Numerical Simulation of Stably Stratified Flow over Hills

Mr. Roderick Johnstone
Department of Mechanical Engineering
University of Surrey

Thesis submitted for the degree of Doctor of Philosophy
Monday 25th June 2001
The aim of the work described in this thesis is the development and application of a method to simulate computationally flows such as those investigated by Castro and Snyder [17], specifically flow over three-dimensional hills at high Reynolds and moderate to low obstacle Froude number. For hills elongated in the spanwise direction, this flow regime is characterized by breaking lee waves and accelerated flow near the lower surface downstream of the obstacle.

Simulations were performed by discretizing the three-dimensional Reynolds-averaged Navier-Stokes equations, and solving these numerically by a finite volume method. Buoyancy was modelled using the Boussinesq approximation, and modified k-ε models employed for turbulence closure.

The results obtained are found to be in reasonably good agreement with experimental flow visualizations. Critical Froude numbers for wave breaking are also found to be in reasonable agreement. Further, comparison is made with the nonlinear hydrostatic theory of Smith [101]; agreement is found to be fair, although the theory postulates a flow configuration differing from those observed in simulations.

Also investigated were the effects of modifications to the turbulence model, Reynolds number, small departures from linear stratification, of wall, symmetry, and wave-permeable boundary conditions, and of the size of the computational domain. The last of these was found to affect the transient development of the flow, but to have only a weak effect on the steady state converged to, pending the arrival of reflected internal waves. Grid independence of the solution was investigated, and found to be satisfactory. One subsequent grid dependence test, however, yielded more equivocal results.
Acknowledgments

I would like to thank my parents for their patience and encouragement. I would also like to acknowledge my supervisors, Professors Peter Voke and Ian Castro, and the financial support of the UK Meteorological Office and the NERC.
# Table of Contents

1 Nomenclature ........................................... 6

2 Introduction ........................................... 8

   2.1 Theoretical Background ............................ 8

      2.1.1 Waves of Large Amplitude - Long's Model ... 14

      2.1.2 A Hydrostatic Theory of Flow with Wave Breaking ... 19

   2.2 Relevance of Stratified Flow over Hills ............... 24

   2.3 Upstream Influence in Stably Stratified Flows .......... 31

   2.4 Experimental Data .................................. 34

   2.5 Literature Review .................................. 36

   2.6 Turbulence Modelling ................................ 44

      2.6.1 The $k$-$\varepsilon$ Model of Turbulence ........... 48

      2.6.2 Modelling the Influence of Stability on Turbulence ... 54

   2.7 Summary of Earlier Work with another Method .......... 59

3 Method .................................................. 65

   3.1 The Finite Volume Method .......................... 65

   3.2 Velocity Decomposition and Grid Arrangement .......... 69

   3.3 Calculation of Cell Volumes and Surface Vectors ....... 73

   3.4 Discretization of Momentum and Scalar Transport Equations ... 76
3.4.1 Time Discretization ........................................ 76
3.4.2 Discretization of Advection Terms ...................... 78
3.4.3 Discretization of Diffusion Term ......................... 81
3.4.4 Buoyancy .................................................. 83
3.4.5 Discretization of the Pressure Source Term .......... 84

3.5 Calculation of the Pressure .................................. 84
3.5.1 The SIMPLE Method ..................................... 85
3.5.2 Rhie-Chow Interpolation ................................. 88

3.6 Boundary Conditions ......................................... 90
3.6.1 Pressure Boundary Conditions ......................... 90
3.6.2 Inlet Boundary Conditions ............................... 90
3.6.3 Wall Boundary Conditions ............................... 90
3.6.4 Symmetry Boundary Conditions ....................... 91
3.6.5 Outflow Boundary Conditions ......................... 92
3.6.6 Advective Boundary Conditions ....................... 92

3.7 Turbulence Modelling ....................................... 94
3.7.1 Transport Equations .................................... 95
3.7.2 Source Terms ........................................... 95
3.7.3 Wall Boundary Condition .............................. 96
3.7.4 Modifications for Buoyant Flows ..................... 98

3.8 Implementation ............................................... 99
3.8.1 Convergence Criteria ........................................... 99
3.8.2 Calculation of Gradients ........................................ 100
3.8.3 Solution of Linear Equation System ....................... 101

3.9 Application to Stratified Flow over Hills ..................... 102
3.9.1 Hypotheses to be Tested ....................................... 102
3.9.2 Flow and Fluid Properties .................................... 104
3.9.3 Nondimensionalization ......................................... 104
3.9.4 Definition of Computational Domain ....................... 105
3.9.5 Inflow and Initial Conditions ................................. 110

4 Results ........................................................................ 112
4.1 Summary of some Validation Tests ............................. 112
4.2 COS1 Hill ................................................................. 120
4.3 COS2 Hill ................................................................. 124
4.4 COS3 Hill ................................................................. 130
4.5 Wave Drag and Comparison with Neutral Flow .......... 147
4.6 Critical Froude number as a function of Aspect Ratio .... 149
4.7 Comparison with the Theory of Smith ......................... 150
4.8 Influence of Turbulence Modelling ............................. 157
4.8.1 The Turbulent Schmidt Number .............................. 157
4.8.2 The $C_{13}$ Constant ............................................... 170
4.8.3 Limitation of the Turbulence Time Scale ................. 173
<table>
<thead>
<tr>
<th>4.9</th>
<th>Nonlinear Stratification .................................................. 177</th>
</tr>
</thead>
<tbody>
<tr>
<td>4.10</td>
<td>Influence of Domain Extent ............................................. 184</td>
</tr>
<tr>
<td>4.11</td>
<td>Effect of Mid-Channel Symmetry Condition .......................... 195</td>
</tr>
<tr>
<td>5</td>
<td>Discussion ........................................................................ 206</td>
</tr>
<tr>
<td>5.1</td>
<td>Grid and Time Step Dependence ........................................ 206</td>
</tr>
<tr>
<td>5.2</td>
<td>Effects of Wall Function Treatment and Near-Wall Grid .......... 227</td>
</tr>
<tr>
<td>5.3</td>
<td>Influence of Advective Boundary Condition ......................... 230</td>
</tr>
<tr>
<td>5.4</td>
<td>Reynolds-Number Sensitivity ............................................ 233</td>
</tr>
<tr>
<td>5.5</td>
<td>Sensitivity of Wave-breaking to Wall Function Formulation .... 235</td>
</tr>
<tr>
<td>6</td>
<td>Conclusions ....................................................................... 237</td>
</tr>
<tr>
<td>6.1</td>
<td>Findings relating to the Hypotheses made .......................... 237</td>
</tr>
<tr>
<td>6.2</td>
<td>Relevance to Atmospheric Flows ....................................... 240</td>
</tr>
<tr>
<td>6.3</td>
<td>Suggestions for further Work .......................................... 240</td>
</tr>
</tbody>
</table>
1. Nomenclature

\begin{align*}
C & \quad \text{A constant} \\
\mathcal{C} & \quad \text{Wave speed} \\
D & \quad \text{Number of spatial dimensions (1-3), or channel depth} \\
E_p & \quad \text{Potential energy} \\
E_k & \quad \text{Kinetic energy} \\
\mathbf{F} & \quad \text{Flux through a surface} \\
F_h & \quad \text{Froude number (U/Nh)} \\
F_p & \quad \text{Froude number (NU/g)} \\
G & \quad \text{A large number \(10^{30}\)} \\
G_k & \quad \text{Production of turbulence kinetic energy by buoyancy} \\
g & \quad \text{Gravitational acceleration} \\
h & \quad \text{Obstacle height} \\
i & \quad \text{Cartesian unit vector} \\
k & \quad \text{Wavenumber vector} \\
k & \quad \text{Turbulence kinetic energy} \\
l_m & \quad \text{Mixing length} \\
\mathbf{Me} & \quad \text{Velocity vector \(U,V,W\) at a cell face \(e\)} \\
N & \quad \text{Brunt-Väisälä frequency} \\
p & \quad \text{Pressure} \\
P_k & \quad \text{Production of turbulence kinetic energy by shear} \\
Q & \quad \text{Source term} \\
q & \quad \sqrt{U^2 + W^2} \\
r & \quad \text{Position vector (typically, one joining two mesh nodes)} \\
\text{Re} & \quad \text{Reynolds number} \\
\mathbf{S} & \quad \text{A surface vector} \\
\hat{\mathbf{S}} & \quad \text{Unit surface normal vector} \\
U & \quad \text{Velocity component (conventionally horizontal)} \\
u & \quad \text{Perturbation velocity component} \\
V & \quad \text{Velocity component (conventionally transverse)} \\
v & \quad \text{Perturbation velocity component} \\
W & \quad \text{Velocity component (conventionally vertical)} \\
w & \quad \text{Perturbation velocity component} \\
x & \quad \text{Horizontal coordinate} \\
y & \quad \text{Spanwise coordinate} \\
z & \quad \text{Vertical coordinate} \\
\alpha & \quad \text{Obstacle aspect ratio (width to length, in 3-D)} \\
\beta & \quad \text{Coefficient relating density with either salinity or temperature} \\
& \quad \text{(e.g. the volumetric expansion coefficient)}
\end{align*}
δ  Perturbation streamline height
ε  Dissipation rate of turbulence kinetic energy
η  Vorticity
φ  Scalar quantity
ν  Kinematic viscosity
ν_τ  Eddy viscosity
φ  Scalar quantity
ψ  Stream function
ρ  Density
σ  Schmidt or Prandtl number
θ  Density determining scalar (eg. salinity)
Ω  Cell volume
ω  Angular frequency

Superscripts
  τ  Turbulent fluctuation

Subscripts
  0  Reference or original value
t  Turbulent
2. Introduction

2.1 Theoretical Background

This thesis deals with the flow of nonhomogenous fluids over topography, specifically hills. These are taken to be obstacles with smoothly varying curvature, and small enough to permit the effects of the Earth’s rotation to be neglected\(^1\). In the atmosphere or ocean, density normally decreases with elevation, and varies little in the horizontal. In such layers, any fluid that is displaced vertically will usually tend to return to its original elevation; such stratification is therefore referred to as stable (unstable stratification is generally such that density increases with height).

A more precise criterion for stability may be obtained by considering a stratified fluid in hydrostatic equilibrium, such that

\[
\frac{dp}{dz} = -\rho g, \tag{2.1}
\]

where \(\rho\) is the density, \(g\) the acceleration due to gravity, and \(\frac{dp}{dz}\) the hydrostatic pressure gradient. Following Lighthill [56, p 288], we examine a parcel of fluid of density \(\rho_0\), displaced by a small distance \(\delta z\). At its new elevation the equilibrium density and pressure will be

\[
\rho_0 + \frac{dp}{dz} \delta z, \tag{2.2}
\]

and

\[
p_0 - \rho_0 g \delta z. \tag{2.3}
\]

We assume the parcel of fluid will assume the new pressure isentropically. Therefore its density will alter; if \(\delta z\) is positive, for instance, the pressure will drop, and so the fluid may expand, reducing its density. The new density of the displaced fluid will be given by

\[
\rho_0 - \frac{\rho_0 g \delta z}{c^2}, \tag{2.4}
\]

where \(c\) is the speed of sound, the square of which is the ratio of pressure to density changes at constant entropy.

The density of the displaced fluid will therefore differ from that of its surroundings by

\[
-\frac{\rho_0 g \delta z}{c^2} - \frac{dp}{dz} \delta z. \tag{2.5}
\]

\(^1\)The term Orography, by contrast, is usually reserved for obstacles not small compared to the Rossby radius of deformation (see section 2.3 below).
The fluid will experience a force equal to equation 2.5 multiplied by $g$, per unit volume. This force will oppose the motion provided that

$$\frac{-\rho_0 g}{c^2} > \frac{d\rho}{dz}.$$  

(2.6)

If this criterion is satisfied, then the stratification is stable. It will be assumed below that this is so.

The restoring force is equal to

$$-\left(\frac{g}{c^2} + \frac{1}{\rho_0} \frac{d\rho}{dz}\right) \rho_0 \delta z,$$  

(2.7)

which can be expressed thus

$$N^2 \rho_0 \delta z,$$  

(2.8)

where $N$ is

$$\sqrt{-\frac{g}{c^2} - \frac{1}{\rho_0} \frac{d\rho}{dz}}.$$  

(2.9)

$N$ is called the Brunt-Väisälä frequency, which is the highest frequency buoyancy-driven oscillations may attain (see equation 2.33 below). Typically this is of the order of $10^{-2}$ rad/s in both the atmosphere and the ocean \[8, p 4\].

The existence of a restoring force in the form of buoyancy permits oscillations, and therefore, in a continuous medium, waves. The motion will be in the vertical direction, while the wave will propagate horizontally. This, then, is a vertical transverse wave (see figure 2.3 at the end of this section). A layer of stratified fluid requires only an initial perturbation to set it in motion in this way, which can be supplied in a number of ways; an obvious (and indeed common) one is topography\(^\text{2}\). Fluid surmounting a hill or ridge will be raised above its original elevation and subsequently allowed to subside. This can lead to a variety of phenomena, depending in part on the shape of the hill, but in large measure determined by the Hill Froude number:

$$F_h = \frac{U}{Nh},$$  

(2.10)

where $U$ is the fluid velocity, and $h$ some representative vertical length scale (such as the height of the hill). Some workers (\[8, p 14\], for instance) prefer to use the inverse of the Froude number and call it $N_h u$. However, in this thesis, the usual practice will be followed, which is to restrict a dimensionless group to a single number (regardless of the length scale).

\(^\text{2}\)Other mechanisms include, for instance, buoyant motion, such as the intrusion into a stable layer either of a thermal (from below, \[112, p 37\]), or a gravity current (horizontally, \[59\])

9
The earliest theoretical work on lee waves is that of Lyra, Queney, and Scorer ([122, p 73][112, p 34][119], citing [63, 64, 91, 92, 99]) These workers considered an inviscid, stratified atmosphere, subject to small perturbations. Although compressibility and Coriolis forces were not generally neglected by them, their effects are only apparent at large scales, and will be ignored below.

If viscosity is neglected, the (Navier-Stokes) equations of motion reduce to the Euler equations:

\[
\frac{\partial \rho u}{\partial t} + \rho u \cdot \nabla u = -\nabla p + g \rho. \tag{2.11}
\]

Making the Boussinesq approximation, variations of density are retained only in the final (buoyancy) term. Elsewhere \( \rho \) is replaced by a reference density \( \rho_0 \). The following is then obtained

\[
\frac{\partial u}{\partial t} + u \cdot \nabla u = -\nabla \frac{p}{\rho_0} + g \frac{\rho}{\rho_0}. \tag{2.12}
\]

The transport equation for the density is

\[
\frac{\partial \rho}{\partial t} + u \cdot \nabla \rho - \frac{\rho_0 N^2}{g} \rho = 0. \tag{2.13}
\]

Neglecting compressibility, the continuity equation is

\[
\nabla \cdot u = 0. \tag{2.14}
\]

In order to obtain a set of linear equations, the advection term must be linearized. This can be done if we restrict ourselves to waves of small amplitude. The following will also be restricted to 2 dimensions (x and z, where z is vertical) for simplicity, although this is not necessary. If we consider the velocities \( u \) and \( w \) as small perturbations to some background velocity \( U \) then equations 2.12 and 2.13 may be written as:

\[
\frac{\partial u}{\partial t} + (U + u) \frac{\partial u}{\partial x} + w \frac{\partial U + u}{\partial z} = \frac{1}{\rho_0} \frac{\partial p}{\partial x}, \tag{2.15}
\]

\[
\frac{\partial w}{\partial t} + (U + u) \frac{\partial w}{\partial x} + w \frac{\partial U + w}{\partial z} = \frac{1}{\rho_0} \frac{\partial p}{\partial z} + g \frac{\rho}{\rho_0}, \tag{2.16}
\]

\[
\frac{\partial \rho}{\partial t} + (U + u) \frac{\partial \rho}{\partial x} + w \frac{\partial \rho}{\partial z} = \frac{\rho_0 N^2}{g} \rho. \tag{2.17}
\]

Neglecting products of \( u, w, \) and \( \rho \) or their derivatives with each other, the equations 2.15, 2.16, and 2.17 can be further simplified to:

\[
\frac{\partial u}{\partial t} + U \frac{\partial u}{\partial x} + w \frac{\partial U}{\partial z} = \frac{1}{\rho_0} \frac{\partial p}{\partial x}, \tag{2.18}
\]

\[
\frac{\partial w}{\partial t} + U \frac{\partial w}{\partial x} = \frac{1}{\rho_0} \frac{\partial p}{\partial z} + g \frac{\rho}{\rho_0}, \tag{2.19}
\]

\[
\frac{\partial \rho}{\partial t} + U \frac{\partial \rho}{\partial x} = \frac{\rho_0 N^2}{g} \rho. \tag{2.20}
\]
Following Turner [112, p 24], we consider the case $U = 0$

$$\rho_0 \frac{\partial u}{\partial t} = -\frac{1}{\rho_0} \frac{\partial p}{\partial x},$$

$$\rho_0 \frac{\partial w}{\partial t} = -\frac{1}{\rho_0} \frac{\partial p}{\partial z} - \rho g,$$

$$\frac{\partial p}{\partial t} = \rho_0 N^2 u,$$

$$\frac{\partial u}{\partial x} = -\frac{\partial w}{\partial z}.$$  

(2.21)  

(2.22)  

(2.23)  

(2.24)

Removing the pressure gradient terms by cross-differentiation and subtraction of the momentum equations leads to

$$\rho_0 \frac{\partial^2 u}{\partial t \partial z} = -\frac{1}{\rho_0} \frac{\partial^2 p}{\partial x \partial z},$$

$$\rho_0 \frac{\partial^2 w}{\partial t \partial x} = -\frac{1}{\rho_0} \frac{\partial^2 p}{\partial z \partial x} - \rho g \frac{\partial^2 p}{\partial x^2},$$

and

$$\rho_0 \frac{\partial}{\partial t} \left( \frac{\partial u}{\partial z} - \frac{\partial w}{\partial x} \right) = \rho g \frac{\partial p}{\partial x}.$$  

(2.25)  

(2.26)  

(2.27)

The term in $\rho$ can be removed by differentiating equation 2.23 with respect to $x$ and equation 2.27 with respect to $t$, giving

$$\rho_0 N^2 \frac{\partial w}{\partial x} = \rho_0 \frac{\partial^2 \rho}{\partial x \partial t} = \rho_0 \frac{\partial}{\partial t} \left( \frac{\partial u}{\partial z} - \frac{\partial w}{\partial x} \right).$$

(2.28)

We can obtain an equation in $w$ only (or, of course, in any other individual variable), by differentiating equation 2.28 with respect to $x$ and the continuity equation 2.24 with respect to $z$. We obtain

$$\rho_0 N^2 \frac{\partial^2 w}{\partial x^2} = \rho_0 \frac{\partial}{\partial t} \left( \frac{\partial^2 u}{\partial x \partial z} - \frac{\partial^2 w}{\partial x^2} \right),$$

$$\frac{\partial^2 w}{\partial x \partial z} = -\frac{\partial^2 w}{\partial z^2},$$

which yield

$$N^2 \frac{\partial^3 w}{\partial x^3} = \frac{\partial}{\partial t} \left( \frac{\partial^2 w}{\partial z^2} + \frac{\partial^2 w}{\partial x^2} \right).$$

(2.29)  

(2.30)  

(2.31)

When $N$ is constant, equation 2.31 has a solution of the form

$$w = w_1(z) e^{i(\omega t - kx - ly - mz)},$$

where $w_1(z)$ is the wave amplitude, and the frequency $\omega$ is given by

$$\omega^2 = \frac{N^2(k^2 + l^2)}{(k^2 + l^2 + m^2)}.$$  

(2.32)  

(2.33)
The constants $k, l, m$ are the $x, y, z$ components of the wavenumber vector. Note that $\omega$ can never exceed $N$. If $N$ varies, the propagation of waves can be affected by this restriction; waves can then be trapped in a region where $\omega < N$.

Substituting 2.32 into 2.31 yields
\[
\frac{\partial^2 w_1}{\partial z^2} = \left( 1 - \frac{N^2}{\omega^2} \right) k^2 w_1.
\]

Waves for which $k$ is zero always satisfy this equation; these are called columnar modes, and will be referred to again below. We are, however, most interested in stationary waves downstream of obstacles, generally referred to as lee waves. In an infinite and stably stratified atmosphere, there will always be waves such that their phase velocity $c$ (equal to $\omega/k$) is equal to the fluid velocity $U$. In a finite domain this may not be the case; the amplitude must then be zero at both boundaries, which restricts the possible values of $k$. In order to obtain a unique solution in an infinite domain, however, it is necessary to impose a radiation boundary condition, so as to permit the propagation of energy in an upward direction only.

Further insight can be obtained by considering the energy equation, which can be formed [30] by multiplying equation 2.18 by $u$, 2.19 by $w$, and 2.20 by $\rho$. The result is
\[
\frac{1}{2} \frac{\partial u^2}{\partial t} + \frac{\partial}{\partial x} \left( \frac{U}{2} \frac{\partial u^2}{\partial x} + uw \frac{\partial U}{\partial z} \right) = -\frac{u}{\rho_0} \frac{\partial p}{\partial x},
\]
\[
\frac{1}{2} \frac{\partial w^2}{\partial t} + \frac{\partial}{\partial x} \left( \frac{U}{2} \frac{\partial w^2}{\partial x} + uw \frac{\partial U}{\partial z} \right) = -\frac{w}{\rho_0} \frac{\partial p}{\partial z} + g U \frac{\partial \rho}{\rho_0},
\]
\[
\frac{1}{2} \frac{\partial \rho^2}{\partial t} + \frac{\partial}{\partial x} \left( \frac{U}{2} \frac{\partial \rho^2}{\partial x} \right) = \frac{\rho \rho_0 N^2}{g} w.
\]

Again ignoring products of perturbations or their derivatives, and summing, we obtain
\[
\frac{1}{2} \frac{\partial}{\partial t} (u^2 + w^2 + (\rho_0 N \rho_0)^2) + U \frac{\partial}{\partial x} (u^2 + w^2 + (\rho_0 N \rho_0)^2) + uw \frac{\partial U}{\partial z} = -\frac{u}{\rho_0} \frac{\partial p}{\partial x} - \frac{w}{\rho_0} \frac{\partial p}{\partial z}.
\]

If we identify $(\rho_0/2)(u^2 + w^2)$ as the kinetic energy $E_k$ of the wave perturbation, and $(\rho_0/2)(\rho_0 N \rho_0)^2$ as the potential energy $E_p$, equation 2.38 can be written as
\[
\frac{\partial (E_k + E_p)}{\partial t} + \frac{\partial (U(E_k + E_p) + pu)}{\partial x} + \frac{\partial pw}{\partial z} = -\rho_0 uw \frac{\partial U}{\partial z}.
\]

If we restrict the analysis to steady flows by ignoring derivatives in time, and assume $u, w$ and $\rho$ all vanish at $x = \pm \infty$, horizontal averaging yields the following
\[
\frac{\partial pw}{\partial z} = -\rho_0 \frac{\partial U}{\partial z}.
\]
overbars denoting averaging. If, on the other hand, equation 2.18 is multiplied by \((\rho_0 U + p)\), the following is obtained (again neglecting the unsteady term)

\[
\left( \rho_0 U \frac{\partial u}{\partial x} + \frac{\partial p}{\partial x} \right) (\rho_0 u U + p) + \rho_0 u w U \frac{\partial U}{\partial z} = -\rho_0 p w \frac{\partial U}{\partial z}.
\] (2.41)

Averaging horizontally again, we obtain

\[
\rho_0 U \overline{w} \frac{\partial U}{\partial z} = -\rho_0 p w \frac{\partial U}{\partial z}.
\] (2.42)

We have made no assumption about the value of \(\partial U/\partial z\), which may therefore be different from zero. Hence we obtain

\[
\rho_0 U \overline{w} = -\overline{p w}.
\] (2.43)

Equation 2.43 also implies that the vertical flux of momentum is opposite in sign to that of wave energy \(\overline{p w}\). Hence, if wave energy is transported upwards from the topography (which we should expect to be the usual situation, since the topography is impermeable to it), momentum must be transported downwards. The vertical flux of wave energy is also proportional to \(U\). On this basis, we would expect the flux to be reversed if the background velocity did the same.

This is one condition giving rise to a so-called critical level, through which wave propagation is impossible. Other possible causes for this are disappearance or reversal of the density stratification (since waves can only propagate through stably stratified fluid), and self-induced flow reversal. The latter can be brought about when an internal wave overturns; then, a streamline becomes locally vertical, so that the local flow velocity may become zero or negative, even though the background \(U\) may be positive. Such nonlinear effects cannot be adequately described using linear theories such as this, but will be treated below using a different approach.

However, for waves of small amplitude, and \(U \neq 0\), we find, by comparing 2.40 and 2.43, that

\[
\rho_0 U \overline{w} = \text{constant}.
\] (2.44)

This is the Eliassen-Palm theorem [30], which implies that the momentum flux is constant with height at constant \(U\). In the absence of viscous forces, wave breaking, or critical levels (where \(U\) changes sign, for instance), therefore, wave energy is transported upwards indefinitely.

