaluations.

**Future Directions**

Implementing techniques used by Jardin for the closed form calculation of integrals involved with the Argyris element could yield a much faster version of the FEM. Also, the phase speed error analysis techniques of Durran [4] would provide a platform upon which we could compare the stability of the PSM and FEM on an equal footing. Alternatively, incorporating the error analysis techniques proposed by Cullen and Morton [1], which use relaxation and prolongation operators, could provide a more focused method of error analysis.
BIBLIOGRAPHY