We shall be most concerned, however, with lee waves that overturn and break. The theorem is then violated, as wave energy may be lost to turbulence or even reflected downwards, back towards the surface. As already discussed, theories such as the ones above cease to be useful at that point. In order to investigate waves that may break, and in fact waves of large amplitude in general, it is necessary to avoid linearizing the advection term.
2.1.1 Waves of Large Amplitude - Long’s Model

Following Long [60, 61, 68] and Yih [122, p 103] \(^3\), the (steady-state) Euler equation 2.11 may be rewritten by introducing the streamfunction and vorticity \(\psi\) and \(\eta\). This necessitates restriction to two dimensions, such that:

\[
\rho U \frac{\partial U}{\partial x} + \rho W \frac{\partial U}{\partial z} = -\frac{\partial p}{\partial x},
\]
(2.45)

\[
\rho U \frac{\partial W}{\partial x} + \rho W \frac{\partial W}{\partial z} = -\frac{\partial p}{\partial z} + g\rho.
\]
(2.46)

Note that \(U\) and \(W\) in the above are not perturbation quantities, and also that the unsteady terms have been dropped.

The streamfunction is defined in the usual way

\[
\frac{\partial \psi}{\partial z} = -U,
\]
(2.48)

\[
\frac{\partial \psi}{\partial x} = W.
\]
(2.49)

and the vorticity by

\[
\eta = \frac{\partial W}{\partial z} - \frac{\partial U}{\partial x}.
\]
(2.50)

We note that the advection terms (the left-hand sides of equations 2.45 and 2.46) can be written as follows by introducing \(\psi\) and \(\eta\)

\[
\rho \frac{\partial (U^2 + W^2)/2}{\partial x} - \rho \frac{\partial \psi}{\partial x} = \rho U \frac{\partial U}{\partial x} + \rho W \frac{\partial U}{\partial z},
\]
(2.51)

\[
\rho \frac{\partial (U^2 + W^2)/2}{\partial z} - \rho \frac{\partial \psi}{\partial z} = \rho U \frac{\partial W}{\partial x} + \rho W \frac{\partial W}{\partial z}.
\]
(2.52)

When these relations are substituted into equations 2.45 and 2.46 the following are obtained

\[
\rho \frac{\partial}{\partial x} \left( \frac{U^2 + W^2}{2} \right) = \rho \eta \frac{\partial \psi}{\partial x} - \frac{\partial p}{\partial x},
\]
(2.53)

\[
\rho \frac{\partial}{\partial z} \left( \frac{U^2 + W^2}{2} \right) = \rho \eta \frac{\partial \psi}{\partial z} - \frac{\partial p}{\partial z} + g\rho.
\]
(2.54)

This can be simplified somewhat by introducing

\[
q = \sqrt{U^2 + W^2},
\]
(2.55)

\(^3\)The alternative derivation of Yih [121] is more general, but will not be given here.
so that

\[
\rho \frac{\partial q^2/2}{\partial x} = \rho \frac{\partial \psi}{\partial x} - \frac{\partial \rho}{\partial x}, \tag{2.56}
\]
\[
\rho \frac{\partial q^2/2}{\partial z} = \rho \frac{\partial \psi}{\partial z} - \frac{\partial \rho}{\partial z} + g \rho. \tag{2.57}
\]

The pressure can be removed by cross-differentiation, as with 2.27 above. The result is

\[
\frac{\partial \rho}{\partial z} \frac{\partial q^2/2}{\partial x} - \frac{\partial \rho}{\partial x} \frac{\partial q^2/2}{\partial z} = \frac{\partial \eta}{\partial z} \frac{\partial q}{\partial x} - \frac{\partial \eta}{\partial x} \frac{\partial q}{\partial z} + \eta \frac{\partial \rho}{\partial z} \frac{\partial \psi}{\partial x} - \eta \frac{\partial \rho}{\partial x} \frac{\partial \psi}{\partial z} + g \rho \frac{\partial \psi}{\partial x}. \tag{2.58}
\]

Substituting 2.48 and 2.49 for derivatives of \( \psi \), we find that

\[
\frac{\partial \rho}{\partial z} \frac{\partial q^2/2}{\partial x} - \frac{\partial \rho}{\partial x} \frac{\partial q^2/2}{\partial z} = W \rho \frac{\partial \eta}{\partial x} + U \rho \frac{\partial \eta}{\partial x} + \eta \frac{\partial \rho}{\partial x} + \eta \frac{\partial \rho}{\partial x} + g \frac{\partial \rho}{\partial x}. \tag{2.59}
\]

Identifying the time derivative as

\[
\frac{d}{dt} = U \frac{\partial \rho}{\partial x} + W \frac{\partial \rho}{\partial z}, \tag{2.60}
\]

we obtain

\[
\frac{\partial \rho}{\partial z} \frac{\partial q^2/2}{\partial x} - \frac{\partial \rho}{\partial x} \frac{\partial q^2/2}{\partial z} = \frac{\partial \rho}{\partial t} + \rho \frac{\partial \eta}{\partial t} + g \frac{\partial \rho}{\partial x}, \tag{2.61}
\]

which can be written as

\[
\frac{\partial \rho}{\partial z} \frac{\partial q^2/2}{\partial x} - \frac{\partial \rho}{\partial x} \frac{\partial q^2/2}{\partial z} = \frac{\partial \rho}{\partial t} (\rho \eta) + g \frac{\partial \rho}{\partial x}. \tag{2.62}
\]

The derivatives of \( \rho \) in the above can be expressed with respect to the stream function (defined by 2.48, 2.49) as follows

\[
\frac{\partial \rho}{\partial x} = \frac{\partial \rho}{\partial \psi} \frac{\partial \psi}{\partial x} = W \frac{\partial \rho}{\partial \psi}, \tag{2.63}
\]
\[
\frac{\partial \rho}{\partial z} = \frac{\partial \rho}{\partial \psi} \frac{\partial \psi}{\partial z} = -U \frac{\partial \rho}{\partial \psi}. \tag{2.64}
\]

Then

\[
- \frac{\partial \rho}{\partial \psi} \left( U \frac{\partial q^2/2}{\partial x} + W \frac{\partial q^2/2}{\partial z} \right) = \frac{\partial \rho \eta}{\partial t} + g W \frac{\partial \rho}{\partial \psi}, \tag{2.65}
\]

and

\[
\frac{\partial \rho \eta}{\partial t} + g \frac{\partial \rho}{\partial x} + \frac{\partial \rho}{\partial \psi} \left( \frac{\partial q^2/2}{\partial t} + g \frac{\partial \eta}{\partial t} \right) = 0. \tag{2.66}
\]

Note that \( W \) becomes \( dz/dt \); the variable \( z \) is to be interpreted as the vertical displacement of a streamline.

Since the medium is assumed incompressible, and diffusion is neglected, \( \rho \) may be supposed to be constant on a streamline (advection being possible only along the streamline). Therefore \( \frac{\partial \rho}{\partial \psi} \) is not a function of time. Hence, we integrate equation 2.66 with respect to time, obtaining

\[
\eta + \frac{1}{\rho} \frac{\partial \rho}{\partial \psi} \left( \frac{q^2}{2} + g z \right) = H(\psi), \tag{2.67}
\]
the constant of integration (with respect to $t$) $H$ being a function of $\psi$. Using the definitions of $\psi$, 2.48 and 2.49 and that of $\eta$, 2.50, we can rewrite the above as

$$\nabla^2 \psi + \frac{1}{\rho} \frac{d}{d\psi} \left( \frac{(\nabla \psi)^2}{2} + gz \right) = H(\psi). \quad (2.68)$$

Long [60] makes the assumption that $H$ may be determined by the conditions far upstream of the obstacle, which are assumed to be unaffected by its presence. The right hand side of the above is then equal to the left hand side evaluated sufficiently far upstream, where, it is assumed, $W$ is zero.

$$\nabla^2 \psi + \frac{1}{\rho} \frac{d}{d\psi} \left( \frac{(\nabla \psi)^2}{2} \right) = \nabla^2 \psi_0 + \frac{1}{\rho} \frac{d}{d\psi} \left( \frac{U_0^2}{2} + g(z_0 - z) \right), \quad (2.69)$$

where $\psi_0$ and $U_0$ are the stream function and the velocity far upstream, related by

$$U_0 = \nabla \psi_0. \quad (2.70)$$

The velocities $U$ and $W$ can be rewritten in terms of $U_0$ as follows

$$U = -\frac{\partial \psi}{\partial z} = -\frac{\partial \psi}{\partial z_0} \frac{\partial z_0}{\partial z} = U_0 \frac{\partial z_0}{\partial z}, \quad (2.71)$$

$$W = \frac{\partial \psi}{\partial z} = \frac{\partial \psi}{\partial z_0} \frac{\partial z_0}{\partial z} = -U_0 \frac{\partial z_0}{\partial z}. \quad (2.72)$$

$W$ can be neglected far upstream, so the vorticity there can be written as

$$\nabla^2 \psi_0 = \frac{\partial}{\partial z_0} \left( U_0 \frac{\partial z_0}{\partial z_0} \right) = \frac{\partial U_0}{\partial z_0}, \quad (2.73)$$

since $z = z_0$. The downstream vorticity $\nabla^2 \psi$ assumes the slightly more complicated form

$$\nabla^2 \psi = \frac{\partial W}{\partial z} - \frac{\partial U}{\partial z} = -\frac{\partial U_0}{\partial z_0} \frac{\partial z_0}{\partial z} - U_0 \frac{\partial^2 z_0}{\partial z^2} - \frac{\partial U_0}{\partial z} \frac{\partial z_0}{\partial z} - \frac{\partial U_0}{\partial z_0} \frac{\partial^2 z_0}{\partial z^2}. \quad (2.74)$$

$z_0$ is not a function of $x$, so

$$\nabla^2 \psi = -U_0 \nabla^2 z_0 - \frac{\partial U_0}{\partial z} \frac{\partial z_0}{\partial z}. \quad (2.75)$$

The second term on the RHS may be rewritten as

$$\frac{\partial U_0}{\partial z} \frac{\partial z_0}{\partial z} = \frac{\partial U_0}{\partial z_0} \frac{\partial z_0}{\partial z} \frac{\partial z_0}{\partial z}. \quad (2.76)$$

Subtracting the upstream vorticity 2.73 from 2.76, we obtain

$$\nabla^2 \psi - \nabla^2 \psi_0 = -U_0 \nabla^2 z_0 - \frac{\partial U_0}{\partial z_0} \frac{\partial z_0}{\partial z} \frac{\partial z_0}{\partial z} + \frac{\partial U_0}{\partial z_0}. \quad (2.77)$$
\begin{equation}
\nabla^2 \psi - \nabla^2 \psi_0 = -U_0 \nabla^2 z_0 - \frac{\partial U_0}{\partial z_0} (1 - \nabla^2 z_0) .
\end{equation}

Substituting equation 2.78 into equation 2.69, we find that

\begin{equation}
-\frac{\partial U_0}{\partial z_0} (1 - (\nabla z_0)^2) + \frac{1}{\rho} \frac{d \rho}{d \psi} \frac{(\nabla \psi)^2}{2} = -U_0 \nabla^2 z_0 + \frac{1}{\rho} \frac{d \rho}{d \psi} \left( \frac{U_0^2}{2} + g(z_0 - z) \right) .
\end{equation}

The stream function \( \psi \) can be eliminated, as it is a known function of \( z_0 \)

\begin{equation}
d\psi = -U_0 dz_0 .
\end{equation}

Using equation 2.80, we can rewrite the terms in \( d\rho/d\psi \) on both sides of the equation as

\begin{equation}
\frac{1}{\rho} \frac{d \rho}{d \psi} \frac{(\nabla \psi)^2}{2} = -\frac{1}{U_0} \frac{d (U_0 \nabla z_0)^2}{d z_0} ,
\end{equation}

\begin{equation}
\frac{1}{\rho} \frac{d \rho}{d \psi} \left( \frac{U_0^2}{2} + g(z_0 - z) \right) = -\frac{1}{U_0} \frac{d \rho}{d z_0} \left( \frac{U_0^2}{2} + g(z_0 - z) \right) ,
\end{equation}

for the LHS and RHS terms respectively. Equation 2.79 now becomes

\begin{equation}
-\nabla^2 z_0 + \frac{1}{U_0} \frac{\partial U_0}{\partial z_0} (1 - (\nabla z_0)^2) - \frac{1}{\rho} \frac{d \rho}{d z_0} \frac{U_0^2 (\nabla z_0)^2}{2} = -\frac{1}{U_0} \frac{d \rho}{d z_0} \left( \frac{U_0^2}{2} + g(z_0 - z) \right) .
\end{equation}

With some rearrangement, we find that

\begin{equation}
-\nabla^2 z_0 + (1 - (\nabla z_0)^2) \left( \frac{1}{U_0} \frac{\partial U_0}{\partial z_0} + \frac{1}{2 \rho} \frac{d \rho}{d z_0} \right) = -\frac{1}{U_0} \frac{d \rho}{d z_0} \left( g(z_0 - z) \right) .
\end{equation}

We can simplify the second term on the LHS in the following way:

\begin{equation}
\frac{d (\ln U_0^2 \rho)}{dz_0} = \frac{1}{\rho} \frac{d \rho}{d z_0} + \frac{2}{U_0} \frac{d U_0}{dz_0} ,
\end{equation}

so

\begin{equation}
(1 - (\nabla z_0)^2) \left( \frac{1}{U_0} \frac{d U_0}{dz_0} + \frac{1}{2 \rho} \frac{d \rho}{d z_0} \right) = \frac{1}{2} (1 - (\nabla z_0)^2) \frac{d (\ln U_0^2 \rho)}{dz_0} .
\end{equation}

Equation 2.84 now becomes

\begin{equation}
\nabla^2 z_0 + \frac{1}{2} ( (\nabla z_0)^2 - 1) \frac{d (\ln U_0^2 \rho)}{dz_0} = \frac{1}{U_0} \frac{d \rho}{d z_0} \left( g(z_0 - z) \right) .
\end{equation}
This is Long's [60] equation 12, which governs steady, stratified flow in two dimensions.

Equation 2.87 can be further simplified by substituting $\delta$ for $z_0 - z$, so that

$$\nabla \delta = \nabla z_0 - \nabla z = \frac{\partial z_0}{\partial z} - 1 = \frac{\partial \delta}{\partial z},$$

(2.88)

and

$$\nabla^2 \delta = \nabla^2 z_0.$$  

(2.89)

The following is then obtained

$$\nabla^2 \delta + \frac{1}{2} \frac{d(\ln U_0^2 \rho)}{dz_0} \left( (\nabla \delta)^2 + 2 \frac{\partial \delta}{\partial z} \right) = \frac{g}{U_0^2 \rho} \frac{d \rho}{dz_0} \delta.$$  

(2.90)

This equation becomes linear when $U_0^2 \rho$ happens to be constant, and the stratification is linear ($\frac{d \rho}{dz_0} = \text{constant}$). In that case we find

$$\nabla^2 \delta + \left( \frac{g}{U_0^2} \frac{d \rho}{dz_0} \right) \frac{\delta}{U_0^2} = 0,$$  

(2.91)

which can be rewritten in terms of the Brunt-Väisälä frequency $N$

$$\nabla^2 \delta + \frac{N^2}{U_0^2} \delta = 0.$$  

(2.92)

which is of the form of a Helmholtz equation; it, and the assumptions made in its derivation (notably that of no upstream influence), are known collectively as Long's model. Long used equation 2.92 to calculate flows over topography in channels of finite depth by an inverse method (that is, the form of the topography required to produce a flow was a result of the analysis). Comparisons with corresponding experiments were encouraging.

Miles and Huppert [68, 69, 70, 71] succeeded in applying Long's equation to the problem of lee waves in an unbounded two-dimensional domain. They obtained solutions for a variety of shapes and upstream conditions, determining the lee-wave field and the obstacle wave drag. They were also able to predict the appearance of an overturning streamline as a function of Froude number and obstacle aspect ratio. At this point, the flow becomes statically unstable.

In physical flows, turbulence generally results in such cases. A well mixed, stagnant region develops in place of the reversed streamline. Sometimes this region is very deep, and the flow beneath becomes constricted and accelerates. Downstream of the stagnant region, the flow rebounds in a manner resembling a hydraulic jump. Very high drag is exerted on the obstacle, since the flow does not decelerate downstream of the top of the obstacle, but may instead accelerate further. The pressure on the lee side of the obstacle does not then recover to any degree (as is usually the case), but continues to fall. This phenomenon is responsible for downslope windstorms in the atmosphere.
2.1.2 A Hydrostatic Theory of Flow with Wave Breaking

Smith [101] applied Long's model to an idealized severe downslope windstorm configuration, shown below in figure 2.1. It is assumed a dividing streamline exists, at an altitude $H_0$ upstream of the stagnation point. In the lee of the hill the upper and lower branches define a largely stagnant and hydrostatic region, in which the fluid is well mixed by turbulence. The density there, $\rho_C$, is then assumed roughly constant. The lower boundary of this region is at an altitude $H_1$. As before, we denote by $\delta$ the vertical deflection of a streamline from its upstream value.

Figure 2.1: High Drag Configuration, adapted from Smith [101]

The horizontal velocity $U$ can be obtained from:

$$U = U_0(1 - \frac{\partial \delta}{\partial z}).$$  \hspace{1cm} (2.93)

The boundary condition on the surface is simply that

$$\delta = h(x),$$  \hspace{1cm} (2.94)

where $h(x)$ is the height of the terrain above the reference level.

On the upper branch of the dividing streamline, at $H_0$, it is assumed that the pressure is constant, which is believed to be a good approximation provided the disturbance there is small.

$$p(x, H_0) = p^*.$$  \hspace{1cm} (2.95)

Since the fluid in the mixed region is assumed to be hydrostatic and at constant density, the pressure on the lower branch of the streamline is given by

$$p(x, H_0 - \delta_C) = p^* + \rho_C g \delta_C.$$  \hspace{1cm} (2.96)

\footnote{Smith's version differs in the sign of $\rho_C g \delta_C$; this is because he defines $g$ positive pointing downwards. All directional quantities above are defined positive pointing upwards.}
If the velocity at $H_0$ is $U_0$, we find, by application of Bernoulli’s equation,
\[
p + \frac{1}{2} \rho U^2 - \rho g z = \text{constant},
\]
that the velocity at $H_0 + \delta_C$ is also $U_0$.
\[
U(x, H_0 + \delta_C) = U_0.
\]
(2.98)

This means that, by equation 2.93
\[
\frac{\partial \delta}{\partial z} = 0,
\]
(2.99)
on the lower branch of the streamline, although $\delta$ is undefined for all $x$ where $\delta_C(x) \neq 0$, and $z$ such that $H_0 < z < H_0 + \delta_C$.

Equations 2.94 and 2.99 provide boundary conditions for equation 2.92. The latter is satisfied by
\[
\delta(x, z) = A(x) \cos \frac{N}{U_0} z + B(x) \sin \frac{N}{U_0} z,
\]
(2.100)
as
\[
\frac{\partial^2 \delta}{\partial z^2} = -A \frac{N^2}{U_0^2} \cos \frac{N}{U_0} z - B \frac{N^2}{U_0^2} \sin \frac{N}{U_0} z,
\]
(2.101)
which corresponds to equation 2.92. Substituting equations 2.94 and 2.99 in turn, we obtain
\[
h = A \cos \left( \frac{N}{U_0} h \right) + B \sin \left( \frac{N}{U_0} h \right),
\]
(2.102)
\[
0 = A \frac{N}{U_0} \sin \left( \frac{N}{U_0} (H_0 + \delta_C) \right) - B \frac{N}{U_0} \cos \left( \frac{N}{U_0} (H_0 + \delta_C) \right).
\]
(2.103)

Further, equation 2.100 should equal $\delta_C$ when $z = H_0 + \delta_C$,
\[
\delta_C = A \cos \left( \frac{N}{U_0} (H_0 + \delta_C) \right) + B \sin \left( \frac{N}{U_0} (H_0 + \delta_C) \right).
\]
(2.104)

Defining
\[
\hat{h} = h \frac{N}{U_0},
\]
(2.105)
\[
\hat{H}_0 = H_0 \frac{N}{U_0},
\]
(2.106)
\[
\hat{\delta}_C = \delta_C \frac{N}{U_0},
\]
(2.107)
\[
\hat{A} = A \frac{N}{U_0},
\]
(2.108)
\[
\hat{B} = B \frac{N}{U_0},
\]
(2.109)
we can rearrange equations 2.102, 2.103 and 2.104 to obtain

\[ h = \delta_C \cos(H_0 + \delta_C - \hat{h}), \quad (2.110) \]
\[ A = \delta_C \cos(H_0 + \delta_C), \quad (2.111) \]
\[ B = \delta_C \sin(H_0 + \delta_C). \quad (2.112) \]

Smith solves the transcendental equation 2.110 for \( \delta_C \), then finding \( A \) and \( B \) from 2.111 and 2.112. Possible solutions are shown in figure 2.2 for a range of values of \( H_0 \). Solutions in the lower right-hand quadrant correspond to flow that accelerates across a ridge. How this corresponds to the postulated physical flow shown in figure 2.2 can be understood by varying \( x \), and considering the effect on \( h \) and \( \delta_C \). Initially, both \( h \) and \( \delta_C \) are zero. As we reach the foot of the hill, \( h \) rises. As it does so, the solution for \( \delta_C \) in the lower quadrant decreases, for sufficiently high \( \hat{H}_0 \). This corresponds to the beginning of the stagnant region shown in figure 2.1 above, as the lower branch of the dividing streamline descends. Eventually, a minimum is reached. As the hill height then decreases in height, we see that for sufficiently high \( \hat{H}_0 \), \( \delta_C \) continues to fall. This corresponds to a situation in which transition to an accelerated flow occurs. Those lines that continue into the lower left-hand quadrant correspond to flows over asymmetric hills, so that \( h \) continues to fall to below zero.

Figure 2.2:
Solutions to equation 2.110.
Lines are at constant \( \hat{H}_0 \), where \( \hat{H}_0 = n \frac{\pi}{6} \)

Smith [102] shows that in order for a solution to enter the lower right-hand quadrant, the value of \( \hat{H}_0 \) must be at least

\[ \hat{H}_0 = \hat{h}_M - \hat{\delta} + \arccos(\hat{h}_M/\hat{\delta}) + 2\pi n, \quad n = 0, 1, 2, 3, ... \quad (2.113) \]
where $\hat{h}_M$ is the maximum hill height, and $\hat{\delta}$ is given by
\[
\hat{\delta} = -\frac{1}{\sqrt{2}} \left[ \hat{h}_M^2 + \hat{h}_M (\hat{h}_M^2 + 4)^{1/2} \right]^{1/2}. \tag{2.114}
\]

A limitation of Smith's theory is that no solutions exist in the high drag regime for $\hat{h} > 1$ (corresponding to $F_h < 1$). It is unclear why this is so - such flows very often do exhibit high drag states [17]. Smith suggests that upstream blocking occurs under such conditions. This lowers the effective height of the hill upstream, since only fluid above the dividing streamline will flow over it. Interpreted in this way, the theory predicts $d = h - U_0/N$; here $d$ is the height of the dividing streamline, or, in nondimensional form
\[
d = \frac{d N}{U_0} = \hat{h} - 1. \tag{2.115}
\]
provided, of course, that the RHS is positive.

Redefining $\hat{h}$ as $\hat{h} = (h - d)N/U_0$ would then permit the theory to remain valid. The height of the hill with respect to the surface downstream, however, would not be affected. Hence one way of testing this hypothesis would be to compare Smith's theory with observations for $F_h < 1$, approximating the hill as an asymmetric obstacle. The height relative to the upstream terrain height would be limited to $\hat{h} \leq 1$, but the downstream height would be equal to its true value. As noted above, such asymmetric obstacles are allowed for in Smith's theory.

One objection to this interpretation is that it relies on blocking; that is, the propagation of disturbances upstream. This would seem to invalidate Long's hypothesis of no upstream influence.

Smith [102] adjusts the maximum value of $\hat{h}$ slightly to 0.985, and extends the theory to Froude numbers less than unity by redefining $\hat{h}$ as $\hat{h} = (h - d)N/U_0$. $\hat{H}_0$ then becomes
\[
\hat{H}_0 = \frac{3\pi}{2} + \hat{d}. \tag{2.116}
\]
Agreement between this modified theory and real or numerical experiments appears to be more satisfactory [102, 17] than linear theory.
Figure 2.3: Types of Atmospheric Wave Motion, adapted from [10]
2.2 Relevance of Stratified Flow over Hills

Flow over hills, stratified or otherwise, is of interest for reasons unconnected with the wave motions already mentioned. Hills have the effect of accelerating the approaching flow as it passes over them, much like an airfoil. It may be desirable to predict the magnitude of this effect in order to estimate the likely wind loading on structures, or to find a suitable site for wind energy devices. The dispersal of pollutants is also affected [29], more so if the stratification is stable (and therefore traps the pollutant near the ground instead of dispersing it). However, we shall concern ourselves principally with phenomena unique to stratified flows. These also have, as we shall see, effects further afield than the immediate vicinity of the hill itself.

Although in general invisible, mountain waves are sometimes revealed by their effect on cloud structures. As the pressure falls at the crest of the wave, water vapour is precipitated, which may form a cloud there. Such clouds then assume the shape of the wave crests. So-called 'lenticular' clouds (see figure 2.5 at the end of this section) are of this type. The wind underneath the wave crests is decelerated, and may even reverse direction (figure 2.4 below), while the waves gradually diminish in amplitude downstream of the obstacle. Thus, a long train of wave clouds sometimes forms.

![Figure 2.4: Lee Waves](image)

When a wide ridge rather than an individual peak is involved, the wave clouds will reflect this in shape and extent (see figure 2.6 at the end of this section). Such large cloud structures may be visible from space (see figure 2.8, for example), even on other planets. On Mars, clouds of ice crystals often reveal waves in the lee of craters and volcanoes (see [119], citing [88, 90, 9], and figure 2.7).

At very low Froude numbers, the fluid is so strongly stratified that its behaviour begins to resemble that of a free surface. Obstacles placed within the fluid shed horizontal von Kármán vortex streets [19], or sometimes 'ship waves', wedge shaped fields of waves resembling the wakes of ships [87]. Both these phenomena are observed in the atmosphere (see figure 2.8 for an example of a ship wave).
Sometimes the presence of a wave also can be deduced from the motion of birds and insect swarms affected by it [59],[89, p 14], [100, p 443-460]. Flocks of birds, in particular, can be tracked by radar (which the waves themselves generally cannot).

Although the clouds associated with them have long been known, lee waves themselves were not investigated closely until sailplane pilots took an interest in them in the 1930's [52][40, p 251]. They discovered that such waves could be used to soar to very high altitudes. Powered aircraft have subsequently found waves at altitudes of up to 20 km.

Mountain waves have subsequently come to be viewed more as a hazard to aviation than as a benefit. At high altitude, the waves themselves are an inconvenience - the aircraft is buffeted as it passes through them, which is usually merely uncomfortable, but sometimes results in injuries. They represent a disproportionate amount of clear air turbulence since they can travel from afar and need not be associated with any visible or otherwise detectable local weather.

At lower levels, standing waves in the lee of mountains are sometimes a much greater hazard. Rotors (regions of recirculating flow, and consequently, severe wind shear) can form under the crests of waves; these have been blamed for many accidents near mountain ranges, as have the lee waves themselves. In the United States the 11 western mountain states suffer from an accident rate exceeding that of the rest of the country by almost 40 % 5 (which is also more or less the proportion of accidents attributed to weather as a whole).

Since the propagation of wave energy upwards is accompanied by a flux of momentum downwards (see the Eliassen-Palm theorem above, particularly equation 2.43), mountain waves exert a drag force on the atmosphere. This effect was ignored in early global circulation models used in weather prediction [83]. This was largely due to the fact that the early models were limited by constraints of grid resolution, to the extent that the artificial drag due to numerical diffusion obscured the lack of physical drag due to gravity waves. As the models were refined, their accuracy initially deteriorated. It was eventually determined that gravity waves due to orography needed to be parameterized in some way to allow for the drag they caused.

A great deal of interest in breaking lee waves in recent decades has, however, centred not around waves travelling far from their point of origin, but around those that break in its immediate vicinity. These can be the cause of the already mentioned violent windstorms. Examples of such storms include a westerly downslope windstorm in the lee of the Pennines in 1962. This affected Sheffield to the extent that two thirds of all buildings in that city were officially recorded as having sustained damage [65, p 318]. Another example is that of a dust storm in the San Joaquin Valley near Bakersfield (California) in 1977. Surface winds gusting to 50 m/s were recorded, and dust rose to 1500m [119].

However, the example most frequently referred to is the storm of the 11th January 1972, in the lee of...
of the Rocky mountains. This chiefly affected the city of Boulder, Colorado. The interest shown is unsurprising, as the maximum wind speed (approximately 60 m/s, although the wind speed exceeded the limits of the instruments used to measure it) and the damage caused by it were, by any standards, remarkable. Detailed measurements were taken in and around the storm at the time, which have subsequently been used in a great number of simulations of it. These have been very successful in reproducing the main features of the storm; the outstanding question is not so much how such storms can happen (although there are a number of competing theories), but why they are in fact so rare [119].

Downslope winds of a gentler variety are, by contrast, very common. Such winds have historically been classified on the basis of temperature [50]. Often, such winds are warmer than the air they displace. Such warm downslope winds are known as Föhn in the Alps.

The warmth of downslope winds has historically been explained by reference to two different mechanisms; one relying on precipitation, and the other on the effects of stable stratification. The former assumes that the air windward of the topography in question is moist. As it rises, the decrease of hydrostatic pressure with altitude (as well as an extremely small nonhydrostatic fall in pressure caused by the acceleration of the flow over the obstacle) causes water to precipitate out as rain, thus releasing its latent heat of vapourization, which increases the potential temperature of the air. As the latter descends from the mountains, now (depending on the height of these) to a lesser or greater degree dry, it is compressed again, thus raising its actual temperature.

Assuming the air to windward to be completely saturated, a warming effect of approximately 4 °C per kilometre height of topography can be explained by this mechanism [65, p 320], although the possible maximum temperature rise is limited by the moisture content of the air (to approximately 8 or 10 °C; [100, p 420], [113, p 55]). However, this cannot be the only possible explanation, since downslope winds are sometimes very warm even in the absence of any precipitation, as well as, on occasion, simply too warm to be explained fully by it (whether or not it actually present). For instance, temperature rises of 20 °C have been recorded [65, p 320]; this requires an alternative explanation.

This is usually given in terms of the stable stratification of the air impinging on the mountain range. When such conditions exist, the potential temperature of the air rises with altitude. If the stratification is strong enough, some air at low altitude may not have sufficient kinetic energy to overcome the difference in potential energy represented by the height of the hill. What happens then depends on the aspect ratio of the obstacle - if it is not very wide (such as in the case of an isolated hill), the air may flow around it. If the obstacle is too wide to permit this (e.g. it is a mountain range), it will act as a dam, trapping the cold air on the windward side. Only air with sufficient potential energy (and thus, altitude) would be able to surmount the obstacle; since the

6 Some other names for such winds are: the Halny Wiatr of Poland, the Koschava and Ljuka of Yugoslavia, the Chinook of Colorado, the Santa Ana of California, the Zonda of Argentina, the Germich of the SW Caspian, the Afganet and lbe of Central Asia, the Kachchan of Sri Lanka, the Koembang of Java, and the Berg wind of South Africa.
air is stably stratified, its potential temperature would be greater than the air trapped windward of the obstacle. This effect is only limited by the strength of thermal stratification within the atmosphere.

The temperature of these winds is not directly related to the downslope windstorm conditions described above (although the 'blocking' explanation for their warmth relies on similar conditions being present). The effects of the temperature itself are less dramatic, but do include melting of snow and ice, and, since the moisture content of the air is reduced by the first of the two mechanisms described above, an increase in the risk of forest fires (particularly associated with the Santa Ana wind of California, for instance).

Sometimes, however, downslope winds are colder than the atmosphere through which they move. This can happen when the air is cooled at the surface, such that its stability then becomes very high. If permitted to descend, the airstream picks up more and more speed, as it loses potential energy. Such katabatic winds are very common in the Arctic and Antarctic.

The Bora of the Adriatic has also been regarded as a katabatic wind [113]. This wind descends on the coast from the Dinaric Alps, regularly reaching speeds of 50 m/s [113], and therefore a hazard to shipping and low flying aircraft. It is a cold wind, unlike the Föhn or the Chinook. Nonetheless, recent observations (such as those made during the ALPEX experiment) support the view that the Bora is a downslope wind of the same kind as these (see also [106]). One observation that led to this conclusion was that Bora winds sometimes accelerated uphill, which of course cannot happen if the source of the acceleration is katabatic.

There are also some (it must be said, quite circumstantial) reasons to believe mountain waves may be implicated in the development of severe hailstorms [62, p 243]. In order for hailstones to grow, they need to remain in the air for long enough for enough water to freeze on them. The longer they stay aloft, the larger they grow. For this, strong vertical winds are required to counteract gravity; it is possible to infer from the size of the largest observed hailstones that vertical winds of the order of 100 m/s must have been present to keep them aloft while they were growing to their final size.

Mountain waves (and the rotors associated with them) are one source of such winds. Buoyant convection is another, likely more important one. However, many regions of the world unusually prone to severe hailstorms are in fact situated in the lee of mountain ranges, where high vertical winds are to be expected as a result (for instance, the Po valley in Italy, the east coast of New Zealand, eastern Colorado, southern Alberta, and the west of Argentina).
Figure 2.5: Altocumulus Lenticularis.
Location: Rocky Mountains, Fort Collins, Colorado
Photo Date: February 21, 1940
Photographer: Mr. Maxwell Parshall
Photograph courtesy of the National Oceanic and Atmospheric Administration

Figure 2.6: Wave clouds extending from southwest to northeast. The base of the clouds is at 10,000 to 12,000 feet.
Location: Washington, D.C.
Photo Date: November 30, 1959
Photograph courtesy of the National Oceanic and Atmospheric Administration
Figure 2.7: Wave clouds in the lee of Perepelkin Crater
'Detailed Cloud Patterns in Martian Northern Hemisphere'
Location: 53 N, 65 W, Mars
Photo Date: June 4, 1998
Taken by: Mars Global Surveyor Orbiter
Catalog number: PIA01436
Photograph courtesy of NASA/JPL/Malin Space Science Systems
Figure 2.8: Ship Wave
'Stationary Atmospheric Waves'
Location: Bouvet Islands, 54 S, 3 E
Photo Date: March 27, 2000
Photograph courtesy of EUMETSAT
2.3 Upstream Influence in Stably Stratified Flows

Hills and ridges can be simulated in the laboratory using towing tanks. However, very stable \((F_h < 1)\) flows over such obstacles are believed to be affected by the upstream propagation of columnar modes (gravity waves of zero wavenumber) \([8, \text{p 179}]\). It has in fact been suggested (by Snyder et al, \([103]\)) that for two-dimensional obstacles that extend across the entire span of the tank, steady state conditions may never be attained, even intermittently, as previously suggested by Baines \([7]\).

This was attributed to the fact that since there is no way for the fluid to flow around the obstacle, it must flow over the top or under the bottom in order to satisfy continuity. The rate at which the obstacle moves may be infinitesimally slow, which was reasoned to imply that this is essentially a static phenomenon. As a consequence, fluid is assumed to be displaced along the entire length of the tank; in other words, the volume of fluid upstream of the obstacle is deformed in the same way regardless of the upstream distance. Hence this phenomenon was referred to as ‘squashing’. The consequence is to alter the stratification everywhere upstream of the obstacle, since the fluid is now stretched in the vertical direction. This is not a problem for obstacles small relative to the width of the tank, since the fluid can then flow around them. It is unclear whether this phenomenon can be identified with the columnar modes; if so, then the rate of propagation would be finite, and steady states attainable.

In the laboratory, the upstream extent of the blocked region is in principle limited by viscosity (see \([16] [111, \text{p 204}]\), but of the order of \(Re h/F_h^2\). For \(F_h < 1\), and \(Re\) high enough to be realistic, this tends to be too large to contain in a towing tank (for \(Re = 10^4, F_h = 0.7\), and \(h = 0.1m\), the resulting length would be 2km). In the real atmosphere, however, upstream effects are limited by Coriolis forces, to a distance of the order of the Rossby radius of deformation \(Nh/f\), where \(f\) is the Coriolis parameter.

Three-dimensional obstacles of finite width also suffer from such effects, although to a decreasing degree as the obstacle width decreases \([82]\) relative to the width of the channel through which it is towed. However, care must still be taken to avoid contamination of the solution by columnar modes reflected from the upstream boundary. These travel at a finite speed, so can be avoided if the upstream boundary is sufficiently far away, and the maximum time integrated to sufficiently low.

The dispersion relation for waves in a linearly stratified medium of finite depth is \([16]\):

\[
\omega^2 \left( k^2 + \frac{4n^2 \pi^2}{D^2} \right) - N^2 k^2 = 0.
\]  

(2.117)

For columnar modes, \(k = 0\), so

\[
\omega^2 \left( \frac{4n^2 \pi^2}{D^2} \right) = 0.
\]  

(2.118)
The frequency \( \omega \) is zero, but the group velocity is not
\[
c_g = \frac{\partial \omega}{\partial k} = \frac{N}{(k^2 + \frac{\pi^2 n^2}{U^2})^{1/2}} - \frac{Nk^2}{(k^2 + \frac{\pi^2 n^2}{U^2})^{3/2}}.
\] (2.119)

This is maximum for some low value of \( k \), and thereafter decreases monotonically with increasing \( k \) (see figure 2.9 for continuous plot of equation 2.119). If \( k = 0 \), equation 2.119 yields
\[
c_g = \frac{ND}{n\pi}.
\] (2.120)
c\(_g\) is the velocity at which wave energy is transported, relative to the fluid. Hence \( c_g \) must be greater than \( U \) for a particular mode to propagate upstream. Thus, for the fastest mode \((n = 1)\),
\[
K = \frac{ND}{\pi U} > 1.
\] (2.121)
The velocity of the wave energy relative to the upstream boundary is
\[
\frac{ND}{n\pi} - U.
\] (2.122)
On the way back from the boundary, instead of being retarded by the mean flow, the latter will be assisting the wave, so that it will propagate at
\[
\frac{ND}{n\pi} + U.
\] (2.123)
The total contribution from \( U \) will therefore cancel out. So, if the distance from the hill to the upstream boundary is \( L \), a total of
\[
\frac{2L}{UK}
\] (2.124)
units of time is available before arrival of the fastest columnar mode. In practice, we should like to be conservative, as some features of the flow just upstream of the hill may be interesting.
Figure 2.9: Group Velocity as a (continuous) function of Wavenumber, Equation 2.110
2.4 Experimental Data

The aim of the work described in this thesis is to simulate computationally the flows investigated by Castro and Snyder [17]. These workers simulated low Froude number flows over hills in the laboratory, using a towing tank. The flows were visualized using dye streamers. Linear density stratification was generated by means of salt, and obstacles of various shapes towed along the top of the tank. The obstacles with which we shall be concerned were of the following shape:

\[ h(x) = \frac{1}{2} h_m [1 + \cos(\pi x / L)], \]  

(2.125)

where \( h \) is the height of the hill, \( h_m \) the maximum height, \( x \) the coordinate in the streamwise direction, and \( L \) the axial length of the hill at the half-height point \( h = h_m / 2 \). In the spanwise direction, the hills were defined by a central two-dimensional section of width \( W_c \), with the profile given by equation 2.125 above. The ends were volumes of revolution generated by rotating the same profile through 180°.

![Figure 2.10: Hill Geometry](image)

In the table below, the aspect ratio \( \alpha \) is the ratio of \( L \) to the width of the hills at the half height point, and \( W_t \) is the width of the towing tank. The three hills COS1, COS2, and COS3 are the hills of those names in Castro and Snyder's paper, and it is for these that results will be presented below.

<table>
<thead>
<tr>
<th></th>
<th>COS1</th>
<th>COS2</th>
<th>COS3</th>
</tr>
</thead>
<tbody>
<tr>
<td>( h_m )</td>
<td>10.0 cm</td>
<td>10.0 cm</td>
<td>10.0 cm</td>
</tr>
<tr>
<td>( L )</td>
<td>18.5 cm</td>
<td>18.5 cm</td>
<td>18.5 cm</td>
</tr>
<tr>
<td>( W_c )</td>
<td>0 cm</td>
<td>25 cm</td>
<td>50 cm</td>
</tr>
<tr>
<td>( \alpha )</td>
<td>1.00</td>
<td>2.34</td>
<td>3.70</td>
</tr>
<tr>
<td>( W/W_t )</td>
<td>0.074</td>
<td>0.174</td>
<td>0.275</td>
</tr>
</tbody>
</table>

The depth of the channel \( D \) in all cases was 1 m. Castro and Snyder performed some tows with a lower depth, in order to investigate the effect of this variable. On the basis of these results, they felt able to conclude tentatively that the critical Froude number (at which wave breaking first took place) would not differ from the infinite-depth case by more than about 25%. They pointed out...
that reducing the ratio $h/D$ further, although in itself highly desirable, would reduce the Reynolds number also, and it would therefore be difficult to maintain Reynolds number independence.

The influence of the parameter $W/W_t$ was not investigated, but reasoned to be tolerably low. It was conceded, however, that it might not be entirely negligible for the wider hills. The widest hills in their study have not been investigated here.

Tows with Froude numbers varying from 0.1 to about 0.8 were undertaken. The Froude number was apparently varied by modifying the towing speed, so that results obtained for very low Froude numbers would also have exhibited low Reynolds numbers. Reynolds numbers for the flows investigated below were apparently of order $10^4$, based on the hill height. The lowest Reynolds number in the experimental work was about 1300.

All the flows (with the possible exception of the very widest obstacles, which have not been investigated here), were steady (presumably with the exception of turbulence). Critical Froude numbers $F_{hc}$ for the appearance of wave-breaking were obtained for the various obstacles, and plotted as a function of the aspect ratio $\alpha$. Both lower and upper critical Froude numbers were obtained - hence there presumably exists a minimum Froude number below which vertical motion is so inhibited that wave breaking cannot take place. However, the wave amplitudes are then so small that it is conceivably that some wave breaking could have taken place unobserved.

Another phenomenon noted was that of 'merging' flow, in which the wave breaking region aloft was observed to merge with the recirculation zone below. The table below summarizes the experimentally observed upper ($F_{hc}$) and lower ($F_{hl}$) critical Froude numbers, as well as any ($F_{hm}$) at which merging was observed, for the hills COS1, COS2, and COS3.

<table>
<thead>
<tr>
<th></th>
<th>COS1</th>
<th>COS2</th>
<th>COS3</th>
</tr>
</thead>
<tbody>
<tr>
<td>$F_{hc}$</td>
<td>0.3</td>
<td>0.7</td>
<td>0.7</td>
</tr>
<tr>
<td>$F_{hl}$</td>
<td>0.3</td>
<td>0.15</td>
<td>0.15</td>
</tr>
<tr>
<td>$F_{hm}$</td>
<td>N/A</td>
<td>N/A</td>
<td>0.6</td>
</tr>
</tbody>
</table>

Note that wave-breaking was marginal for the COS2 hill at a Froude number of 0.7.

The results were compared to the linear hydrostatic theory of Smith [102]. It was found that this overpredicted the upper critical Froude numbers substantially. Comparisons were also made with the nonlinear hydrostatic theory of Smith [101] detailed in section 2.1 above. This theory does not predict a critical Froude number, since wave breaking is assumed in its development. The height of the dividing streamline $H_0$ was compared instead. Agreement was found to be good, although the configuration of the experimental flows differed substantially from that postulated by Smith, in that the upper branch of the dividing streamline did not remain at its upstream height.
2.5 Literature Review

Gal-Chen and Somerville [37, 36] discretized the Navier-Stokes equations on a staggered mesh in general coordinates, using a contravariant velocity decomposition. They applied their method to the problem of thermal convection in the presence of a two-dimensional mountain, which gives rise to anabatic, or upslope, winds. A free slip boundary was applied on the hill, in order to avoid the need to resolve the boundary layer. Turbulence was modelled using constant eddy diffusivities for heat and momentum, with Prandtl numbers of 1 and also 3. The results were compared qualitatively with a Cartesian simulation and with observations of orographic convection, and found to be reasonable.

Peltier and Clark [86, 87, 22], investigated the dynamics of mountain waves, particularly in connection with the formation of severe downslope windstorms such as the Boulder windstorm of January 1972. They used the numerical model of Clark [20], which makes the anelastic approximation, so that the continuity equation becomes

\[ \nabla \cdot (\rho_0 \mathbf{u}) = 0. \]  

(2.126)

This allows the background density \( \rho_0 \) to be varied, which permits the calculation of flows of large vertical extent, where the standard incompressible form of the continuity equation is no longer satisfactory. However, equation 2.126 still neglects phenomena such as sound waves, which can only exist in fully compressible flow. This greatly increases the permissible time step size (at the cost of restriction to low Mach number flows, which is not a problem in this case). As in the work of Gal-Chen and Somerville [37, 36], a staggered mesh was used, as well as a terrain-following coordinate transformation. However, in this case, a Cartesian velocity decomposition was applied.

Turbulence was modelled using the Smagorinsky subgrid model, modified to allow for the reduction of eddy viscosity with stability. The original model is

\[ \nu_t = (C_S \Delta)^2 |S|, \]  

(2.127)

where \( C_S \) is a constant and \( \Delta \) is the filter length scale (that is, the maximum size of turbulent eddy to be modelled), in this case (as in most others when this model is used) related to the grid resolution by

\[ \Delta = \Omega^{1/D}, \]  

(2.128)

where \( \Omega \) is the cell volume or area (for a two-dimensional simulation) and \( D \) is the number of

---

\footnote{Essentially the reverse of katabatic winds, in that both are driven by surface heat fluxes of opposite sign, giving rise to buoyancy forces and hence vertical velocities also of opposite sign.}
dimensions. The quantity $|S|$ is the magnitude of the total deformation

\[ S_{ij} = \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right), \tag{2.129} \]

defined as

\[ |S| = \sqrt{\frac{1}{2} S_{ij} S_{ij}}. \tag{2.130} \]

This model was modified by Clark and Peltier in the following manner, as suggested by Lilly [57]:

\[ \nu_t = (C_s \Delta)^2 |S| \sqrt{1 - Ri}, \tag{2.131} \]

where $Ri$ is the gradient Richardson number. It is perhaps for this reason that Baines [8] calls this model 'essentially inviscid'; for sufficiently stable stratification, no viscosity will be present at all, unless the Richardson number is locally reduced to below unity.

The eddy diffusivity was modelled by assuming a Prandtl number of one. The boundary condition on the lower boundary is free-slip, that on the outlet is a Sommerfeld radiation condition of the type described by Orlanski [78]. This boundary condition relates the space and time derivatives of a variable on the boundary as follows:

\[ \frac{\partial \phi}{\partial t} = c \frac{\partial \phi}{\partial n}, \tag{2.132} \]

where $n$ is the direction normal to the boundary and $c$ is a phase velocity normal to the boundary. This is calculated from the interior solution as suggested by Orlanski. An additional constraint of global mass conservation is also imposed.

The top and inlet boundary conditions are conventional (variables fixed, free slip). However, artificial Rayleigh friction was introduced near these boundaries in order to prevent wave reflection from them.

A steady-state linear model was also developed by approximating the atmosphere as a stack of layers, within each of which the density and velocity are constant. This model was tested against analytic solutions for a homogenous atmosphere, and also compared to solutions obtained using the finite-difference model described above. It was found that the theory initially agreed with these qualitatively, even for weakly supercritical flows, but that the result resembled nonlinear solutions obtained with rather lower mountains. However, as the nonlinear model was integrated in time, the solutions for supercritical flows diverged markedly from the theoretical prediction, with increasing surface drag and wind speeds.

When the finite-difference model was initialized with measurement data from the 1972 Boulder windstorm, the general features of the predicted flow field were found to agree very well with
observations. The maximum computed wind speed (58 m/s) was within a few percent of the observed maximum. Surface drag was found to exceed linear predictions by a factor of 20, as waves of far larger amplitude were generated. It was argued that this was good evidence that the storm was a consequence of nonlinear effects.

Peltier and Clark attributed the high drag state not to wave reflection from the tropopause, as Klemp and Lilly did [50], but from the stagnant and turbulent region generated by the breaking wave itself.

The use of linear theory in the wave-drag parameterizations employed in general circulation models was criticized on these grounds. According to the Eliassen-Palm theorem 2.44, linear theory implies that momentum is transported upwards without attenuation unless a linear critical level exists where the wave is predicted to break. Since, as their work shows, nonlinear effects can cause waves to create their own critical level, Peltier and Clark reason that this will lead to error in those cases. This happens, moreover, when wave amplitudes, and consequently momentum flux, are particularly high and accurate parameterization of most consequence.

They tested their theory in [22] by performing numerical experiments similar to their previous work [86] described above, but this time including a critical level at which the wind reversed direction, set so as to force the mountain wave to break at that level. It was found that this led to solutions differing significantly from linear theory only in those cases where the critical level height was at the same height as the self-induced critical level, or an integer number of vertical wavelengths in excess of it. This was taken as evidence that resonant reflection from the critical level was involved in the nonlinear amplification observed in those cases.

Smith [101], however, argues that, were this the case, resonance would also occur at critical level heights differing from the self-induced level by half a wavelength. This was not observed. Instead, Smith presents the hydraulic analysis reproduced above in section 2.1, and shows that it is consistent with the results of Peltier and Clark [22].

The same workers [87] also considered the development of atmospheric ‘ship-waves’ in the lee of isolated islands, once again applying the finite-difference model outlined above. Measurements and observations pertaining to the islands of Jan Mayen and Bear Island (in the Norwegian and Barents Seas respectively) were used to initialize the model. For these cases, it was found that both linear and nonlinear models reproduced the observations well. The linear model employed was, however, different from the one described in [86]; instead of approximating the atmosphere as a series of layers, this model was based on perturbation equations similar to 2.18, 2.19, and 2.20 above, but including effects of compressibility. Solutions to these were obtained using a spectral method.

Clark and Farley [21] extended the finite-difference model of Clark [20] to include nested grids. Subdomains within the large scale grid were solved separately with a different grid resolution;
communication between the two solutions was effected by internal boundary conditions. This permitted them to greatly increase grid resolution in regions where it was particularly needed.

This model was then used to simulate the 1972 Boulder windstorm in three dimensions. It was found that the solution was notably more unsteady than the two dimensional one. Although this was qualitatively in better agreement with observations (which included pronounced gusts), the computed gusts were somewhat too strong (the strongest observed gusts were of about 65 m/s, while the model produced at least one gust of 80 m/s). The discrepancy was attributed partly to neglect of moisture in the air upstream of the mountain, and partly to a lack of grid resolution in the horizontal direction. A decrease in surface drag was also found in the three dimensional solution when compared to the two dimensional one.

Scinocca and Peltier [98] presented yet more simulations of the Boulder windstorm. These were two-dimensional, but differed from previous work in that the influence of domain size and boundary conditions was systematically investigated, and also in that the simulations were continued to much greater (model) times than the previous ones.

It was found that the domain size affected the results considerably, as did the depth of the sponge layers used to prevent wave reflection from boundaries. It was reasoned, therefore, that this effect must be important. Numerical experiments were performed to determine the domain size and sponge layer depths that would be adequate. With a sufficiently large domain, the results were found to converge approximately with respect to the predicted wave drag. No solution could ever be reproduced in detail, however; this was attributed to chaotic dynamical effects. The depth of the sponge layers on the top and upstream boundaries was found to have only a transient effect. However, since the effect was fairly strong, extended sponge layers were also introduced.

The authors drew attention to the fact that such an investigation had never been carried out before for a simulation of this nature. The results obtained, however, were acknowledged to be particular to the model employed and the flow simulated. Indeed, the wave drag histories plotted show very little dependence on the domain size until well into the simulation, when previous computations (such as the ones described by Peltier and Clark [86]) had already been terminated.

As already noted, the time to which this simulation was integrated greatly exceeded similar previous calculations. About $5^{3/4}$ hours were simulated, as opposed to $2^{1/4}$ by Peltier and Clark [86] and slightly more than $2^{1/2}$ by Clark and Farley [21]. It was found that only after about 200 minutes did the wave drag reach saturation; after this time, it fluctuated strongly, but no longer displayed any definite trend. The conclusion was drawn that the two-dimensional flow was dynamically unstable, and that there was therefore no need to explain the strong gusts observed in windstorms in terms of three-dimensional instabilities, as was done by Clark and Farley [21]. Indeed, it was suggested that the unsteadiness observed in their simulation was possibly two-dimensional in origin.
Durran and Klemp [26, 27] employed a two-dimensional numerical model based on the compressible form of the equations of motion. The expense of simulating the sound wave modes was partially avoided by integrating these separately from the terms of meteorological interest. A much larger time step could therefore be used for the latter. Topography was included using a coordinate transformation, and a free slip boundary condition imposed on the surface. A subgrid scale turbulence closure similar to that of Peltier and Clark [86] was used, which sets an eddy viscosity on the basis of local shear and stability.

This code was used to simulate stably stratified flows over mountain ranges, in particular stratifications with a multilayered structure. It was found that such structure enhanced nonlinear effects. The Boulder windstorm was also simulated, with several modifications to the upstream profile of stability in order to determine the factors important in the development of the storm. It was found that presence or absence of an elevated inversion was critical.

Durran found that linear theory was not only generally in error, but also that the sign of the error varied. The analogy between the windstorm configuration and (nonlinear) hydraulic theory was emphasized, as was the height of the critical layer (when one was present).

Durran and Klemp compared their results for various critical level heights to the theories of Peltier and Clark [86] and Smith [101], finding them to be inconsistent with the former, but generally in good agreement with the latter. An analogy were drawn between Smith's approach and hydraulic theories such as that of Houghton and Kasahara [43], based on the shallow water equations. The analogy was found to have some quantitative predictive power with regard to the critical Froude number for transition to supercritical flow, which is associated with downslope windstorms.

Apsley and Castro [5, 2] simulated stably stratified flow over an isolated hill (Cinder Cone Butte), and the dispersion of a passive scalar released upwind of it. The Reynolds-averaged Navier-Stokes equations were discretized on a curvilinear staggered grid, and a Cartesian velocity decomposition employed. Control volume faces were positioned halfway between nodes (the cell-vertex scheme). Central differencing was used for diffusive fluxes, and the van Leer scheme for advection.

Buoyancy was modelled using the Boussinesq approximation, and the flow therefore assumed incompressible. The $k$-$\epsilon$ model was used for turbulence closure. The constants used were the same as those of the standard, high Reynolds number form of Launder and Spalding [53], apart from the following modifications:

1. The $\epsilon$ Prandtl number $\sigma_\epsilon$ was set to a value of 1.11, to be consistent with a Kármán constant of 0.4 with regard to equation 2.187.

2. The $\epsilon$ production term was modified by replacing the constant $C_{\epsilon 1}$ with

$$C_{\epsilon 1} + \left(C_{\epsilon 1 a} - C_{\epsilon 1 b}\right) \frac{l}{l_{\text{max}}},$$

(2.133)
where $l$ is the mixing (strictly speaking, dissipation) length $C_\mu^3 k^{3/2} / \epsilon$, and $l_{\text{max}}$ some maximum value determined on the basis of stability or Coriolis forces. In this application $l_{\text{max}}$ was made a function of the Monin-Oboukhov length.

The behaviour of the passive scalar was found to be in good agreement with laboratory simulations. Simple dispersion models used by regulatory authorities were found to be inadequate, however.

The limited length-scale formulation was also applied to calculation of the neutrally stratified atmospheric boundary layer [2], and neutrally stratified flow over hills [3], generally with success, insofar as the results were much improved over those obtained using the standard $k$-$\epsilon$ model. However, it was necessary to specify the maximum length scale a priori.

Paisley and Castro [79, 81, 80, 82] used the same finite volume code to simulate a variety of two and three-dimensional stratified flows of finite depth. The solution was advanced in time using backward Euler time stepping. Turbulence closure was effected by mixing length (using the formula 2.146) and $k$-$l$ turbulence models, modified as follows for stratification:

\[
\begin{align*}
\text{Eddy Viscosity} & \quad Ri \\
0 & \quad Ri > R_i C \\
\nu_t \left(1 - \frac{R_i}{R_i C}\right)^2 & \quad 0 \leq Ri \leq R_i C \\
\nu_t \left(1 - Ri\right)^{1/2} & \quad Ri < 0
\end{align*}
\]

where $R_i$ is the local gradient Richardson number, and $R_i C$ a critical value above which turbulence must cease. It can be shown [67, 44, 2] that a Richardson number exceeding 0.25 everywhere is a sufficient condition for this. This is the value suggested by Paisley and Castro [81].

The flows simulated were principally those previously investigated experimentally by Castro and Snyder [17], which are also the subject of this thesis. Reasonable agreement was obtained; however, it was found necessary to increase the Reynolds number substantially over that used in the experiments, in order to obtain a grid-independent solution. The mixing-length model was found to be particularly affected, and also to be generally more diffusive than the $k$-$l$ model.

The occurrence of 'merging' flow, in which a breaking wave merges with a rotor beneath it, was also investigated. A feature resembling a hydraulic jump is then formed, resulting in very high levels of turbulence and drag. The computations were able to reproduce this behaviour, at least in a qualitative sense. Interestingly, it was not found to be present in two-dimensional computations, suggesting a three-dimensional mechanism is responsible.

Huser et al [46] calculated the flow in a valley with a view to predicting the dispersal of pollutants from proposed road developments. The atmosphere was considered stable and the surface rough; the modified $k$-$\epsilon$ model of Duynkerke ([28], described in [46]) was used for turbulence modelling.
These workers found the results encouraging from a qualitative point of view, but concluded that either the assumed surface roughness or the turbulence model constants (or the isotropic nature of the eddy viscosity model) required further adjustment.

Kim et al [48] carried out wind tunnel experiments on neutral flow over two-dimensional ridges, and also on two ridges running parallel to each other. They also used several $k$-$\varepsilon$ models to simulate the same flows numerically, and compared the results to those of the experiment and to the linear theory of Jackson and Hunt [47].

Measures were taken to assess the effect of false diffusion, since the hybrid differencing scheme was used for advection. This scheme reduces to the first order upwind scheme for cell Peclet numbers higher than two, and is therefore diffusive. Evidence of numerical diffusion was indeed found; however, it was also determined that this was reduced by the use of an orthogonal grid. This is to be expected, since in neutral flow over hills streamlines tend to follow the surface contours in much the same way such a grid would do. Numerical diffusion is greatly reduced if this is the case, since upwind schemes of low order are badly affected if the grid is skewed with respect to the flow\(^8\).

All turbulence models were found to perform satisfactorily for attached flows. The situation was different for steep hills over which the flow separated. When the standard $k$-$\varepsilon$ model was used, the simulated recirculation zone was too small. However, the reverse was true when the low Reynolds number and RNG models were used. Although Kim et al did not comment on the fact, it also appears that the trend of the RNG model results with regard to the hill height was not very satisfactory - the recirculation zone length was larger as a fraction of the hill height when the latter was increased, rather than smaller, as indicated by the measurements and the other models.

Montavon [74] applied a RANS solver to the problem of stably stratified flows over 2-D mountain ranges. Modifications were made to permit the use of potential temperature as the density-determining scalar, in order to be able to simulate flows of large vertical extent (several kilometres). Turbulence was treated using the $k$-$\varepsilon$ model. The solver was validated by comparison with linear theory (with which good agreement was found) and with non-hydrostatic mesoscale models (with good agreement also). Finally, comparison was made with observations of the Boulder windstorm of 1972 [50]. Agreement was found to be good both with observations and also with the results of Peltier and Clark [86].

The results of both models agree in respect of the maximum wind speed (60 m/s, which is also roughly the maximum observed), and its location, the wavelength of the lee waves (25km), and the height of the flow reversal zone (5km).

\(^8\)In problems where such skewness is absent by definition (such as the one-dimensional convection-diffusion equation) it is possible to show that first order advection schemes can be constructed that are exact, whereas central differencing (for instance) will not be [84]. Such schemes are not, however, accurate in multidimensional flows.
Kim and Patel [49] simulated neutrally stratified flow over complex terrain with a variety of two-equation turbulence models. The terrain considered was that of the Sirhowy valley in Wales, an embankment on the Rhine, and the Askervein Hill in Scotland. The models used were the standard high Reynolds number $k$-$\epsilon$ model, the modified model of Duynkerke [28], the RNG $k$-$\epsilon$ model of Yakhot and Orszag [120], the preferential dissipation modification $k$-$\epsilon$ model of Leschziner and Rodi [55], and the $k$-$\omega$ model [117]. Only the first three were, however, applied to any cases involving real terrain; the latter two were only tested against some idealized configurations (an empirical law for the atmospheric boundary layer, and a triangular ridge). The embankment and Askervein test cases were only simulated using the RNG model.

The RNG model was stated to have been found to yield the most satisfactory results. However, its nonlinear eddy-viscosity formulation failed to produce physically plausible results in the near-wall region, and was therefore not used. Comparisons with other models were also rather limited, since only the triangular ridge was simulated with all five models. For this case, the RNG model (with linear eddy-viscosity formulation) produced results most closely in agreement with measurements. The model of Duynkerke notably failed to simulate the atmospheric boundary layer successfully, which is surprising, since that was its intended purpose.
2.6 Turbulence Modelling

Flows over real hills are always at very high Reynolds numbers whenever winds are high enough to be significant. Turbulence is, therefore, expected. This may either be directly simulated, partially simulated and partially modelled, or wholly modelled. These three approaches are represented by direct numerical simulations (DNS), large eddy simulations (LES), and turbulence models, of which the eddy viscosity models are the most prevalent.

Direct simulation of turbulence is the most generally accepted method, but is not practical for any flows of geophysical relevance, as the number of mesh nodes becomes prohibitively large with increasing Reynolds number. In order to resolve all turbulent eddies in an arbitrary flow, it is necessary to store information on a number of points given by

\[ n = \left( \frac{L}{l} \right)^3, \tag{2.134} \]

where \( L \) is the scale of the largest turbulent eddies and \( l \) that of the smallest. Unfortunately, this range increases with Reynolds number; from experiment

\[ L \sim \frac{U^3}{\epsilon}, \tag{2.135} \]

where \( U \) is the reference velocity and \( \epsilon \) the rate at which the kinetic energy of the turbulent eddies is dissipated by viscosity. This happens only at very small scales, since, at high Reynolds number, the large eddies are little affected by viscosity. If we accept that viscous dissipation can be neglected for all but the smallest eddies, we must conclude that, for turbulence in equilibrium, each length scale transmits its kinetic energy to smaller length scales. Otherwise, the energy contained in vortices of a particular lengthscale would grow without bound. This establishes a connection between the large turbulent motions, which extract energy from the mean flow, and the small eddies that dissipate this energy to heat.

The small scales at which dissipation takes place are also the smallest found in a turbulent flow, since they will have no kinetic energy left to transmit to yet smaller eddies. A measure of the scale at which dissipation becomes important is the Kolmogorov scale [35, p 91, 103]:

\[ l = \left( \frac{\nu^3}{\epsilon} \right)^{1/4}, \tag{2.136} \]

where \( \nu \) is the molecular viscosity. If equation 2.135 is used to eliminate \( \epsilon \), the following is obtained:

\[ l \sim \left( \frac{L^{3/4}L^{1/4}}{U^{3/4}} \right). \tag{2.137} \]

Assuming \( L \) and \( l \) to represent the approximate order of magnitudes of the largest and smallest length scales present in a turbulent flow, we find that their ratio is

\[ \frac{L}{l} \sim \left( \frac{L^3U^3}{\nu^2} \right)^{1/4} \sim Re^{3/4}. \tag{2.138} \]
Hence the number of point required to resolve a turbulent flow fully varies roughly as

\[ n \sim Re^{2/4}, \]  

(2.139)

and, given that the time step needs to vary with the spatial resolution for good accuracy, we conclude that the computational effort required for a direct numerical simulation varies roughly as the cube of the Reynolds number. This makes them impractical for environmental or geophysical flow problems, in which the Reynolds number is always large, owing to the very large length scales involved.

One way of avoiding this expense is to simulate only the very largest eddies, and to use some kind of model (called the subgrid model) to describe those not resolved. This is the approach of large eddy simulation, which is in fact used for flows such as the ones treated in this work. It remains difficult, however, to obtain adequate resolution, even with this approach. The number of grid points required for a LES of a channel flow is roughly related to those required for a DNS by [118]:

\[ n_{LES} \simeq \left( \frac{0.4}{Re^{1/4}} \right) n_{DNS}. \]  

(2.140)

Sufficient resolution for a large eddy simulation could not be afforded for this work, so an approach based on the Reynolds-averaged Navier-Stokes equations was used instead. These are obtained by decomposing the instantaneous turbulently fluctuating velocity into a mean part \( U \) and a fluctuating part \( u' \), and the pressure similarly into a mean \( P \) and fluctuating component \( p' \). These averages are defined as follows:

\[ \overline{\phi} = \frac{1}{\Delta t} \int_0^{\Delta t} \phi dt, \]  

(2.141)

for the average \( \overline{\phi} \) of a quantity \( \phi \). It is necessary to choose a time \( \Delta t \) over which to average. For a flow expected to asymptotically approach a steady state, an approximation to infinity is a reasonable choice. For a flow expected to exhibit unsteady features other than turbulence, this time must be large enough to capture the turbulent fluctuations, but not so large that it captures other unsteadiness. A clear separation of scales is therefore required. Assuming this is so, we can obtain the Reynolds equations from the Navier-Stokes equations, which are (ignoring buoyancy):

\[ \frac{\partial \rho u_i}{\partial t} + \nabla \cdot (\rho u_i u) = -\frac{\partial P}{\partial x_i} + \nu \nabla^2 \rho u_i \]  

(2.142)

in each component \( u_i \) in coordinate direction \( x_i \). Substituting \( U_i + u'_i \) for \( u_i \) and \( P + p' \) for \( P \), we find

\[ \frac{\partial \rho U_i}{\partial t} + \nabla \cdot (\rho U_i U) = -\frac{\partial P}{\partial x_i} + \nu \nabla^2 \rho U_i - \frac{\partial u'_i u'_j}{\partial x_j}. \]  

(2.143)
The result simply amounts to the replacement of the fluctuating velocity component \( u_i \) by the time-averaged \( U_i \), everywhere but in the last term. This is obtained from the advection term \( \nabla \cdot (\rho u_i u_j) \), and is a consequence of the fact that the turbulently fluctuating components \( u' \) advect each other. If these were to be random and independent of each other, these terms would cancel, since the average \( u'_i u'_j \) would be zero. However, this is not the case in general. These nine extra terms are, therefore, unknown, and require modelling in some way, since no additional equations are available to determine them.

One simple model for these terms is to augment the molecular viscosity with a turbulent ‘viscosity’ \( \nu_t \). This could in principle model the \( u'_i u'_j \) terms for \( i \neq j \), so that

\[
\frac{\partial \rho U_i}{\partial t} + \nabla \cdot (\rho U_i U) = -\frac{\partial P}{\partial x_i} + (\nu + \nu_t) \nabla^2 \rho U_i - \frac{\partial u'_i u'_j}{\partial x_i}.
\] (2.144)

A separate model is then usually used for \( u'_i u'_j \); since they act in the same way as pressures, they are sometimes simply absorbed into the pressure term.

One way of prescribing \( \nu_t \) is to specify a constant for it, but this rarely satisfactorily, as the ratio of the stresses \( u'_i u'_j \) to the mean velocity gradient varies widely in turbulent flows. A more common formulation is the mixing length model, which makes the eddy viscosity a function of an algebraically prescribed length:

\[
\nu_t = l_m^2 \left( \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} \right)^{\frac{1}{2}}.
\] (2.145)

Although simple turbulence models of the mixing length type have been used to calculate flow over hills, the results have not always been found to be entirely satisfactory. Even for the case of a neutrally stable atmosphere, the computed drag is usually found to be too high [12, 11]. The most common mixing length formulation for atmospheric boundary layer flows is that of Blackadar:

\[
l = \frac{\kappa z l_0}{\kappa z + l_0},
\] (2.146)

where \( l \) is a mixing length, \( \kappa \) the Kármán constant, \( z \) the height above the surface, and \( l_0 \) some maximum value of \( l \).

In the case of the stable atmospheric boundary layer, parameterizations based on the mixing length model exist [73, 25, 45], and are widely used. However, these are no help for turbulence caused by wave activity (see section 2.7 below). This takes place well away from any boundary, so that a mixing length based on the distance from a surface is inappropriate. If the location and form of the turbulent region were known a priori, a mixing length might be prescribed, but this has never been attempted; such an approach would in any case not only be very arduous, but also open to criticism on the grounds of excessive empiricism. Methods have been devised which permit calculation of a mixing length in general flows [38], but these are very expensive in terms of the computational effort required.
Mixing length models have other serious deficiencies. Local equilibrium must be assumed everywhere (that is, turbulence is not transported by the flow, but is always dissipated in the same place it forms). In very stable boundary layers the length scale also becomes increasingly decoupled from the surface [34], so that the distance from the latter is no longer an appropriate scaling parameter.

Wave-breaking effects are better studied with a turbulence model of the complete form; that is, one that obtains both length and velocity scales of turbulence from differential equations of transport. The reason for this is that it is not practicable to prescribe these in advance for such a flow, as it is not known a priori where, or even whether, such behaviour may occur. The simplest turbulence models of this kind are two-equation models, of which the most well known is the \( k-e \) model. The aim of this work was to employ this model, and variants of it, to simulate the flows investigated experimentally in [17]. Turbulence models developed for neutral shear flows must generally be appropriately modified when used to calculate stratified flows [4, 39, 95].

The precise model to be used will, therefore, possess some modifications for stably stratified flows. The modified model will, however, reduce to the original one in the absence of stratification, so that any claims the existing model may have to generality (even if not great) will not be affected. Several modifications have been investigated; it is believed that most have already been applied to other flows, although not the ones to be considered here. One modification has not, apparently, ever been used previously.

As stated above, the flow of the atmosphere over hills is characterized by high Reynolds numbers. Moreover, hills are generally aerodynamically rough. These properties favour turbulence models that avoid integration through the linear sublayer adjoining a wall. The high Reynolds number implies a thin sublayer, and thus high grid resolution. The presence of roughness presents a more fundamental difficulty, because there is then no sublayer, and thus no alternative to the use of wall functions of some kind.

Low Reynolds number \( k-e \) models are, therefore, likely not to be suitable for computations of flows over real hills. However, they may be appropriate for laboratory experiments, which are generally carried out at much lower Reynolds number. The experiments of Castro and Snyder [17], with which this thesis is much concerned, involved Reynolds numbers of 1300 to 20000, based on the hill height. Since these flows were stratified, the Reynolds number is not the only relevant parameter though. In any case, we need to assume that any laboratory experiments were carried out at sufficiently high Reynolds numbers to represent flows over real hills. Simulations involving low Reynolds number turbulence models might give some indication of whether that is the case - but if it is, high-Reynolds number models are probably more appropriate.

The use of nonlinear \( k-e \) models is not often successful when wall functions are also employed (see [49]), particularly in three-dimensional flows. Some such models require evaluation of second derivatives [104], which is difficult if sufficient grid resolution is not available in regions where
they are high, such as near walls.

Furthermore, nonlinear $k$-$\epsilon$ models generally neglect stratification, and it is not always clear in what way they ought to be modified to incorporate it.

The influence of stratification may enter naturally when two-equation models are derived by simplifying a differential Reynolds stress transport model. Algebraic stress models of various kinds result [96]. However, time did not permit further investigation of these.

For stratified flows, all Reynolds averaged models may suffer from the scale-separation problem discussed above; if the Brunt-Väisälä frequency becomes low enough to permit wave motion at length and time scales comparable with those of turbulence, it is not possible to use Reynolds averaging to separate the two. It is quite possible, therefore, that even very expensive turbulence models will fail to simulate such flows very satisfactorily.

The turbulence model chosen for the work described here was the $k$-$\epsilon$ model, with some modifications appropriate for stably stratified flows. This model will be described below.

2.6.1 The $k$-$\epsilon$ Model of Turbulence

The $k$-$\epsilon$ model is an eddy viscosity turbulence model. Such models augment the molecular viscosity by a turbulent viscosity $\nu_t$, so that additional terms appear in the momentum equations. These represent the Reynolds stresses, so that

$$-\overline{u'_iu'_j} = \nu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \kappa \delta_{ij}, \quad (2.147)$$

where $\nu_t$ is the eddy viscosity, and $\delta_{ij}$ is a Kronecker delta (unity for $i = j$, zero for $i \neq j$). The turbulence kinetic energy $k$ is equal to half the sum of the normal Reynolds stresses

$$k = \frac{1}{2} \overline{u'_iu'_i} = \frac{1}{2} (u'^2 + \bar{v}'^2 + \bar{w}'^2). \quad (2.148)$$

The term in $\delta_{ij}$ is required to make the expression 2.147 applicable to the normal stresses. We want these to sum to the turbulence kinetic energy, as expressed by equation 2.148, but the eddy viscosity does not ensure this:

$$-\overline{u'_iu'_i} = 2\nu_t \frac{\partial U_i}{\partial x_i}, \quad (2.149)$$

the sum of which is equal to

$$k = \nu_t \nabla \cdot \mathbf{U}. \quad (2.150)$$
which is zero by continuity. The terms $\frac{2}{3} k \delta_{ij}$ sum to $k$, so that equation 2.148 is satisfied. This is implemented in practice by redefining the static pressure as follows

$$p + \frac{2}{3} k.$$  (2.151)

The kinetic theory of gases expresses the molecular viscosity as a function of two parameters - a velocity scale, and a length (or time) scale. By analogy, it is assumed that two such scales suffice to determine the eddy viscosity also. The $k$-$\varepsilon$ model is one of a family of two-equation models that employ an eddy-viscosity based on a velocity and either a length or time scale of turbulence, both of which are obtained from differential equations of transport. This approach was first proposed by Kolmogorov in 1942 [118, p 84], who used as the velocity scale the square root of turbulence kinetic energy, and as a time scale the inverse of a representative frequency of the turbulent motions, given by $\omega \sim k^{1/2}/l$. Kolmogorov argued that this quantity, which describes the time scale of the large scale turbulent motions that extract energy from the mean flow, may be related to the time scale of the small scale motions at which energy is dissipated. An equation for the dissipation rate of turbulence kinetic energy would then be an appropriate way of determining the time scale.

The equations governing the kinetic energy of the mean flow $U$, and of the turbulent fluctuations $u'$ can be obtained in an analogous manner to Eliassen and Palm’s derivation of the wave energy equation 2.38 above. The (Reynolds-averaged) Navier-Stokes equations are multiplied by $U$ to obtain the equation for the mean flow kinetic energy. These are then subtracted from the (instantaneous) Navier-Stokes equations multiplied by $u'$ to obtain the equation for the turbulence kinetic energy [109, p 63], [72, p 53]. The result is

$$U_j \frac{\partial}{\partial x_j} \left( \frac{1}{2} u_i u'_i \right) = - \frac{\partial}{\partial x_j} \left( \frac{1}{\rho} u'_i p' + \frac{1}{2} u'_i u'_j - \nu u'_i \left( \frac{\partial u'_i}{\partial x_j} + \frac{\partial u'_j}{\partial x_i} \right) \right)$$  (2.152)

$$- u'_i u'_j \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \beta g u'_i \phi - \nu \left( \frac{\partial u'_i}{\partial x_j} + \frac{\partial u'_j}{\partial x_i} \right)^2.$$  (2.153)

The terms in equation 2.153 may be interpreted as follows. The LHS is the advection of $u_i u'_i$, the normal Reynolds stresses. The first term on RHS represents the correlation of the fluctuating velocity components $u'_j$ with the fluctuating pressure $p'$. This is known as pressure diffusion. The second term is the advective transport of the normal stresses by the fluctuating velocity components. The third term is molecular diffusion, which is followed by production and dissipation terms (described further below).

Summing equation 2.153 for $i = 1, 3$, we obtain a transport equation for the sum of the normal Reynolds stresses, $2k$. This must be modelled so that it may be expressed in terms of known quantities. This means that the fluctuating velocity components $u'_j$ and the fluctuating pressure $p'$ must be removed.
The term in $\overline{u_j^i p}$ cannot be directly measured. It is therefore added to the turbulent transport term $\overline{u_i^i u_j^j}$, and the sum assumed to obey a gradient-transport law of the form

$$\frac{1}{2} \overline{u_i^i u_j^j} + \overline{u_j^j p} = \nu_k \frac{\partial k}{\partial x_j}.$$  \hspace{1cm} (2.154)

The viscous diffusion term is simply

$$\nu u_i^i \left( \frac{\partial u_j^i}{\partial x_j} + \frac{\partial u_j^j}{\partial x_i} \right) = \nu \frac{\partial k}{\partial x_j}.$$ \hspace{1cm} (2.155)

The fourth term on the RHS is the rate at which work is done by the mean strain rate against the turbulent stresses, hence the rate at which the kinetic energy of the mean flow is transferred to turbulence. This is called the turbulence production term (denoted $P_k$ below), and is equal to the modelled turbulent diffusion term in the momentum transport equations, so that

$$P_k = \overline{u_i^i u_j^j} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) = \nu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j}.$$ \hspace{1cm} (2.156)

The fifth term represents the exchange between turbulence kinetic energy and potential energy. Here $\phi$ is a scalar quantity representing either temperature or species concentration, whichever determines variation in density responsible for buoyancy. Typically, a transport equation will be solved for $\phi$, which will be related to $\rho$ by an equation of state (such as the ideal gas law for temperature, or the Knudsen and Unesco formulae for salinity), the constant of proportionality for a given set of conditions being $\beta$ \textsuperscript{9}. This would be the volumetric expansion coefficient were $\phi$ the temperature, for instance.

This term will be denoted $G_k$, and is modelled as

$$G_k = \beta \overline{u_i^i u_j^j \phi} = \beta \nu \frac{\partial \phi}{\partial x_i}.$$ \hspace{1cm} (2.157)

The final term is the dissipation rate $\epsilon$. This is determined by its own transport equation (see below).

Applying these approximations, we obtain the following model equation for $k$:

$$\frac{\partial k}{\partial t} + \frac{\partial \rho U_i k}{\partial x_i} = P_k - G_k - \rho \epsilon + \frac{\partial}{\partial x_i} \left( \nu \frac{\partial k}{\partial x_i} \right).$$ \hspace{1cm} (2.158)

Derivation of the $\epsilon$ equation is a less straightforward affair. It is possible to derive it from the dissipation rate equation in homogenous turbulent flow, by applying rather drastic modelling

\textsuperscript{9}It does not follow that the relationship need be linear, since $\beta$ may be a function of other quantities, including $\phi$ and $\rho$. 

50
assumptions [72, p 55]. However, the assumption of homogenous turbulence is itself an extremely restrictive one. The model that results is perhaps most fairly regarded as an analogy to equation 2.158. Whichever way it is arrived at, the model $\varepsilon$ equation is given by

$$\frac{\partial \varepsilon}{\partial t} + \frac{\partial \rho u_i \varepsilon}{\partial x_i} = C_{1\varepsilon} \frac{\varepsilon}{k} (P_k - \rho C_{\mu} G_k) - \rho C_{\varepsilon} \frac{e^2}{k} + \frac{\partial}{\partial x_i} \left( \rho \nu \frac{\partial \varepsilon}{\partial x_i} \right).$$

(2.159)

The equations 2.158 and 2.159 represent the $k$-$\varepsilon$ model of Launder and Spalding [53].

It remains to define the constants $C_\mu, C_{1\varepsilon}, C_{\varepsilon 1}, C_{\varepsilon 2}, C_{\varepsilon 3}, \sigma_k$, and $\sigma_\varepsilon$. Calibration can be performed either by requiring the model to reproduce a small number of simple flows, for which the model equations yield analytical results for one or more constants, or by optimizing the model to minimize the error over a very large number of different flows. Both approaches were used, but only the first will be described here in detail.

The very simplest possible turbulent flow is that of decaying homogenous turbulence, which may be (approximately) produced by passing a grid through a quiescent fluid. All mean velocities and their gradients are zero, so the model equations reduce to

$$\frac{\partial k}{\partial t} + \varepsilon = 0$$

(2.160)

$$\frac{\partial \varepsilon}{\partial t} + C_{\varepsilon 2} \frac{e^2}{k} = 0.$$  

(2.161)

These equations have solutions of the form

$$k = k_0 t^{-m_1}$$

(2.162)

$$\varepsilon = \varepsilon_0 t^{-m_2},$$

(2.163)

so that

$$\frac{\partial k}{\partial t} = -m_1 k_0 t^{-m_1 - 1},$$

(2.164)

$$\frac{\partial \varepsilon}{\partial t} = -m_2 \varepsilon_0 t^{-m_2 - 1}.$$  

(2.165)

We substitute the RHS of 2.164 for $\varepsilon$ in 2.160 and 2.161, and the RHS of 2.162 for $k$. The time derivatives are replaced by the LHS of 2.164 and 2.165. Then

$$\varepsilon_0 t^{-m_2} = -m_1 k_0 t^{-m_1 - 1},$$

(2.166)

$$-m_2 \varepsilon_0 t^{-m_2 - 1} = C_{\varepsilon 2} \frac{e^2}{k_0 t^{-m_1}}.$$  

(2.167)
For this to be true, the following must be the case:

\[
\begin{align*}
\epsilon_0 &= m_1 \kappa_0, \\
m_2 \epsilon_0 &= \frac{C_{e2} \epsilon_0^2}{k_0}, \\
m_2 &= m_1 + 1.
\end{align*}
\] (2.168, 2.169, 2.170)

Hence \( C_{e2} \) is

\[
C_{e2} = m_2 \frac{k_0}{\epsilon_0} = \frac{m_2}{m_1} = \frac{1 + m_1}{m_1}.
\] (2.171)

Experimentally, \( m_1 \) is found to be approximately 1.2. Hence \( C_{e2} \approx 1.83 \).

We can obtain a further constraint by considering a local equilibrium shear layer. In such a flow, the production \( \overline{P_k} \) and dissipation terms \( \epsilon \) balance. Therefore

\[
\overline{P_k} = \nu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} = \epsilon.
\] (2.172)

From the definition of an eddy viscosity

\[
\nu_t \frac{\partial U_i}{\partial x_j} = \overline{u_i' u_j'}. \] (2.173)

Only one Reynolds stress is nonzero in a two-dimensional shear layer, so we can write this as

\[
\frac{\partial U}{\partial z} = \frac{\overline{u' w'}}{\nu_t}. \] (2.174)

Substituting this into equation 2.172 above, we obtain

\[
\epsilon = \frac{C_{\mu} k^2}{\nu_t} = \nu_t \left( \frac{\overline{u' w'}}{\nu_t} \right)^2. \] (2.175)

Therefore

\[
C_{\mu} = \left( \frac{\overline{u' w'}}{k} \right)^2. \] (2.176)

The quantity \( \overline{u' w'}/k \) is a structure constant found experimentally to be equal to approximately 0.3 in many shear flows, at or near equilibrium. This implies \( C_{\mu} = 0.09 \).

Next, we consider the logarithmic law, which holds in turbulent boundary layers close to a wall. There, the shear stress is constant, and the following velocity profile holds
\[ U = \frac{u_r}{\kappa} \ln(E^+n^+), \]  

(2.177)

where \( \kappa \) is the Kármán constant, \( u_r \) the friction velocity, \( E^+ \) is a constant (which depends on the surface roughness), and \( n^+ \) is a nondimensional height above the surface. \( u_r \) and \( n^+ \) are given by

\[ u_r = \sqrt{\frac{\tau}{\rho}}, \]  

(2.178)

\[ n^+ = \frac{n u_r}{\nu}, \]  

(2.179)

where \( n \) is the distance from the wall.

Differentiating 2.177 with respect to \( n \), we obtain

\[ \frac{\partial U}{\partial n} = \frac{u_r}{\kappa n}. \]  

(2.180)

Substituting this result into the model k-equation 2.158, we obtain

\[ \nu_t \left( \frac{\partial U}{\partial n} \right)^2 + \frac{\partial}{\partial n} \left( \frac{\nu_t k}{\sigma_k} \frac{\partial k}{\partial n} \right) = \epsilon = 0. \]  

(2.181)

Doing the same with the \( \epsilon \) equation 2.159 yields

\[ \frac{C_{11} \nu_t}{k} \left( \frac{\partial U}{\partial n} \right)^2 + \frac{\partial}{\partial n} \left( \frac{\nu_t k}{\sigma_k} \frac{\partial \epsilon}{\partial n} \right) - \frac{C_{12} \epsilon}{k} = 0. \]  

(2.182)

The solution to equations 2.181 and 2.182 is

\[ k = \frac{u_r^2}{\sqrt{C_{12}}}, \]  

(2.183)

\[ \epsilon = \frac{u_r^3}{\kappa n}. \]  

(2.184)

Substituting these into the \( \epsilon \) equation 2.159, we obtain a further constraint on the model constants, as follows:

\[ C_{11} u_r \kappa n \left( \frac{\sqrt{C_{12}} u_r^2}{\kappa n} \right)^2 + \frac{\partial}{\partial n} \left( \frac{u_r \kappa n}{\sigma_{\epsilon}} \left( -\frac{u_r^3}{\kappa n^2} \right) \right) - \frac{C_{12} \sqrt{C_{12}} u_r^4}{u_r^3 \kappa^2 n^2} = 0. \]  

(2.185)

\[ \frac{C_{11} \sqrt{C_{12}} u_r^4}{\kappa^2 n^2} + \frac{u_r^4}{\sigma_{\epsilon} n^2} = \frac{C_{12} \sqrt{C_{12}} u_r^4}{\kappa^2 n^2}. \]  

(2.186)

\[ \kappa^2 = \sigma_{\epsilon} \sqrt{C_{12}}(C_{12} - C_{11}). \]  

(2.187)
Equation 2.187 can be used as a constraint on \( C_{\nu} \) by requiring a particular value of the Kármán constant \( \kappa \). In addition to the constraints described above, Launder and Spalding performed extensive optimization with respect to many flows. The following constraints were arrived at:

\[
\begin{align*}
C_{\mu} & = 0.09 \\
C_{\alpha} & = 1.44 \\
C_{\alpha 2} & = 1.92 \\
\sigma_k & = 1.00 \\
\sigma_\varepsilon & = 1.30
\end{align*}
\]

The constant \( C_{\alpha 3} \) is a special case, and not part of the original model (which neglected density stratification). With the \( k \) and \( \varepsilon \) equations as constituted above, it is found that a single value is not appropriate for arbitrary stratification [96, 95]. For stable stratification, zero is apparently indicated, whereas unity is more satisfactory for unstable stratification. A value of zero will therefore be used in general, although the influence of this parameter will be investigated by using a value of unity to obtain some of the results below.

### 2.6.2 Modelling the Influence of Stability on Turbulence

Some modifications are made to the standard \( k-\varepsilon \) model to account for the effects of stability. It can be shown that the constant \( C_{\mu} \) should vary with stability. We return to the local equilibrium shear layer 2.172, but now permit \( G_k \) to be nonzero. The sum of production and dissipation terms must still be zero, so

\[
P_k + G_k = \epsilon. \tag{2.188}
\]

Using the same substitutions as in section 2.6.1 above, we obtain

\[
\nu_t \left( \frac{u'w'}{\nu_t} \right)^2 + G_k = \epsilon. \tag{2.189}
\]

We introduce the flux Richardson number, which is the ratio of turbulence production by buoyancy to that by shear:

\[
R_f = \frac{-G_k}{P_k}, \tag{2.190}
\]

which permits us to write equation 2.189 as

\[
\nu_t \left( \frac{u'w'}{\nu_t} \right)^2 (1 - R_f) = \frac{C_{\mu} k^2}{\nu_t}. \tag{2.191}
\]

Therefore

\[
C_{\mu} = \left( \frac{u'w'}{k} \right)^2 (1 - R_f). \tag{2.192}
\]
We have assumed that the structure constant $\bar{u}'w'/k$ is approximately 0.3, in unstratified flow. Making this assumption again, we obtain

$$C_\mu = 0.09(1 - R_f).$$

(2.193)

The assumption that the structure constant is the same in non-stratified and stratified flows is somewhat questionable. It appears that it might be somewhat reduced in stable flows [95]. However, equation 2.193 already represents a fairly strong effect, at least on this particular constant.

Equation 2.193 does show that flux Richardson number is a natural parameter to adopt as a measure of stability, particularly in the context of the turbulence kinetic energy equation. It is the ratio of two terms in that equation, rather than a merely a property of the local scalar and velocity fields, like the gradient Richardson number. The relationship between the two is

$$R_f = \frac{-G_k}{P_k},$$

(2.194)

$$R_f = \frac{\nu_i}{\sigma_t \partial x_i} \left[ \nu_p \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} \right]^{-1},$$

(2.195)

$$R_f = \frac{1}{\sigma_t} R_i,$$

(2.196)

where $\sigma_t$ is the turbulent Schmidt or Prandtl number.

$R_i$ is an important factor determining transition to turbulence, since it is possible to show that instabilities leading to turbulence cannot occur in parallel shear flow if its value exceeds $1/4$ everywhere (the Miles-Howard criterion, [67, 44, 2]). However, as soon as transition to turbulence actually occurs, $R_i$ becomes different everywhere and fluctuates; nor can it be expected that the turbulent velocity field will contain only parallel shear flows. If Reynolds averaging is used, this may be true for the mean flow, but then the fine structure in $R_i$ cannot be simulated explicitly. In fact, in the flows to be simulated, the (molecular) Schmidt number is very large, which means that the smallest scales on which structure in the density field occurs are very much smaller than the smallest possible turbulent eddies. Hence, the value of $R_i$ everywhere may not be available even to simulations that resolve these eddies.

If $R_i$ is available only in a mean sense, it ceases to have the same meaning it possesses as an intensive property, and indeed, whether much meaning remains is debatable. Existing turbulence is not suppressed for mean gradient $R_i$ exceeding $1/4$. Very little turbulence would be observed in the ocean and atmosphere were this the case, yet the atmosphere is not laminar below about 85 km. Rather than suppressing turbulence entirely, stratification appears to confine it to small patches of intense turbulence [66]; this only makes any averaging procedure even more suspect.

Although in practice an averaged $R_i$ is used to predict probable atmospheric turbulence, that is not because of the Miles-Howard criterion, but for want of measurement techniques able to
determine other, more meaningful quantities (such as accurate values of $Ri$, or the flux and stress Richardson numbers [13]). It is then assumed that an averaged $Ri$ continues to have the same qualitative meaning as an intensive one, and some empirically or otherwise obtained value (like 1) is used instead of $\frac{1}{\tau}$.

Returning, however, to the $k$-$\varepsilon$ model, the question arises whether model constants other than $C_j$ might be functions of stability. In general, we might expect that constants appearing only in the $\varepsilon$ equation would not be affected by stability. This is because $\varepsilon$ describes the behaviour of small eddies, which are of small vertical extent. The amount of work such eddies do against gravity should therefore also be small, whereas the work done against viscous stresses is large (the Reynolds number for such an eddy being small). Indeed, it is found experimentally that stability does not appear to affect the decay of grid turbulence [14, 58]. The parameter controlling this decay is, as we have seen, $C_2$. Hence we may conclude that it at least is largely unaffected.

Some caution is in order, however, since the purpose of the $\varepsilon$ equation is to prescribe a length scale for the large scale eddies, however indirectly. The model constants have been tuned so as to give good results for the mean flow, rather than for the turbulence quantities. The constant $C_\varepsilon$ is itself an example of this - the value adopted lies outside the bounds of experimental error established for it by the decay of grid turbulence. However, there appears to be little positive evidence that $C_\varepsilon$, $C_2$, or $\sigma$ are much affected by stability (although admittedly there does not appear to be much negative evidence either in respect of $C_\varepsilon$ and $\sigma$).

The constant $C_\varepsilon$, as we have seen, is indeed a function of stability. This must be considered regrettable. For some flows, the sign of the stability might be known everywhere in advance, but for many others this is not the case. The flows to be investigated here fall into the latter category, since they are on the whole stably stratified, but expected to develop unstably stratified regions with time. The data available at this time do not appear sufficient to establish any clear functional relationship between $C_\varepsilon$ and a quantity such as $R_f$. Rodi [96] suggests that instead of using $G_k$ in the definition of $R_f$, the production of the 'lateral' component $\overrightarrow{v'}$ be used instead, and the flux Richardson number defined as $R_f = G_{\overrightarrow{v'}}/(P_k + G_k)$. It is unclear exactly what is meant by 'lateral'. It appears that this direction is required to be in the direction of gravity in a horizontal shear layer, but normal to it in a vertical one. Hence it might be defined adequately as parallel to the local momentum gradient. If this is done, a single value of $C_\varepsilon$ can apparently be used.

However, time did not permit further investigation of this matter. Instead, most of the results to be presented were obtained using $C_\varepsilon = 0$, although the relevance of this parameter was investigated by also obtaining some with $C_\varepsilon = 1$.

The turbulent Schmidt or Prandtl number has long been known to be a function of stability [73, 39, 31, 32]. Nevertheless, numerical modellers generally assume a constant value, usually unity. This would appear to be hard to justify. Hence, one of the aims of this work will be to investigate
the effect of this assumption. The turbulent Schmidt number is therefore multiplied by a function $f_s$, which is either unity or some function of stability.

The semi-empirical function of Ellison [31] was chosen for this purpose. It is given by

$$f_s = \frac{(1 - R_f)^2}{1 - R_f / R_{fc}}$$  \hspace{1cm} (2.197)

where $R_{fc}$ is the critical flux Richardson number. A value of 0.2 was always used when this function was employed.

Equation 2.197 is believed to be a good approximation for moderate to large stability. The form of the $R_f/Ri$ curve implied by it is shown in figure 2.11 below; Figure 2.12 is a plot of $R_f$ against $f_s \sigma_t$ as given by equation 2.197. It is clear that the deviation from unity is rather substantial for large $R_f$.

The effect of modifying the turbulent Schmidt number in the way described is to permit turbulence to persist to some extent, even under conditions of very high stability (although it then causes very little mixing). This effect manifests itself in two parts of the $k$-$\varepsilon$ model described above. Firstly, the buoyancy production $G_k$ is limited, since $\sigma_t$ occurs in its denominator. Secondly, the flux Richardson number is now limited to the value of $R_{fc}$, so we might expect equation 2.193 to be much less effective in limiting the eddy viscosity.

In order to determine the importance of the second mechanism relative to the first, the following modification to 2.193 was tested:

$$C_n = 0.09 (1 - R_f / R_{fc})$$  \hspace{1cm} (2.198)

This compensates for the limit imposed on the flux Richardson number by equation 2.197. There is no particularly good physical justification for this, however.
Figure 2.11: $R_f$ vs. $R_i$, Equation 2.197

Figure 2.12: Turbulent Schmidt number $f_c \sigma_t$ vs. $R_f$, Equation 2.197
2.7 **Summary of Earlier Work with another Method**

The development of the numerical model described above was motivated by earlier investigations using the numerical model of Paisley and Castro [79, 81, 80, 82]. This model has been successfully applied to the flows investigated by Castro and Snyder [17], which are also the subject of the current work. These investigations are described above in section 2.5.

One feature of all of the work that had been done previously with this model was the use of either the mixing-length or of the $k - l$ model of turbulence. Although the salient features of the flows investigated appear to have been reproduced satisfactorily (with the latter model, at least), their treatment with these models is not very appealing from a theoretical point of view; both require that the turbulence lengthscale be prescribed by the investigator. Realistic models for $l_m$ exist for engineering boundary layer flows (where $l_m$ is assumed to vary linearly with the distance from the wall, in the immediate vicinity of the wall) and for atmospheric boundary layers (where the Blackadar relation is often used).

However, prescriptions made on the basis of the location of the turbulent flow considered, relative to some fixed point, are not particularly appropriate for recirculation zones away from any fixed boundary, such as those giving rise to the downslope windstorm configuration investigated here (see section 2.6 above). The logical approach for such a problem would be to define $l_m$ within the recirculation zone in terms of the geometric properties of the rotor and not those of the boundary layer.

However, this is obviously very inconvenient, since the location, extent, and other properties of the rotor are not known a priori, but rather the very unknowns that are the prime object of the investigation. Paisley and Castro chose to use the Blackadar relation (equation 2.146) instead. This means that a recirculation zone aloft, sufficiently far from the surface to be outside the boundary layer, will be simulated using roughly the value of $l_m$ to which that equation asymptotes. Since this value is that used for the extreme outer edge of the boundary layer, the size of the turbulent eddies for which it would be appropriate are of the order of the thickness of the boundary layer. There is no particular reason why it would be appropriate for the rotor.

The mismatch would presumably be particularly great if the breaking zone were to be very small, and the true lengthscale therefore also necessarily small, since it is bounded by the size of the breaking region, which is surrounded by laminar flow above and below. The eddy viscosity would then be greatly overestimated. This additional source of diffusion would suppress any sharp gradients in the solution, thus possibly preventing a high-drag state from being formed.

It might be reasoned that the $k - l$ model should be much less susceptible to this that the mixing-length model, since the eddy viscosity is then also a function of turbulence kinetic energy as well as of $l_m$. This would limit the diffusive effects of the mixing length prescription to regions where $k$ assumes significant values. Paisley and Castro did indeed obtain much more satisfactory results.
using this model than using a mixing length, and found that the high-drag state was suppressed when the latter was used.

These reasonably encouraging results suggest that the results could be improved on further by proceeding to a two-equation model of the complete form, which would remove any need to prescribe the mixing length. Although there is no guarantee that the length-scale determining transport equation would prove superior to the prescription previously adopted, there are at least two reasons for believing that it would be a more promising approach. One is that the fundamental basis for the prescription is not very satisfactory, and the other is that a similar transport equation for $k$ was found to perform better than the simple mixing length model. It was therefore considered important to implement a two-equation model for this work.

This was in fact done, by implementing the $k$-$\epsilon$ turbulence model. However, it proved impossible to obtain converged solutions for the geometry to be considered, unless very high (5 %) background turbulence intensity was imposed upstream of the obstacle. This is obviously unsatisfactory, since the experiments were conducted by towing an obstacle through a tank of quiescent fluid. The appropriate value for the turbulence intensity upstream of the obstacle is, therefore, zero.

It was found that converged solutions could be obtained for the same geometry using the mixing length and $k-l$ models. It was also found that converged solutions could be obtained using the $k$-$\epsilon$ model on grids that were not, or only weakly, curvilinear. This appeared to indicate a problem with the implementation of the $\epsilon$ equation on curvilinear grids, but despite extensive scrutiny, none was discovered. Nor was the $\epsilon$ equation observed to diverge before the others.

Since the difficulty only arose when turbulence levels were low, it seemed likely that the problem might be the appearance of negative values of the turbulence variables $k$ and $\epsilon$. This explanation was, however, ruled out, as it was determined that such negative values did not precede divergence, and modifications preventing negative $k$ and $\epsilon$ did not affect the problem.

Eventually, explanations involving other variables were sought. It was found that the pressure gave by far the clearest indication of nonphysical behaviour immediately before the solutions diverged. A very sharp peak in the value of the pressure was observed to develop just upstream of the hilltop and outside the boundary layer, in a region where the curvature of the grid was relatively high, and the $z$-component of momentum highest. This peak developed before anything untoward was observed in the solutions for momentum and the turbulence equations.

The numerical method used by Paisley and Castro (based on an earlier code by Apsley [2]) solved for the Cartesian velocity components on a curvilinear staggered grid, using the SIMPLE algorithm for the pressure solution. For reasons which have been discussed above in section 3.2, this is not a stable discretization on sufficiently strongly curvilinear grids, the part of the algorithm affected being the pressure solution.
On the available evidence, it was concluded that the discretization used was most likely the fundamental reason for the problem. It proved difficult to show exactly why the problem only manifested itself when the $k$-$\varepsilon$ was used. However, when the flow that proved impossible to simulate using the model of Paisley and Castro was instead computed using a commercial CFD code, the eddy viscosity calculated using the $k$-$\varepsilon$ model in the region where the pressure peak had appeared was found to be very low. The effect of this would be to reduce the diffusion of momentum, thus preserving sharp gradients in the velocity field, which might be caused by an unsmooth pressure solution.

This is unsurprising, because the flow just windward of the hilltop is subjected to very strong acceleration, which damps turbulence. Hence the aspect of the $k$-$\varepsilon$ model that proved difficult may have been one that is in fact somewhat physically realistic.

In fact, it seems not unlikely that those aspects of the $k$-$l$ and mixing length models that render them somewhat undesirable for the modelling of wave breaking, are in fact the same as those that enable them to simulate flows on moderately curvilinear grids using potentially unstable discretizations. As noted above, these models impose much too large an eddy viscosity on small regions of the flow that develop high velocity gradients (and hence turbulence), particularly if these are not very near a wall. Such small regions may be due to wave-overturning, but also to high-frequency errors that would be caused by a divergent pressure solution. If these errors could be smoothed away by strong diffusion, convergence might be obtained using a method that would otherwise diverge.

This explanation is also consistent with the observation that high background turbulence values permitted a converged solution to be obtained even using the $k$-$\varepsilon$ model.

Since the use of something like the $k$-$\varepsilon$ model was considered essential, the decision was made to produce a new code. This had some added benefits; although the solver of Apsley was fully three dimensional, that of Paisley and Castro was not. Since the laboratory experiments were three dimensional, it was considered essential to simulate them as such, particularly since the results indicated strong effects due to the ratio of hill width to height. It had also been found that three-dimensional laboratory experiments suffered less from problems associated with the propagation of disturbances upstream than did two-dimensional ones. Any such problems would most likely also affect numerical simulations. It was therefore decided to make the new code three-dimensional.

Some work on the numerical methods used in the earlier code was also done. This mainly centred around the implementation of multigrid methods for the turbulence transport equations. Such methods transfer intermediate solutions from the original grid to coarser grids for further solution; this improves the rate at which low frequency errors are removed. Another transfer method is

$^{10}$A region of very high eddy viscosity was, however, found just above the location of the peak.
then used to correct the original solution using the coarse grid one. Transfer from fine to coarse
grid is called 'restriction', and coarse to fine grid 'prolongation'.

It is not felt that an in-depth discussion of multigrid methods is appropriate here, since none
were used in the method presented below. However, it was determined in the course of the earlier
work that prolongation techniques in common use could be modified to perform better with fairly
unsmooth solutions such as are typically obtained for variables with large source terms such as $k$
and $\epsilon$.

Usually, the prolongation method interpolates an absolute correction from the coarse grid to the
fine one. The solution on the fine grid is then modified by this correction. It was found to be
beneficial to interpret the correction as a relative value instead; that is, as a fraction of the fine
grid solution. This ensured, in particular, that large corrections were not applied to small values
of the variable thus treated. Large corrections to small values might be expected to be particularly
troublesome for $\epsilon$, since the rate at which the eddy viscosity varies with $\epsilon$ approaches infinity as
$\epsilon \rightarrow 0$. The limited tests that were carried out did suggest that the modification was beneficial in
turbulent flows.
Figure 2.13:
Paisley and Castro Model (with $k$-$\varepsilon$ turbulence closure).
Hill is of the shape of the COS hills, but of half the height. This is to slow down the rate at which the solution diverges, so that the manner in which it does so could be studied.
Figure 2.14: Solution obtained using a commercial CFD code.
3. Method

3.1 The Finite Volume Method

The finite volume method is based on the integral form of a general conservation law. For some
discrete control volume of size \( \Omega \), bounded by a surface \( S \), such a law can be written as [42, p 10]:

\[
\frac{\partial}{\partial t} \int_{\Omega} \phi \, d\Omega - \int_{S} \mathbf{F} \cdot dS = \int_{\Omega} Q_{s} \, d\Omega + \int_{\Omega} Q_{q} \, d\Omega.
\] (3.1)

Where \( \mathbf{F} \) is the flux through \( S \), and source terms are represented by \( Q \). These have been further
subdivided into volume sources \( Q_{q} \), which are sources in the usual sense of the word (such as
body forces, in the case of the momentum equations), and surface sources \( Q_{s} \), of which a good
example would be the integral of the pressure over the volume surface.

The scalar \( \phi \) will be conserved in the absence of such sources; hence, the fluxes \( \mathbf{F} \) over a control
volume surface will sum to \( d\phi/dt \), the time rate of change of the conserved quantity within the
volume. If this is true, then the transport equation is said to be conservative.

It also follows that the integrated time rate of change of \( \phi \) of a number of adjacent control volumes
equals the sum of the fluxes through their external surfaces only; that is, that the fluxes through
surfaces joining adjacent control volumes volumes should cancel. Conversely, if a control volume
is subdivided, the internal fluxes that result shall cancel. This is called the ‘telescoping property’
of the flux terms [42, 94], and ensures that a conservation law will be satisfied on both a local
and a global basis when, as is the practice in finite volume methods, a domain is subdivided into
a set of control volumes.

Invoking Gauß’s divergence theorem (and therefore assuming the first derivative of \( \phi \) is continuous
everywhere on the surface \( S \), 3.1 can be written

\[
\frac{\partial}{\partial t} \int_{\Omega} \phi \, d\Omega + \int_{\Omega} \nabla \cdot \mathbf{F} \, d\Omega = \int_{\Omega} \nabla \cdot Q_{s} \, d\Omega + \int_{\Omega} Q_{q} \, d\Omega.
\] (3.2)

from which it is possible to obtain the differential form of the conservation law in the limit \( \Omega \to 0 \).

\[
\frac{\partial \phi}{\partial t} + \nabla \cdot \mathbf{F} = \nabla \cdot Q_{s} + Q_{q}.
\] (3.3)

Although this is the usual form for a conservation law, 3.1 is more general, since it is not necessary
to assume that the spatial derivatives of $\phi$ are continuous. Otherwise, the two forms are equivalent, and a discretization of 3.3 should once again yield 3.1. However, it is not necessary to view the discretization as an approximation to 3.3; rather, a discrete approach can be viewed as being directly derived from 3.1. Either equation is, however, conservative.

It is also possible to derive alternative forms of the differential conservation equation 3.3 that are mathematically equivalent to it, but do not arise out of 3.1 in the limit $\Omega \to 0$. These forms will not be considered in detail here (see [42] for details), except to draw attention to the fact that a discretization not of the form 3.1 will not in general be conservative, and equations that are not directly derived from it (in the manner of 3.3) will not be in conservation form. By basing our method on 3.1, however, we ensure that it shall be conservative.

It is clear that the conservative form is more physically meaningful, since it corresponds clearly to a physical law of some kind (conservation of mass, for instance, or momentum or energy). Whether it is necessary to use it in practice is less clear. Roache [94, p 32] cites a number of cases where the conservative form has been found to lead to more accurate results. Of course, if the criteria for accuracy amount to satisfaction of the conservation law, this is perhaps tautological, if nonetheless welcome. However, experience appears to indicate that this does appear to be true for more general criteria also. Thus, one case cited is [110], where a conservative method of only first order accuracy in the advection terms was found to be more accurate than a second order accurate non-conservative method for cavity flows driven by buoyancy and a moving lid. Another reason often given [42] is the failure of non-conservative methods to satisfy the Rankine-Hugoniot relations in the presence of a shock, which is of great importance in the simulation of compressible flow through the (inviscid) Euler equations (although not so pertinent to the flows to be considered in this work¹).

In incompressible flows, the cost of using a conservative discretization is small, so a small advantage is sufficient to justify its use². Based, as they are, directly on 3.1, finite volume methods are conservative by construction. They are therefore preferred when this property is desired, particularly when irregular or curvilinear geometries render Cartesian grids, and finite difference methods based on them, unsatisfactory.

In practice, the source and flux terms appearing in equation 3.1 need to be approximated in some way. The accuracy of the approximation determines the rate at which the discretization error is reduced as the size of the control volumes is reduced. As a general rule, if the approximation is

¹Although a hydraulic jump, which is a phenomenon to be examined here, has often been compared to a shock wave [8, p 26,77]. The Froude number is analogous to the Mach number insofar as it is a ratio of a flow speed to a wave speed, and in that a discontinuity (accompanied by high drag) appears when the ratio falls from above unity to below. However, the analogy breaks down at that point, since the relationships valid across the discontinuity are not the same [8, p 35].

²This is because most of the additional cost of the conservative form is due to the need to always use $pu$ inside the differential operators, rather than $u$. This cost only manifests itself in compressible flows. However, as noted above, the benefits are then clearer also.
exact for polynomials of degree \( p \), then the leading term in the remaining error varies as \( \Delta x^{p+1} \), where \( \Delta x \) is the length of the control volumes into which the domain has been divided [42, p 168,277] (for control volumes of unit aspect ratio - the accuracy may not be as good if they are stretched). Hence the logarithmic rate at which the error decreases with respect to \( \Delta x \) is \( p + 1 \). This is usually called the order of accuracy. We might therefore expect, for instance, that approximations exact for linearly varying \( \phi \) will be second-order accurate in space, and this is in fact found to be the case [33, p 73], [42, p 219].

The details of most of the approximations made will be described below. It is appropriate, however, to discuss some fundamental aspects here. The surface of each control volume is divided into several faces, each with associated surface vector \( S \). The fluxes through each of these are integrated as follows:

\[
\int_S F \cdot dS = F_C \cdot S, \tag{3.4}
\]

where \( F_C \) is the value of \( F \) at the centre of the cell face. This approximation is exact for \( F \) varying linearly over \( S \), and therefore satisfactory for an otherwise second-order accurate method. Surface sources are evaluated in the same way, with the central value of \( Q_S \) replacing \( F \) in the above. Volume sources are approximated as

\[
\int_\Omega Q_\Omega \, d\Omega = Q_{\Omega,C} \Omega, \tag{3.5}
\]

where \( Q_{\Omega,C} \) is the value of \( Q_\Omega \) at the control volume centre. The time dependent term is treated analogously. This, again, is linearly accurate.

This approximation (for both volumes and surfaces) is known as the midpoint rule. It is not adequate for methods of higher than second order; in such cases, the integrand must be evaluated at more than one place on the surface or within the volume, and a numerical integration method applied (Simpson’s rule is sometimes used [33, p 70]). However, the work to be presented below relies on the midpoint rule, and approximations of better than second order accuracy are not attempted.

When, for a particular control volume \( P \), we sum all the flux and source terms of equation 3.1, we obtain an algebraic equation relating the value of \( \phi_P \) to \( \phi \) in neighbouring control volumes. Ignoring time-dependence (which will be addressed in section 3.4.1 below), this may be written

\[
A_P \phi_P + \sum_F A_F \phi_F = Q_S + Q_\Omega, \tag{3.6}
\]
where the $A_F$ terms are the flux coefficients for each control volume face $F$, and

$$A_P = - \sum_F A_F.$$  \hfill (3.7)

Considering all the control volumes in the domain, we obtain a system of equations all of the form of 3.6. This can be written in matrix notation as

$$A\phi = Q,$$  \hfill (3.8)

where $A$ is the matrix of which $A_P$ and $A_F$ are coefficients (the latter are then the diagonal elements). With the addition of suitable boundary conditions, this may now be solved by any of a great number of methods suitable for solving systems of linear equations. The matrix $A$ is sparse, that is, most of the elements are zero, reflecting the fact that each control volume exchanges fluxes only with its immediate neighbours. This may be taken advantage of in the solution algorithm.

When the governing conservation equation is nonlinear, the coefficients $A_F$ will in general themselves be functions of $\phi$. This is usually addressed by the use of iterative solution methods; an approximation to the solution of 3.8 is obtained, the coefficients of the matrix $A$ re-evaluated, and the process repeated until the solution converges (no longer changes with additional iterations).

The governing equations of fluid dynamics are the Navier-Stokes equations. We shall restrict ourselves to incompressible flows; when written in the conservation form 3.3 these equations are

$$\frac{\partial U}{\partial t} = \nabla \cdot F_U + \nabla \cdot P + Q_U,$$  \hfill (3.9)

$$\frac{\partial \phi}{\partial t} = \nabla \cdot F_\phi + Q_\phi,$$  \hfill (3.10)

$$\nabla \cdot U = 0,$$  \hfill (3.11)

where $U$ is the velocity vector, and $\phi$ the scalar quantity determining density (energy, for instance, or concentration). The sources have been divided into volume sources $Q_U, Q_\phi$, and the pressure, $P$, which is a surface source for momentum.

Section 3.4 below will deal with each of these terms in turn; the time dependent term, the fluxes $F$ (advective and diffusive), and the pressure. First, however, we should establish how the computational domain should be subdivided and the discrete control volumes constructed.
### 3.2 Velocity Decomposition and Grid Arrangement

One of the first and most obvious decisions to make is whether to use the same grid for all the quantities to be computed. This is obviously much simpler than defining different grids. However, it has not always been a popular approach. The reason is that, unless special measures are taken, the pressure solution develops unphysical wiggles, as the pressure and momentum equations decouple from each other.

To explain this behaviour, we note that the pressure is a surface source in the momentum equations, and therefore required on cell faces. Consequently, it is necessary to interpolate it. Consider three adjacent nodes 0, 1, and 2 on a uniform grid; the pressure on the face between 0 and 1 is

\[ p_{0.5} = \frac{1}{2}(p_0 + p_1). \]  

(3.12)

The pressure between 1 and 2

\[ p_{1.5} = \frac{1}{2}(p_1 + p_2). \]  

(3.13)

We subtract these two to find the net source

\[ \Delta p_1 = \frac{1}{2}(p_2 - p_0). \]  

(3.14)

It is evident that the pressure at point 1 is not relevant. Hence it can assume any value without affecting the velocity at point 1. If we were now to consider the point 2, in between points 1 and 3, we would find the velocity there to be a function of the pressure at 1 and 3, but not 2. The effect is to separate the pressure into (in one dimension) two separate, entirely independent fields, with every other point belonging to either one or the other. In two dimensions the appearance is rather like a checkerboard (although in fact there are four independent fields now, so in fact things become more complicated), hence this behaviour has popularly come to be known as ‘checkerboard oscillation’.

A simple and effective cure for this problem is to relocate all the velocity nodes onto the cell faces of the pressure cells. No interpolation is then required, since all the pressure nodes will also be located on the cell faces of the velocity cells. This also means that, in more than one dimension, the grids for components of the velocity must all be different. Each must be on whatever face of a pressure cell is normal to the direction of the velocity - so nodes for U must be on cell faces orthogonal to the x-axis, V-nodes must be on faces orthogonal to the y-axis, W-nodes on faces orthogonal to the z-axis. Otherwise the other pressure nodes around the momentum control volume would contribute to the pressure gradient source - since these are not on momentum cell faces, interpolation would once again be required, and the strong coupling between pressures and velocities would be lost.
Figure 3.1 shows this grid arrangement. This method was first developed by Harlow and Welch [41]. It is also sometimes referred to as the Arakawa C-grid.

The disadvantage of the staggered grid (leaving aside the complexity involved in dealing with several different grids) is that the Cartesian velocity components are required to be orthogonal to the cell faces. This is easily achieved on a Cartesian grid, but if a body-fitted curvilinear grid is used, it is no longer possible to satisfy this requirement and simultaneously continue using the Cartesian velocity decomposition. Instead, the decomposition then becomes a function of the curvature of the grid, and the momentum components can no longer be treated as conserved quantities. This manifests itself in the appearance of source terms (Christoffel symbols) in the momentum transport equations.

There are two possible solutions to this problem. The first approach is to use interpolation to evaluate the pressure contribution from momentum CV faces without a pressure node, but to continue using the Cartesian momentum components. This is often a workable strategy when the non-orthogonality is weak [2]. The majority of the previous work in the field here treated has followed this approach (such as, for instance [20]).

However, this approach will always fail if the grid becomes sufficiently curved (see figure 3.2 for an example). Further, the instability inherent in such a discretization may be aggravated by predisposing factors, such as perhaps a lack of diffusivity. The development of the code described in this chapter was motivated by the realization that the k-ε turbulence model could be such an aggravating factor, if the background turbulence levels (and hence the turbulent diffusivity) were low enough. Alternatives such as the mixing length and k-l models were found to be less
susceptible. This reasons for this are rather unclear; however, both of the latter two tend to damp any velocity fluctuations away from a wall very heavily (since the eddy viscosity varies as the square of the distance from the wall\(^3\)). It seems reasonable to suppose that this would render them less vulnerable, since the background diffusivity is not very important near a wall, where velocity gradients (and, therefore, diffusive fluxes) are high anyway. The use of such a model presupposes that turbulent effects are restricted to a boundary layer, so that shear due to the presence of the wall is the only source of turbulence. As noted above, that renders them rather unsuitable for calculations involving wave-breaking. It is therefore desirable to employ a model such as \(k\)-\(\varepsilon\) instead.

![Curved Staggered Grid with Cartesian velocity decomposition](image)

**Figure 3.2:** Curved Staggered Grid with Cartesian velocity decomposition

If the non-orthogonality cannot be ignored, it can instead be dealt with by subsuming it into the momentum transport equations themselves, in the form of a co- or contravariant velocity decomposition [123, 114]. While there is no reason why such an approach cannot be both accurate and efficient, it is not conservative, which makes it awkward to implement in a finite volume method. Particularly in three dimensions, very many source terms are required to ensure accuracy. It is principally on the grounds of difficulty that such a method was not considered.

Since a staggered grid was not considered suitable, special measures are necessary to prevent decoupling of pressures and velocities. The method selected was that of Rhie and Chow [93], which will be described in section 3.5.2 below.

---

\(^3\)If the Blackadar relation 2.146 is used for the mixing length, the latter will asymptote to a limiting value sufficiently far from the surface. This value will be appropriate for the outer (wake) region of the atmospheric boundary layer (and therefore still rather high).
It remains to define the relationship between the central node that represents the control volume and the control volume surface. Two arrangements are in common use, namely the cell-vertex and cell-centered schemes. The latter imposes the constraint that the location of the node should be at the centre of the control volume, while the former instead constrains the cell faces to always lie halfway in between nodes. These two constraints cannot in general be satisfied simultaneously - figure 3.3 below shows both schemes on a non-uniform grid.

The advantage of the cell-vertex scheme is that central differencing using the nodal values is always exact for linear variations. It is therefore easy to calculate fluxes accurately. The disadvantage is that it will be difficult to calculate sources accurately. The reason for this is that the node is not at the cell centre, which is advantageous for the same reason that it is advantageous for cell faces to be centrally located between nodes - linear variations can then be calculated exactly. The average value of a linearly varying variable within the control volume is equal to the value at centre. The cell-centered scheme can take advantage of this.

The relative merits of the two schemes, therefore, usually depend on the relative importance of the terms in the transport equations that are expected to dominate. If diffusive and advective fluxes are considered more significant, it is likely that the cell-vertex scheme will be more accurate. Time-dependent terms and volume source terms favour the cell-centered scheme. Volume sources are not very significant for the momentum equations, so for steady flows, the cell-vertex scheme would appear to be preferable. However, the flows to be considered are unsteady. Moreover, a turbulence model is used; the transport equations for the turbulence quantities involved are dominated by source terms. Finally, on curvilinear grids in particular, it is necessary to transfer parts of the advective and diffusive fluxes to the source term in order to avoid excessively large computational molecules. This can also be used to help stabilize the advection scheme, and to discretize the cross-diffusion terms in the momentum equations (see sections 3.4.2 and 3.4.3 below). While the accuracy of these so-called deferred corrections is unlikely to be affected by the discretization of the volume source terms proper, they do reduce the importance of the accuracy of the implicit fluxes, since these will be corrected anyway.

For these reasons, a cell-centered scheme was selected.
3.3 Calculation of Cell Volumes and Surface Vectors

The surface vector $S$ is required to evaluate the flux terms in 3.1. It is of course important that it be calculated accurately, even on distorted grids, where a cell face may be twisted or curved. Returning to the divergence theorem

$$\int_{\Omega} \nabla \cdot \phi d\Omega = \int_{S} \phi \cdot dS,$$

and letting $\phi = 1$, we obtain an important property of $S$,

$$\int_{S} dS = 0. \tag{3.16}$$

Hence it must be possible to remove any part of the surface with a closed boundary, replace it with another, which may differ from it in any and every way save for the boundary, and still satisfy 3.16. This means it is impossible for the surface vector of a part of the surface to depend on anything other than the boundary between it and the rest of the surface of the control volume. It is not, therefore, necessary to concern ourselves with the geometric properties of the surface of a cell face (such as curvature), but only its boundary.

The grids to be used are structured, using hexahedral control volumes. The cell face surfaces will therefore be quadrilaterals of arbitrary shape. Since their vertices may not be coplanar, we decompose these into triangles, which do have this property. It follows from equation 3.16 that the sum of the surface vectors of two adjoining triangles must equal the surface vector of the quadrilateral surface defined by them.

The surface vector of a triangle is half the cross product of any two sides. Hence, for the quadrilateral $ABCD$ shown in figure 3.4,

$$S_{ABCD} = S_{ABC} + S_{CDA}. \tag{3.17}$$

We may choose any two vectors to define each triangle, but the following simplification is shorter if one of the vectors is shared. We choose the vector $x_{AC}$.

$$S_{ABC} + S_{CDA} = \frac{1}{2} \left( (x_{AC} \times x_{CD}) + (x_{BC} \times -x_{AC}) \right). \tag{3.18}$$

Note that the direction of the vector (outwards or inwards) is determined by the sense (clockwise or anticlockwise) in which we define the vectors. For instance, $x_{AC} \times x_{AB} = -x_{BA} \times x_{AC}$. This follows readily from the anticommutative property of cross products; that is, for any two vectors...
Another property of cross products is that for any vectors $a, b$ and $c$

$$(a + b) \times c = (a \times c) + (b \times c). \quad (3.19)$$

Hence we can simplify equation 3.18 to

$$S_{ABC} + S_{CDA} = \frac{1}{2} \left( (x_{AC} \times x_{CD}) + (x_{AC} \times x_{BC}) \right) \quad (3.20)$$

$$= \frac{1}{2} \left( x_{AC} \times (x_{CD} + x_{BC}) \right). \quad (3.21)$$

Therefore

$$S_{ABCD} = \frac{1}{2} (x_{AC} \times x_{BD}). \quad (3.22)$$

Equation 3.22 is used in preference to 3.18, as it is simpler and equivalent to it. Since we have not made the assumption that the points A, B, C and D are coplanar, this means 3.22 is accurate even for twisted surfaces.

It remains to define the volume $\Omega$ contained within $S$. This is important if volume sources and time-dependent terms are to be computed accurately. The method adopted is that of Kordulla and Vinokur [51].

These workers decompose a hexahedron into six tetrahedra in such a way that the partitioning diagonals on each cell face have the same orientation for each of the two cells they divide (see figure 3.5 above). This ensures that neighbouring cells are contiguous (i.e. that there are no gaps between their cell faces), even when the faces are twisted. The volume of a tetrahedron is equal to one-sixth of the triple product of any three edge vectors that meet at a vertex.
If the decomposition shown in figure 3.5 above is adopted, then each tetrahedron has two edges on the surface of the volume, and one edge in the interior. This edge is shared by all six tetrahedra; if choose this as the vector \( \mathbf{r}_i \), in equation 3.23 above, we obtain

\[
6\Omega = \mathbf{r}_1 \cdot (\mathbf{r}_2 \times \mathbf{r}_3),
\]

(3.23)

where the vectors \( \mathbf{r}_2 \) and \( \mathbf{r}_3 \) are two of the edges, of each tetrahedron, that connect with \( \mathbf{r}_1 \).

Since there are always three such edges, we have some discretion. For a control volume such as the one shown in figure 3.5 above, we choose as follows

\[
6\Omega = \mathbf{r}_1 \cdot \sum_{i=1}^{6} (\mathbf{r}_{2i} \times \mathbf{r}_{3i}),
\]

(3.24)

so that all the vectors connect to \( A \) (we could also choose \( H \) instead). These are all on the surface of the control volume; each pair, therefore, defines a triangle, two of which make up a cell face, just as the triangles in equation 3.18 do. Therefore, we can use 3.22 to replace each pair of triangles with one cell face surface vector, and reduce the number of cross products from six to three. We obtain

\[
6\Omega = \mathbf{r}_{AH} \cdot \left[ (\mathbf{r}_{GA} \times \mathbf{r}_{FA}) + (\mathbf{r}_{FA} \times \mathbf{r}_{EA}) + (\mathbf{r}_{BA} \times \mathbf{r}_{GA}) + (\mathbf{r}_{CA} \times \mathbf{r}_{BA}) + (\mathbf{r}_{DA} \times \mathbf{r}_{CA}) + (\mathbf{r}_{EA} \times \mathbf{r}_{DA}) \right],
\]

(3.25)

(3.26)

Hence the volume can be found by simply taking the dot product of an interior diagonal such as \( \mathbf{r}_{AH} \) with three surface vectors that have a vertex in common with each other and the selected...
diagonal. In this case

\[ 3\Omega = r_{AH} \cdot (S_{ABCD} + S_{AFGB} + S_{ADEF}). \]  

(3.28)

3.4 Discretization of Momentum and Scalar Transport Equations

3.4.1 Time Discretization

The flow of stably stratified fluid over a hill requires time accurate simulation for several reasons. Firstly, as noted above in section 2.3, unsteadiness owing to internal wave motion is possible. It is believed, on the basis of the relevant experimental results [17], that the lee waves in the physical flow are stationary. However, flow regimes do exist that are unsteady [18, 17, 119], and low-frequency pulsations have been observed in simulations similar in some respects to this one [98, 21, 1]. The assumption of steady flow is therefore not made a priori. Secondly, the columnar wave motions upstream of the obstacle will eventually be reflected from the upstream boundary, thus contaminating the solution. The simulation cannot, therefore, be integrated to \( t = \infty \), but only to some finite value of \( t \) (estimated above in section 2.3). This requires that time be treated as a variable.

Limits can be put on the accuracy required, however. In particular, since a closure scheme is used to filter out turbulent motions, these need not be resolved. The permissible time step is not, therefore, a function of the Reynolds number; since unsteadiness owing to internal wave motion is possible, however, it may be a function of the Froude number.

The highest possible wave frequency in a stably stratified flow is \( N \), the Brunt-Väisälä frequency. Provided \( \delta t \) is sufficiently small compared to \( 2\pi/N \), we can be confident the internal wave motions are well resolved. Since \( N \) is not much less than unity, we obtain an oscillation period of several non-dimensional time units. Considering the strongest stratification to be applied, on the coarsest grid, this would correspond roughly to a Courant number of thirty or so, suggesting that a time step sufficient to resolve the oscillation would still exceed a Courant number of unity. Finer grids and weaker stratification would raise this further.

It is desirable that time steps should be as large as would still be consistent with the requirements of accuracy in time. As we have seen, there is no fundamental reason why a Courant number larger than unity would not be acceptable. We should therefore like to use a differencing scheme which is stable and accurate at relatively high Courant numbers.

Forward differencing schemes employ the derivative from a previous time level to obtain the solution at the new time level. Such schemes generally require Courant numbers of rather less than unity to preserve stability. Backward differencing schemes (which calculate the time derivative at the new time level) do not have such limitations, although the Courant number is still subject to
restriction on accuracy grounds (Courant numbers very greatly in excess of unity are hardly ever satisfactory in this regard [77]). A backward differencing scheme is therefore preferred.

The simplest possible implicit time discretization is the backward Euler method, which approximates the time derivative at the current time level $t$ as

$$\left(\frac{\partial \phi}{\partial t}\right)^t = \frac{\phi^t - \phi^{t-1}}{\delta t}.$$  

(3.29)

This is used in an implicit finite volume method based on 3.1 in the following way

$$\frac{1}{\delta t} \int_{\Omega} \phi^t d\Omega + \oint_{\mathcal{S}} \mathbf{F}^t \cdot d\mathbf{S} = \frac{1}{\delta t} \int_{\Omega} \phi^{t-1} d\Omega + \oint_{\mathcal{S}} Q_S^t \cdot d\mathbf{S} + \int_{\Omega} Q_{\Omega}^t d\Omega.$$  

(3.30)

Considering a particular control volume $P$, this may be written

$$A_F \phi^t + \sum_F A_F \phi_F^t = Q_S + Q_{\Omega} + Q_T,$$  

(3.31)

where the $A_F$s denote the flux coefficients (one for each neighbouring control volume), $Q_S$ the surface source term, and $Q_{\Omega}$ the volume source. The central coefficient $A_F$ and the unsteady source term $Q_T$ are

$$A_F = -\sum_F A_F + \frac{\rho}{\delta t},$$  

(3.32)

$$Q_T = \frac{\rho}{\delta t} \phi^{t-1},$$  

(3.33)

where $\rho$ denotes the density. Unlike forward differencing schemes, the backward Euler scheme has no stability limitations. It is, however, only first order accurate. We would like a differencing scheme of second-order accuracy in time. One way to obtain such a scheme is to approximate the value of $\phi$ over the interval $t_0 < t < t_2$ by a parabola, thus:

$$\phi(t) = c_0 + c_1(t - t_0) + c_2(t - t_0)(t - t_1).$$  

(3.34)

By letting $t = t_0, t_1, t_2$ in turn we can obtain $c_1, c_2, c_3$

$$c_0 = \phi(t_0),$$  

(3.35)

$$c_1 = \frac{\phi(t_1) - \phi(t_0)}{t_1 - t_0},$$  

(3.36)

$$c_2 = \frac{\phi(t_2) - \phi(t_1) - \phi(t_1) - \phi(t_0)}{t_2 - t_0}.$$  

(3.37)
Differentiating equation 3.34 with respect to $t$:

$$\frac{\partial \phi(t)}{\partial t} = c_1 + 2c_2 t - c_2 t_0 - c_2 t_1. \quad (3.38)$$

We require a backward differencing scheme. This entails the use of gradients computed at the current time level. Let the current time level be $t = t_2$, and $t_1 = t_2 - \delta t$, $t_0 = t_2 - 2\delta t$ (assuming a constant time step $\delta t$).

$$\frac{\partial \phi(t)}{\partial t} = c_1 + 3c_2 \delta t. \quad (3.40)$$

Substituting equations 3.36, 3.37, for $c_1$ and $c_2$:

$$\frac{\partial \phi(t)}{\partial t} = \frac{\phi(t_1) - \phi(t_0)}{\delta t} + 3\delta t \left( \frac{\phi(t_2) - \phi(t_1)}{2(\delta t)^2} - \frac{\phi(t_1) - \phi(t_0)}{2(\delta t)^2} \right). \quad (3.41)$$

$$\frac{\partial \phi(t)}{\partial t} = \frac{3\phi(t_2) - 4\phi(t_1) + \phi(t_0)}{2\delta t}. \quad (3.42)$$

Using equation 3.1, we once again obtain 3.31, but with $A_P$ and $Q_T$ now given by

$$A_P = -\sum F A_F + \frac{3p}{2\delta t}, \quad (3.43)$$

$$Q_T = \frac{2p}{\delta t} \delta^{t-1} - \frac{p}{2\delta t} \delta^{t-2}. \quad (3.44)$$

### 3.4.2 Discretization of Advection Terms

The advection terms in the Navier-Stokes equations are nonlinear. To linearize them, the advection of a momentum component is treated in the same way as that of a scalar:

$$\frac{\partial (\rho u_i u_j)}{\partial x_j} \Omega \approx \dot{m} \phi_e. \quad (3.45)$$

The mass flux $\dot{m}$ through the cell face is stored separately from the momentum components $u_i$. It is obtained from the latter by interpolation; in order to ensure a smooth pressure solution, Rhie-Chow interpolation is used (see section 3.5.2 below).

Second order central differencing may be written as

$$\dot{m} \phi_e = \frac{\dot{m}}{2} \left( \phi_P(x_E - x_e) + \phi_E(x_e - x_P) \right) \frac{1}{x_E - x_P}, \quad (3.46)$$

78
where \( P \) and \( E \) denote the control volumes considered, and \( e \) the surface common to both. On a uniform grid, this reduces to:
\[
\tilde{m}_e \phi_e = \tilde{m} \frac{\phi_P + \phi_E}{2}.
\] (3.47)

A correction is required when the grid is curvilinear. This is because the line joining the cell centres \( P \) and \( E \) may then not pass through the cell face centre \( e \). Hence the approximation can be in error even for linearly varying \( \phi \). The value obtained from equation 3.47 is therefore treated as provisional. To correct it, the position vector \( x_e - x_e' \) is evaluated, where \( e' \) is an auxiliary node located at the point where equation 3.47 is valid (that is, the point where the cell face intersects the vector joining \( P \) and \( E \)). Equation 3.47 is used to obtain the value of \( (\nabla \phi)_e \) (this requires that \( (\nabla \phi)_P \) and \( (\nabla \phi)_E \) be available; see section 3.8.2 below for details). The value of \( \phi_e \) is then calculated as follows (see also figure 3.6 below):
\[
\phi_e = \phi_e' + (\nabla \phi)_e' \cdot (x_e - x_e').
\] (3.48)

![Figure 3.6: Curvilinear correction to Central Differencing Scheme](image)

Second-order central differencing is accurate and economical. However, for high cell Peclet numbers\(^4\), it is unstable. One way to avoid this is to use first order upwind differencing.

The symbol MAX below refers to the Fortran intrinsic function of that name (see, for instance, table 5 in section 15.10 of [107]).
\[
\tilde{m}_e \phi_e = \text{MAX}(\tilde{m}, 0) \phi_P - \text{MAX}(\tilde{m}, 0) \phi_E.
\] (3.49)

The disadvantages of this formulation are detailed extensively in the literature [54, 33, 77]. To summarize, use of 3.49 leads to considerable numerical diffusion, particularly in multidimensional flows, where artificial diffusion is generated normal to the flow direction [33, p 72]. This can easily overwhelm physical diffusion, which is of course unacceptable.

\(^4\)The Peclet number of a cell is simply the Reynolds number based on the cell width.
However, in order to take advantage of the stabilizing properties of this scheme, it is used to set the implicit advection term in the relevant coefficient $A_F$. A deferred correction is used to replace this with a different advection scheme of higher order, the difference between the two schemes being added to the source term $Q_H$. This may slow down convergence slightly on occasion, but results in much better stability than using the higher order scheme alone, and is just as accurate. In fact, accuracy is easier to obtain, since interpolations may be performed using contributions from as many neighbouring nodes as is desirable. This is very hard to do implicitly.

It remains to choose a suitable scheme of second-order accuracy, and sufficient stability. Second order upwind differencing, which possesses both of these properties, may be written as

$$\hat{m}_f \phi_e = \text{MAX}(\hat{m}, 0) \frac{(\phi_F(x_F - x_W) - \phi_W(x_F - x_H))}{x_F - x_W} - \text{MAX}(-\hat{m}, 0) \frac{(\phi_E(x_E - x_E') - \phi_{EE'}(x_E - x_{EE'}))}{x_E - x_E'}$$

(3.50)

On a uniform grid, this reduces to:

$$\hat{m}_f \phi_e = \text{MAX}(\hat{m}, 0) \frac{3\phi_E - \phi_W}{2} - \text{MAX}(-\hat{m}, 0) \frac{3\phi_E - \phi_{EE'}}{2}$$

(3.51)

The formulation given in equation 3.51 takes proper account of the non-uniformity of the grid. It fails, however, where the grid is non-orthogonal. A simple way of correcting for both these factors is to move whichever node is furthest upstream (either $E$ or $W$ above). This is done by constructing an auxiliary node (shown as $EE'$ in figure 3.7 below), and obtaining the value of $\phi$ there by interpolation. This is performed as follows:

$$\phi_{EE'} = \phi_{EE} + (\nabla \phi)_{EE'} \cdot (x_{EE'} - x_{EE})$$

(3.52)

where $\nabla \phi$ is calculated and stored as described in section 3.8.2. The position of $EE'$ is chosen so that the vector $x_{EE'} - x_e$ is three times the vector $x_E - x_e$. The simple equation 3.51 is then accurate, whether the grid is curvilinear, non-uniform, or both.

Since both the central and second-order upwind approximations are exact for linear variations of $\phi$, so is a weighted mean of the two. The harmonic mean is used, as first proposed by van Leer [115](see also [54]). This is given by

$$\hat{\phi} = \frac{\hat{m}_F - \phi_{EE}}{\phi_F - \phi_{EE}} \quad \text{IF} \quad \hat{m} < 0$$

$$\frac{\hat{m}_E - \phi_W}{\phi_{EE'} - \phi_W} \quad \hat{m} > 0$$

(3.53)
Figure 3.7: Curvilinear correction to Second-order Upwind Differencing Scheme

Here $\phi_{\text{CDS}}$ and $\phi_{\text{SOU}}$ denote the values of $\phi_\epsilon$ obtained by central differencing (3.46), and second order upwind differencing (3.51) respectively. This formulation was used to obtain almost all the results given below (with the exception of some internal flows at low Reynolds number, for which central differencing is stable).

3.4.3 Discretization of Diffusion Term

The shear stress $\tau_{ij}$ is given as follows. For the $ith$ component of momentum

$$\tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right),$$

(3.55)

$$\frac{\partial \tau_{ij}}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} \right) + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_j}{\partial x_i} \right).$$

(3.56)

The first term on the RHS is of the form of a scalar (Fickian) diffusion. The second term on the RHS is zero if the viscosity $\mu$ is constant. If a variable eddy viscosity is used to model turbulence, this is not the case, and it has therefore been retained. When such turbulence models are employed, it is often the case that the gradients of the eddy viscosity are high. Moreover,
since the production term in the turbulence kinetic energy is equal to the energy lost to the mean flow through turbulent diffusion, it is desirable for these two processes to be treated identically so as to conserve total kinetic energy.

Scalar diffusion (term 1 on the RHS of equation 3.56) is discretized as follows:

\[
\frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} \right) \Omega \approx \int_S \mu (\nabla u_i) \cdot \hat{S} \, dS. \tag{3.57}
\]

This requires the derivatives normal to cell face surfaces. These are very difficult to obtain implicitly, unless cell faces are constrained to always be orthogonal to the vector joining cell centres. This would be the case for an orthogonal grid. Such grids can be generated for curvilinear cases, but this is often difficult, particularly if the topography is unsmooth. There is also the additional problem of the vector joining the cell centres not passing through the centre of the cell face. For second-order accuracy, the diffusive flux must be evaluated there (see section 3.1 above). This cannot easily be done implicitly.

Therefore, the term is split up as follows:

\[
(\nabla u_i) \cdot \hat{S} = \frac{\partial u_i}{\partial r} + \left( \frac{\partial u_i}{\partial S} - \frac{\partial u_i}{\partial r} \right). \tag{3.58}
\]

Here \( r \) is the vector joining the cell centres, and \( S \) the surface vector. The first term on the RHS is treated implicitly. The second term (in brackets) is simply the difference between the surface normal derivative and the implicit approximation. This is added to the source term as a correction.

The required surface normal derivative is calculated using the method of Muzaferija (see [33, p 220], citing [75]). This is probably best explained graphically:

\[\text{Figure 3.8: Curvilinear correction to diffusive fluxes}\]
To obtain $\frac{\partial u_i}{\partial S}$ at the cell face $e$ (between cell centres $P$ and $E$), $u_i$ is evaluated at two auxiliary nodes, $P'$ and $E'$, by interpolation. This requires that accurate derivatives are available (see 3.8.2 below for details of the calculation of these). These auxiliary nodes are positioned in line with and equidistant from the cell face centre, and normal to the cell face surface. The derivative is then evaluated using central differencing. This approach ensures that the diffusive flux is correctly evaluated using the surface normal derivative, and also that it is evaluated at the centre of the cell face. Grid non-uniformity is also automatically allowed for.

If the viscosity is not constant in space, it must be obtained at the cell face centre by interpolation. For the case of a variable viscosity, the second term on the RHS of equation 3.56 must also be included. This is treated explicitly, as it contains contributions from all three momentum components, which is difficult to deal with implicitly in a sequential solution method.

$$\frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} \right) \Omega \simeq \int_S \left( \mu \frac{\partial u_j}{\partial x_i} \right) \cdot \hat{S} \, dS. \quad (3.59)$$

### 3.4.4 Buoyancy

The Boussinesq approximation is used to model buoyancy. Hence the density is treated as a constant in the flux terms, which greatly simplifies solution. Effects of variable density are restricted to the buoyancy source term in the momentum equations. For a momentum component $U_i$ in coordinate direction $x_i$

$$Q_B = (\rho - \rho_0)g \cdot i = -\rho_0 \beta (\theta - \theta_0)g \cdot i. \quad (3.60)$$

where $\theta$ is either temperature or species concentration, depending on which determines the density variations giving rise to buoyancy. The constant $\beta$ is the coefficient relating the two, determined by the relevant equation of state.

The value of $\theta$ is determined from a transport equation. Advection and diffusion terms are treated as above. The diffusivity for scalar transport is determined from the Reynolds analogy; that is, it is related to the diffusivity for momentum by a Prandtl or Schmidt number. A different value is employed for laminar and turbulent diffusion with the latter given either by a constant near unity or by equation 2.197 above. There are no source terms in the scalar transport equation as modelled here.
3.4.5 Discretization of the Pressure Source Term

The pressure term in the momentum equations may be treated as a surface force that can be integrated over the faces of the control volume. Thus the contribution to the equation for \( u_i \) would be:

\[
\int \mathbf{p}_i \cdot \mathbf{n} \, ds = \sum C \mathbf{S}_i p.
\]  

The alternative would be to treat the pressure as a body force. This would entail evaluation of the pressure derivative; for the method to be conservative, this would have to be done in such a way that the resulting source would always be equivalent to the one calculated in the manner detailed above. This is so if the derivative is calculated using Gauss's divergence theorem \[33\] 5. However, there is not very much to be gained by doing so, and the first approach has been adopted instead. The derivative may then be calculated without relying on the divergence theorem 6.

Other body forces (Coriolis forces, for instance) are in fact implemented using the second approach. If these vary in space, the method may not be conservative. However, most of the results to be presented include no body forces of this kind.

Note that the pressure is everywhere employed with the hydrostatic component subtracted. Under the Boussinesq approximation, this quantity is guaranteed to sum to zero over the faces of any cell volume, and thus can have no effect other than to add to the truncation error (which is quite likely, since it is in general large compared to the dynamic pressure). It is therefore better ignored; if the total pressure should be required, it can still be calculated easily.

3.5 Calculation of the Pressure

\[
\frac{\partial U_i}{\partial t} + \nabla \cdot (U U_i) = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \nu \nabla^2 U_i, \quad (3.62)
\]

\[
\nabla \cdot U = 0. \quad (3.63)
\]

5The divergence of a quantity (here the pressure) is then assumed equal to its sum over the surface divided by the volume. It follows that \( \frac{\partial P}{\partial x_i} = S_i p \) (thus satisfying equation 3.61), as the inverse of this is used to calculate \( \frac{\partial P}{\partial x_i} \) in the first place.

6The disadvantage of using the divergence theorem to calculate gradients is that we then require the values of \( \phi \) at cell faces. To obtain these accurately on curvilinear grids generally requires either very extensive and costly interpolation, multiple iterations of the derivative calculation, or storage of the derivatives over outer iterations, in which case they approach their correct values as these proceed. This is much less costly than the alternatives, but the storage requirements are high, since in three dimensions, three derivatives must be stored for every variable.
If 3.62 is used to determine the transport of momentum, that leaves the continuity equation 3.63 to determine the pressure. However, in an incompressible flow, the pressure does not appear in 3.63, which merely imposes a constraint on the velocity field. If the continuity equation is to determine the pressure, then it must do so indirectly, through the effect of the pressure on the velocities. That effect must be such as to ensure the satisfaction of continuity. Hence, we need to combine the momentum equations with the continuity equation, which can be done as follows:

The divergence of the momentum equations is taken

\[
\frac{\partial}{\partial x_i} \left( \frac{\partial U_i}{\partial t} \right) + \left( \frac{\partial}{\partial x_i} U \right) \cdot \nabla U_i + U \cdot \left( \frac{\partial}{\partial x_i} \nabla U_i \right) = -\frac{1}{\rho} \frac{\partial^2 P}{\partial x_i^2} + \nu \frac{\partial}{\partial x_i} (\nabla^2 U_i).
\] (3.64)

Summing the three equations yields

\[
\frac{\partial}{\partial t} (\nabla \cdot U) + \frac{\partial U_i}{\partial x_i} \left( \frac{\partial U_i}{\partial x_i} + \frac{\partial U_j}{\partial x_j} \right) + U_i \frac{\partial}{\partial x_i} (\nabla \cdot U) = -\frac{1}{\rho} \frac{\partial^2 P}{\partial x_i^2} \nabla^2 \cdot (\nabla \cdot U) i.
\] (3.65)

By continuity 3.63, \(\nabla \cdot U = 0\), so the first, third, and last terms in the above equation are zero. The following remains:

\[
\frac{1}{\rho} \frac{\partial^2 P}{\partial x_i^2} = -\frac{\partial U_i}{\partial x_i} \left( \frac{\partial U_i}{\partial x_i} + \frac{\partial U_j}{\partial x_j} \right),
\] (3.66)

\[
\frac{\partial^2 P}{\partial x_i^2} = \nabla^2 P = -\rho \frac{\partial}{\partial x_i} \left( \frac{\partial (U_i U_j)}{\partial x_j} \right).
\] (3.67)

The RHS of this equation is the sum of the derivatives of the advection terms in the momentum equations. Since the only term in \(P\) is the Laplacian on the LHS, this is a Poisson-like equation; an equation of this kind must generally be solved to obtain the pressure, although many different such schemes have been devised, all differing in other details.

Note that the Laplacian is the product of two derivative operators in different equations (the continuity equation and the momentum equations). It is important that these be consistent with each other.

### 3.5.1 The SIMPLE Method

The SIMPLE method of Caretto et al [15] and Patankar and Spalding [85], instead of employing an equation for the pressure, iteratively calculates successive corrections to it such that

\[
p^m = p^{m-1} + p'.
\] (3.68)
Here \( m \) denotes the iteration level. This is the method employed here, so it will be described below. For reasons that will become obvious later, we begin with the discretized momentum equations

The discretized equations of momentum conservation are nonlinear. They are therefore solved iteratively - nonlinear terms are linearized before solution, recalculated afterwards, and the process repeated until the equation is satisfied. At any iteration level \( m \), a momentum component \( U_i \) is a function of the pressure at the same \( m \):

\[
U_i^m = f(U_i^m) - g(p^m). \tag{3.69}
\]

Here both \( f \) and \( g \) are linear; the latter is the (properly linear) pressure gradient term, the former is the linearized momentum flux.

For a particular control volume \( P \), the discretized equation becomes

\[
\sum_F A_F^{U_i} u_i^{m_F} + \sum_F A_F^{U_i} u_i^{m_F} = Q_{U_i}^m - \left( \frac{\partial p^m}{\partial x_i} \right)_P \tag{3.70}
\]

where the \( A_F \)s denote the flux coefficients (one for each neighbouring control volume), \( A_p \) the central coefficient, and \( Q \) the source term. The \( A_F \)s are fixed within each iteration \( m \) (but are strictly functions of \( U_i \)). Since the pressure \( p^m \) is unknown, we cannot solve this equation. However, we can regard the pressure at \( m - 1 \) as an approximation to \( p^m \), with error \( p' \)

\[
p^m = p^{m-1} + p', \tag{3.71}
\]

so that

\[
\sum_F A_F^{U_i} u_i^{m_F} + \sum_F A_F^{U_i} u_i^{m_F} = Q_{U_i}^m - \left( \frac{\partial p^{m-1}}{\partial x_i} \right)_P - \left( \frac{\partial p'}{\partial x_i} \right)_P \tag{3.72}
\]

If we neglect \( p' \), this equation is now soluble. We denote this intermediate estimate of \( U_i \) as \( U_i^{m*} \)

\[
\sum_F A_F^{U_i} u_i^{m_F} + \sum_F A_F^{U_i} u_i^{m_F} = Q_{U_i}^{m*} - \left( \frac{\partial p^{m-1}}{\partial x_i} \right)_P \tag{3.73}
\]

Obviously, since the pressure was taken from a previous iteration level, this is not correct. We denote the error by \( U_i' \), so that

\[
U_i^m = U_i^{m*} + U_i', \tag{3.74}
\]

and

\[
\sum_F A_F^{U_i} u_i^{m_F} + \sum_F A_F^{U_i} u_i^{m_F} + \sum_F A_F^{U_i} u_i^{m_F} + \sum_F A_F^{U_i} u_i^{m_F} = Q_{U_i}^{m*} - \left( \frac{\partial p^{m-1}}{\partial x_i} \right)_P - \left( \frac{\partial p'}{\partial x_i} \right)_P \tag{3.75}
\]

86
since the all the terms have been linearized. It is now evident why it is necessary to proceed from the discrete equations, instead of the continuum form. It is necessary to use the linearized form of the momentum equation, or it would be impossible to separate $U'_i$ from $U_i$, since the $A_{i}$ would be functions of $U_i$. Discretization precedes linearization, so the latter implies the former. The equation for the pressure correction that follows from the above is not really a transport equation in its own right, since it cannot be separated properly from the numerical method. However, this is not as significant as it might appear, since the pressure is not a conserved variable anyway.

Subtracting 3.73 from 3.75, we obtain an equation relating $U'$ to $p'$

$$A^{U'U'}_{i} U'_{i} + \sum_{F} A^{U'U'}_{F} U'_{i,F} = \left( \frac{\partial p'}{\partial x_{i}} \right)_{p} .$$

We can obtain the $U'_i$ by invoking continuity:

$$\nabla \cdot U^{m} = 0,$$

$$\nabla \cdot U^{m*} + \nabla \cdot U' = 0.$$  

An equation for $p'$ in terms of known quantities is sought. We can solve 3.73 to find the three components of $U^{m*}$, so equations 3.76 and 3.78, are sufficient to determine the unknowns $U'$ and $p'$. However, it is much easier to neglect $\sum_{F} A^{U'U'}_{i,F}$. This is the least defensible approximation made in the SIMPLE method, and many variants (SIMPLER and SIMPLEC, for instance) differ from it in this regard. If the SIMPLE approximation is made, however, 3.76 becomes

$$U'_{i,F} = \frac{1}{A^{U'}_{i,F}} \left( \frac{\partial p'}{\partial x_{i}} \right)_{p},$$

which, when substituted into 3.78, yields

$$\frac{\partial}{\partial x_{i}} \left( \frac{1}{A^{U'}_{i,F}} \left( \frac{\partial p'}{\partial x_{i}} \right)_{p} \right) = - \nabla \cdot U^{m*}.$$

If, and only if, both the first and second of the operators $\partial/\partial x_{i}$ on the LHS are discretized identically, this can be written as a Poisson-like equation:

$$\frac{1}{A^{U'}_{i,F}} \frac{\partial^{2} p'}{\partial x_{i}^{2}} = - \nabla \cdot U^{m*},$$

$$\frac{1}{A^{U'}_{i,F}} \nabla^{2} p' = - \nabla \cdot U^{m*}.$$

The first $\partial/\partial x_{i}$ is from equation 3.77; by equation 3.78 it has to be the same as the $\nabla$ on the RHS of 3.80. The second $\partial/\partial x_{i}$ comes from 3.70, and represents the discrete form of the pressure term.
in the momentum equations. Hence we can only write 3.80 in the form 3.82 if the divergence of the velocity field is evaluated in the same way as the pressure source (described in section 3.4.5 above). If these two quantities were evaluated in different ways, the LHS of 3.82 would have to be modified to remain consistent.

Once equation 3.82 is solved for $p'$, $U'$ can be found from 3.79. The pressure $p$ and fluxes $U$ are then corrected by adding these corrections to them; usually a degree of under-relaxation is necessary. The procedure is then repeated, until satisfactory convergence (defined in section 3.8.1 below) is obtained.

### 3.5.2 Rhie-Chow Interpolation

The SIMPLE method, as given above, works without modification on staggered grids. On collocated grids, additional measures are necessary to prevent the 'checkerboard oscillation' problem discussed above in section 3.2. One effective method is that due to Rhie and Chow [93], which will be described below.

As noted above, the root cause of the splitting of the pressure field into multiple independent systems, and the subsequent pressure oscillations, is the use of interpolation to obtain the pressure at cell faces. This effectively means that the pressure gradient is calculated on a grid half as dense (in each direction) as the one the transport equations are discretized on. The staggered grid cures this by moving the velocity nodes so that the pressure gradient for each momentum control volume can be calculated using adjacent pressure nodes. This suggests that we can solve the problem on a collocated grid in a similar way.

The pressure gradient at the momentum cell centres cannot be easily be made sensitive to the value of the pressure there. However, as we have seen, the pressure gradient at the cell face is. On a collocated grid, the momentum components are not stored on cell faces, but are still required there (to calculate the advection terms, for instance). This suggests the solution of modifying the cell face fluxes. Since these appear as source terms in the pressure correction equation, they can be used to influence the pressure field. It is of course necessary that the modification should approach zero as the pressure gradient term appearing in the momentum equations approaches the pressure gradient calculated using immediately adjacent nodes.

Consider the flux through a cell face $e$, with surface vector $S$, located in between two nodes $P$ and $E$. We wish to interpolate between two values of $U \cdot S$ (located at $P$ and $E$) substituting the pressure gradient at the location interpolated to for that used to calculate the values of $U$ at the cell centres (between which interpolation is performed). The pressure gradient at the cell face can be calculated using the pressures located at the cell centres:

$$\frac{\partial p}{\partial r} = \frac{p_E - p_P}{r}.\quad (3.83)$$
where \( r = (x_{i,E} - x_{i,P})i \). We assume that this is parallel to \( S \). If this is not the case, convergence of the resulting method may be slow, but accuracy will not be affected (see 3.86 below). The contribution of the pressure gradient to the \( r \) component of \( U \) is, by equation 3.70 above,

\[
U_r \sim -\frac{1}{A_P} \frac{\partial p}{\partial r}.
\]  

(3.84)

We subtract the mean of this from the interpolated mass flux

\[
\bar{m} = \rho U \cdot S - \rho |S| \left( -\frac{1}{A_P} \left( \frac{\partial p}{\partial r} \right) \right),
\]

(3.85)

and add the pressure gradient from 3.83 instead.

\[
\bar{m} = \rho U \cdot S - \rho |S| \left( \frac{p_E - p_P}{r} - \frac{\partial p}{\partial r} \right).
\]

(3.86)

This satisfies the requirement stated above that the correction should become zero as the pressure gradients \((p_E - p_P)/r\) and \(\partial p/\partial r\) tend to equality. If they do not, the mass flux across the cell face is coupled to the pressure gradient normal to it, so that, for instance, \((p_E - p_P)/r > \partial p/\partial r\), the effect is to drive down the mass flux. The flux out of \( P \) and into \( E \) is therefore reduced, which decreases \( \nabla \cdot U^m \) in the pressure correction equation 3.80 above. Since this is a negative source term in that equation, the effect should be to increase the pressure at \( P \), and conversely, decrease it at \( E \), so bringing \((p_E - p_P)/r\) and \(\partial p/\partial r\) nearer equality. The correction is therefore also in the correct sense.

When applied to every cell face in the domain, the effect of this interpolation on the pressure field can be shown to be similar to a smoothing operation on the third derivatives of the pressure [33, p 188]. Oscillatory solutions are therefore filtered out of the pressure field.

The above stated requirement that \( r \) be roughly parallel to \( S \) at first glance appears rather similar to the need for the cell faces on a staggered grid to be orthogonal to Cartesian velocity components. It is, however, much less stringent, because \( r \) and \( S \) are local vectors, not global. Any mismatch can be ameliorated by refining the grid, by which means it is always possible to make these vectors as nearly parallel as required. Grid refinement has no effect, however, if one of the vectors is globally defined, as is the case for the staggered grid with Cartesian velocity components.

\footnote{Note that \( A_P \) is the same for all three velocity components, which simplifies matters.}

\footnote{This was arrived at by applying the finite difference method, using central differencing, on a two-dimensional uniform grid.}

\footnote{In fact, grid refinement is not always needed, as more nearly orthogonal grids can usually be created if necessary. Even then, however, the grid generation process may be involved.}
3.6 Boundary Conditions

3.6.1 Pressure Boundary Conditions

The mass flux through boundaries is set by the momentum conservation equation, and treated as fixed in the pressure correction step. This implies that $U'$ (the correction to the velocity normal to the boundary) is zero, and that therefore the gradient of $p'$ normal to the boundary must be zero also.

Zero gradient boundary conditions are, therefore, applied on all boundaries on which the mass flux is not corrected. All boundary conditions described in this section below are of this type. The mass fluxes through inlet, symmetry, and wall boundaries are always fixed and cannot not be corrected. For outlet and convective boundary conditions this is less obvious, since the mass flux through these is not fixed. However, it is more convenient to set it by extrapolating the interior solution (in whatever way is appropriate for the boundary condition in question) and then correcting it to satisfy global mass conservation (see section 3.6 below for details). The flux is then treated as fixed for the purpose of calculating pressure correction.

If zero gradient boundary conditions are imposed on all boundaries, the pressure is no longer uniquely determined, since only gradients of the pressure (rather than its integral value) ever appear either in its own equation or in the momentum equations. In order to make the pressure unique, it must be fixed at at least one point in the computational domain. The point chosen is the centre; all pressure corrections are applied relative to the pressure there (which is chosen to be zero).

3.6.2 Inlet Boundary Conditions

At the inflow boundary, all variables are treated as known, except the pressure (see above). Dirichlet conditions are therefore applied.

3.6.3 Wall Boundary Conditions

No-slip (Dirichlet) boundary conditions are applied on all wall boundaries. In addition, the normal viscous and turbulent stresses are set to zero. This follows from continuity:

$$\frac{\partial u_t}{\partial x_t} + \frac{\partial u_n}{\partial x_n} = 0,$$

(3.87)
where \( x_t \) and \( x_n \) are the directions normal and tangential to the boundary, and \( u_t \) and \( u_n \) the velocity components in those directions. Provided \( u_t \) on the wall itself does not vary with \( x_t \), then

\[
\frac{\partial u_t}{\partial x_t} = 0 = \frac{\partial u_n}{\partial x_n}.
\]  

(3.88)

Hence \( \partial u_n / \partial x_n \) is zero also, and so is the normal stress

\[
\tau_{nn} = 2\nu \frac{\partial u_n}{\partial x_n} = 0.
\]  

(3.89)

This is implemented by modifying the derivatives of \( u \) on the boundary in the appropriate way. The source term is then modified in the same way as for the interior cell faces; the correct stresses are calculated explicitly and the difference between them and implicitly calculated ones added to the source term.

Several wall boundary conditions are available for the density-determining scalar. Adiabatic walls require zero flux through the boundary. This is implemented simply by setting the coefficient linking the boundary to the interior cell to zero. Isothermal walls correspond to Dirichlet boundary conditions, also simple to implement. These were used only for the free-convection test cases described below; otherwise, adiabatic boundaries were always applied.

### 3.6.4 Symmetry Boundary Conditions

Symmetry boundary conditions require that there be no flow across the boundary. This means the velocity must become tangential to the boundary as the latter is approached. Hence the normal velocity component must become zero. This implies nonzero derivatives of the normal velocity component in the normal direction, and therefore also a nonzero normal stress. This is the only stress consistent with both symmetry and the law of conservation of momentum - it implies an equal and opposite stress on the opposite side of the symmetry plane. Any shear stress will imply (by symmetry) an equal stress in the same direction, which violates conservation of momentum. Therefore, all other derivatives of momentum normal to the plane are zero. This condition obviously extends to scalars also - since these do not possess a direction, symmetry implies, as for tangential momentum, that the net flux through the boundary must be zero.

The boundary condition on scalars is straightforward. The coefficient representing the flux through the boundary is set to zero. For momentum, the situation is more complicated. Since no individual velocity component may be exactly perpendicular to the boundary, the velocities there are modified as follows:

\[
u_i - \frac{S_i u \cdot S}{S^2},
\]  

(3.90)

where \( u \), and \( u_i \) are the values of velocity, and a component thereof, at the cell centre. The result
is applied to the cell face. The tangential velocity is preserved, but the normal component is eliminated.

3.6.5 Outflow Boundary Conditions

Although these were not used in the calculations to be presented, they will be described here, since they have much in common with the convective boundary conditions described below, and also because they were used to test the rest of the code.

In steady flows, and sufficiently far downstream of the region of interest, it is usually safe to neglect all spatial derivatives in the streamwise direction. A zero gradient boundary condition can then be applied for all variables on a boundary normal to the flow. It is also possible to use less restrictive conditions (e.g. constant momentum gradient normal to the boundary, or extrapolation on streamlines), but these do not usually present significant advantages, since, in general, no such condition is safe to use when zero-gradient is not. The exception might be if there were some good reason for orienting the boundary in some way other than perpendicular to the flow, but this is rarely a problem.

This boundary condition is implemented by extrapolating all variables to the outlet under the assumption that the normal gradient is zero. The result is that, at convergence, there is zero diffusive flux, but not zero advection. Care must be taken, therefore, that the total mass flux out of the outlet boundary balances the incoming flow, or mass conservation may be violated. Since the pressure correction procedure assumes the mass flux to be given as a boundary condition, it cannot be used to correct the boundary mass flux so as to ensure mass conservation. This must instead be determined some other way. The method chosen is to evaluate the total mass fluxes at both inlet and outlet boundaries, and to scale the latter so as to ensure agreement.

3.6.6 Advective Boundary Conditions

Zero gradient extrapolation is less satisfactory in unsteady flows. Instead, an advective boundary condition is used:

\[
\frac{\partial \phi}{\partial t} + c \frac{\partial \phi}{\partial n} = 0,
\]

(3.91)

where the value of \(c\) is the same for all locations on the outflow surface. It is set equal to the anticipated speed of internal waves in the flow considered.

Equation 3.91 is a linear convection equation with convective velocity \(c\). Alternatively, \(c\) can be
interpreted as a wave speed.

Orlanski [78] proposes calculating $c$ separately for each point on the boundary, and also for each flow variable. A procedure of this type is also adopted by Clark [20]. However, such an approach is somewhat involved, and potentially subject to numerical instability. It has not been adopted here.

Instead, a fixed value is chosen for $c$. The time derivative is evaluated using backward differencing, which then permits calculation of the gradient normal to the boundary at the current time step. The time derivative is extrapolated from the interior,

$$\frac{\partial \phi_0}{\partial t} = \frac{\partial \phi_P}{\partial t}, \quad (3.92)$$

where it can be evaluated using the same three-time-level backward difference approximation upon which the time discretization itself is based (see equation 3.42 above). However, although some results were successfully obtained using this scheme, it was found to be too unstable in general (even when its application was heavily under-relaxed). Most results were therefore obtained by using backward Euler differencing.

Additionally, global mass conservation is enforced in the same way as for the zero gradient boundary condition.

Although this simple procedure is not as general as Orlanski's more involved approach, it is reasoned to be adequate for the problem considered. Since the simulations to be carried out are only integrated to finite time, it suffices to deal only with the faster modes. The reflection of the fastest mode from the upstream boundary cannot be prevented in the way described, since the values of all flow variables (save pressure) are fixed there. The time to which the simulation may be integrated is therefore in any case limited by the distance between the obstacle and the upstream boundary. Setting $c$ equal to the speed of the fastest mode permits the distance from the obstacle to the downstream boundary to be half of that to the upstream boundary, since the next fastest mode (which might be reflected to some extent) travels at roughly half this speed (see figure 2.9 above).

Note that it is possible to regard the zero gradient boundary condition as identical to equation 3.91 in the limit $c \to \infty$. It is therefore quite appropriate for flows where only waves of infinite speed are expected; this is true for most incompressible flows that do not support gravity or Rossby waves, since the speed of sound is infinite in perfectly incompressible media.

Because of the use of short time steps and heavily stretched grids in the simulations described below, it actually takes a number of time steps for the front of a columnar mode to pass through
the cell adjoining the boundary. Use of $c = \infty$ would shorten this to one time step, regardless of the physical propagation speed.

It is therefore felt that it would be more appropriate to use the velocity of the fastest wave mode for $c$. As already discussed in section 2.3 above, in this flow, waves of zero wavenumber are anticipated. These are the fastest propagating, and it would therefore make sense to tune the boundary condition to transmit these as well as possible. For modes with finite wavenumber, it makes sense to simply choose the phase velocity for $c$. The group velocity determines when disturbances due to the mode first arrive at the boundary, and thus might matter at that point in time, but the disturbances themselves are characterized by their phase velocity. Columnar modes, however, are of constant phase, so no disturbance exists except at the leading edge, which moves at the group velocity.

The group and phase velocities of a wave are related by

$$c_g = \frac{\partial \omega}{\partial k},$$

(3.93)

where $c_g$ is the group velocity, $k$ the wavenumber and $\omega$, the frequency, equals $kc$. Hence

$$c_g = c + k^2 c,$$

(3.94)

from which we may conclude that as $k \to 0$, the group and phase velocities tend to equality. Which we choose is therefore irrelevant.

Some simulations were in fact carried out using the zero gradient outlet. Comparisons with results obtained using an advective boundary condition could therefore be made.

### 3.7 Turbulence Modelling

The effects of turbulence on momentum and scalar transport are mediated by the eddy viscosity $\nu_t$ and the turbulent Prandtl or (for mass transport) Schmidt number $\sigma_t$. These are related to the turbulence variables $k$ and $\epsilon$ by

$$\nu_t = f_\nu C\nu \frac{k^2}{\epsilon},$$

(3.95)

$$\sigma_t = f_\sigma \sigma_{10},$$

(3.96)

where $C\nu$ and $\sigma_{10}$ are constants appropriate for neutral flow. In the standard $k-\epsilon$ model, $f_\nu$ and $f\sigma$ are unity.
3.7.1 Transport Equations

The turbulence scalars $k$ and $\epsilon$ are subject to the following equations of transport (see section 2.6 above).

\begin{align*}
\frac{\partial k}{\partial t} + \frac{\partial \rho U_i k}{\partial x_i} &= P_k - \rho \epsilon + \frac{\partial}{\partial x_i} \left( \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_i} \right), \quad (3.97) \\
\frac{\partial \epsilon}{\partial t} + \frac{\partial \rho U_i \epsilon}{\partial x_i} &= C_{\epsilon1} P_k \frac{\epsilon}{k} - \rho C_{\epsilon2} \frac{\epsilon^2}{k} + \frac{\partial}{\partial x_i} \left( \frac{\mu_t}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_i} \right), \quad (3.98)
\end{align*}

where $P_k$ is the production of turbulence kinetic energy, which is equal to the rate at which energy is lost to the mean flow through the action of turbulent diffusion due to $\nu_t$. It is equal to

$$P_k = \rho \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j}.$$  \hspace{1cm} (3.99)

The first term on the LHS of equations 3.97 and 3.98 above is the time rate of change of $k$ and $\epsilon$; the second term represents advection, and the last term on the RHS is, for both equations, diffusion. All of these are discretized in the same way as the equivalent terms in the scalar transport equation. Turbulent Prandtl/Schmidt numbers are used to modify diffusion.

3.7.2 Source Terms

The source terms of the $k$ and $\epsilon$ equations are highly nonlinear, and must be linearized prior to solution. The production terms of both equations are added to the right hand side $Q_\Omega$, and the dissipation terms to the central coefficient $A_P$.

$$Q_\Omega \quad P_k \quad C_{\epsilon1} P_k \frac{\epsilon}{k}$$

$$A_P \quad \frac{\epsilon \nu_t}{k} \quad \rho C_{\epsilon2} \frac{\epsilon^2}{k}$$

Thus, instead of the source and sink terms both being added to the source $Q_\Omega$, the sink terms are instead moved to the $A_P$ term. This makes no difference as long as $k$ and $\epsilon$ are positive; if they become negative, however, the sink term in $A_P$ becomes a source. This procedure helps to prevent $k$ and $\epsilon$ temporarily assuming negative values, which, apart from being unrealizable physically, tends to cause numerical difficulties (particularly in the case of $\epsilon$, since this then results in negative $\nu_t$). Additional measures are taken to ensure that $k$ and $\epsilon$ remain positive. The terms $A_P$ and $Q_\Omega$ are modified in the following way when the $Q_\Omega$ term becomes negative [84]

$$A_P \rightarrow A_P + Q_\Omega \phi, \quad (3.100)$$
$$Q_\Omega \rightarrow 0. \quad (3.101)$$

95
3.7.3 Wall Boundary Condition

It is assumed that the cell adjoining a wall boundary is submerged in the layer in which the logarithmic law of the wall holds

\[ U = \frac{u_r}{\kappa} \ln (E^+ n^+) \]  

(3.102)

where \( \kappa \) is the Kármán constant, \( u_r \) the friction velocity, \( E^+ \) is a constant (which depends on the surface roughness), and \( n^+ \) is a nondimensional height above the surface. \( u_r \) and \( n^+ \) are given by

\[ u_r = \sqrt{\frac{\tau}{\rho}} \]  

(3.103)

\[ n^+ = \frac{n u_r}{\nu} \]  

(3.104)

where \( n \) is the distance from the wall.

Differentiating 3.102 with respect to \( n \), we obtain

\[ \frac{\partial U}{\partial n} = \frac{u_r}{\kappa n} \]  

(3.105)

Since

\[ u_r^2 = \frac{\tau}{\rho} = \nu \frac{\partial U}{\partial n} \]  

(3.106)

we can express the near-wall viscosity as

\[ \nu_\tau = u_r \kappa n. \]  

(3.107)

This is implemented in the momentum equations by modifying the eddy viscosity on the wall boundary cell face. However, equation 3.107 is not appropriate for this, because the tangential velocity gradient in the discretized momentum equations is given by

\[ \frac{\partial U}{\partial n} = \frac{U_P - U_0}{n}. \]  

(3.108)

Equation 3.108 is not in agreement with the gradient from 3.105, since it assumes a linear velocity profile instead of a logarithmic one. Worse, the wall law ceases to apply very close to the surface, so there is a minimum value of \( n \) below which it is not accurate. It is not possible, therefore, to bring 3.105 and 3.108 into better agreement by grid refinement. A more suitable value for \( \nu_\tau \) can be found by substituting 3.108 into 3.106, so that

\[ \tau = \nu_\tau \frac{U_P - U_0}{n}. \]  

(3.109)
Therefore
\[
\nu_t = u_t^2 \frac{n}{U_P - U_0}.
\]  
(3.110)

We require that \( U_P - U_0 \) be in agreement with equation 2.177; we then obtain
\[
\nu_t = u_t n \epsilon \frac{1}{\ln(E^n n^+)}.
\]  
(3.111)

One disadvantage of the definition of \( u_r \) given in 3.103 is that it becomes zero at a stagnation point, such as a separation or reattachment line near a recirculation zone in the flow (as the local shear stress \( \tau \) then vanishes). The wall coordinate \( n^+ \) is then zero for any \( n \), so equation 2.177 breaks down. Recirculation zones are expected in the flows to be investigated, so an alternative definition was sought. One can be obtained by considering the \( k \) and \( \epsilon \) equations. Since equation 2.177 implies that \( U^+ \) is a function of \( n \) only\(^{10} \), equations 3.97 and 3.98 become
\[
0 = \left( \frac{\partial}{\partial x_1} \right) \left( C_1 P_k \epsilon - \rho C_2 \frac{\epsilon^2}{k} + \frac{\partial}{\partial x_1} \left( \nu_t \frac{\partial \epsilon}{\partial x_1} \right) \right),
\]  
(3.112)
\[
0 = \frac{\nu_t}{k} \left( \frac{\partial U}{\partial n} \right)^2.
\]  
(3.113)
\[
P_k = \rho \nu_t \left( \frac{\partial U}{\partial n} \right)^2.
\]  
(3.114)

Substituting equation 3.105, two ODEs are obtained \([72, \text{p} 58]\). These are satisfied by the solution
\[
k = \frac{u_r^2}{\sqrt{C_\mu}}.
\]  
(3.115)
\[
\epsilon = \frac{u_r^2}{\kappa n}.
\]  
(3.116)

Equation 3.115 implies that we can express \( u_r \) in terms of \( k \), as follows:
\[
u_t = C_{\mu}^{1/4} \sqrt{k}.
\]  
(3.117)

This gives identical results to 3.103 if the logarithmic law is in fact satisfied. If this is not the case, the boundary condition cannot be accurate; however, as the turbulence kinetic energy is guaranteed to be positive everywhere 2.177 will not break down. Near a recirculation zone \( k \) is in fact generally rather high.

\(^{10}\)This implies the mean flow is steady, which it may of course not be. However, the near wall layer adjusts to the outer layer very rapidly, since the eddies there have a time scale proportional to the distance from the wall.
It remains to define boundary conditions in the \( k \) and \( \epsilon \) equation. In the logarithmic region, the shear stress \( \tau \) is constant. Since this implies, by equation 3.117, that \( k \) is also constant, the appropriate boundary condition for \( k \) is zero gradient in the direction \( n \) normal to the boundary. The boundary condition for \( \epsilon \) is more problematic, since gradients of this quantity near a wall are very high. Integration through the wall layer would be very costly, and also unnecessary, since \( \epsilon \) is not required in the \( k \) and momentum equations there. The issue is avoided by effectively imposing Dirichlet boundary conditions at the node nearest the wall instead of at the wall itself. There, the logarithmic law has already been assumed to apply, so it is appropriate to set \( \epsilon \) to the value given by equation 3.116. This is accomplished by setting all coefficients \( A_P \) to zero, \( A_P \) to unity, and \( \Omega_q \) to 3.116. The transport equation then reduces to \( \phi = Q_\Omega \), which has the desired effect.

It remains to determine the approach to adopt if \( n^+ \) is found to be outside the range in which the wall function is valid, particularly if \( n^+ \) is found to be too small. Two methods were tested. The first simply does nothing except to require that the viscosity \( \nu_t \) be positive. The effect of this is that the wall function will not be imposed for \( u_r \) close to zero, which would be the case in laminar flow. The second, which more closely approaches the usual practice in CFD, is to set \( n^+ \) to some minimum value if it found to be lower and then apply the wall function as usual.

### 3.7.4 Modifications for Buoyant Flows

In stratified flows, turbulence kinetic energy may be produced or removed as a result of turbulent scalar fluxes. This is modelled by adding the following term to the production \( P_k \)

\[
G_k = -\rho \beta \frac{\nu_t}{f_\sigma} \frac{\partial^2}{\partial x_j} g \cdot \nabla \phi, \tag{3.118}
\]

where \( \beta \) is the coefficient relating density to the concentration of a particular species (for instance, salinity), or to temperature (in the latter case, \( \beta \) is the volumetric expansion coefficient). In the \( \epsilon \) equation, this term is multiplied by a constant \( C_{\epsilon^3} \), which has been found to be a function of stability - a value of approximately unity appears to be appropriate for unstably stratified flow, whereas zero appears indicated for stable stratification [96]. The production terms in the two equations become respectively

\[
P_k = \rho \nu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} - \rho \beta \frac{\nu_t}{f_\sigma} g \cdot \nabla \phi, \tag{3.119}
\]

\[
P_\epsilon = \rho \nu_t C_{\epsilon^3} \frac{\epsilon}{k} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} - \rho \beta C_{\epsilon^3} \frac{\nu_t}{f_\sigma} g \cdot \nabla \phi. \tag{3.120}
\]

The term \( G_k \) represents the transfer of energy between turbulent kinetic energy and the potential energy of the density stratification. Turbulent diffusion will tend to erase whatever stratification
exists; if the stratification is stable, then it is in a low energy state, and bringing it to neutral stratification raises its potential energy. Energy is therefore removed from the turbulence, reflected in the fact that $G_k$ is then negative. The opposite is true when unstable stratification exists.

The ratio of the buoyant to shear production terms in the turbulence kinetic energy equation is known as the flux Richardson number

$$R_f = \frac{-G_k}{P_k}. \quad (3.121)$$

If the production and dissipation terms ($P_k, G_k$ and $\epsilon$) are in local equilibrium, it is possible to show that the constant $C_p$ varies as $1 - R_f$ [95]. A modification of this kind is therefore frequently employed when computing stratified turbulent flows [81]. Occasionally, the gradient Richardson number is then used. As discussed above, this quantity is not very meaningful in turbulent flow, and since the turbulence kinetic energy is being solved in any case, the flux Richardson number has been used instead.

A further modification has been tested, of the turbulent Schmidt number $\sigma_t$. This, instead of being a constant close to unity, is instead obtained from equation 2.197 above.

Unless otherwise stated below, all results presented below will have been obtained using $C_{\epsilon 3} = 0$ and $\nu_t$ given by equation 2.193. A turbulent Schmidt number of 0.93 will be used.

### 3.8 Implementation

#### 3.8.1 Convergence Criteria

The reduction of the global residual sum is used as a stopping criterion, in both the inner and outer iterations. The relative residual reduction is used in both cases.

$$\frac{R}{R_0} < \epsilon, \quad (3.122)$$

where $R$ is the global residual sum and $\epsilon$ the tolerance.

It is, however, recognized that this criterion is not very satisfactory, since the residual may not be reduced monotonically. In principle, it is better to estimate the error by comparing results from successive iterations [33, p 116]; however, this approach is much more expensive than using the residual (which is calculated by the solver in any case). Nor is this estimate definitive - it can in fact also be misleading if a great deal of care is not taken [94, p 174].
A secondary criterion is therefore specified, which takes the form of a minimum number of outer iterations to be carried out. Some preliminary investigations established that 50 iterations were generally more than sufficient to reduce all residuals by a factor of at least $10^{-6}$ on the medium grid. This number was therefore chosen as the minimum (for that grid; 75 was felt to be more appropriate on the fine grid).

In practice, it was subsequently found that this was a very conservative estimate, as the final residuals were generally many orders of magnitude lower than the criterion initially thought adequate. The secondary criterion was therefore relaxed somewhat for some of the later calculations. It was found that 25 iterations were always sufficient, except for a short period of typically one timestep, associated with transition to turbulent flow in the wave breaking region. The $\epsilon$ equation then converged more slowly.

### 3.8.2 Calculation of Gradients

Derivatives are obtained by requiring that the difference between two nodal values be equal to the dot product of the derivative and the position vector from one node to the other. It is possible to implement this in several ways, depending on the nodes selected. One way would be to consider the difference between the cell centre at which the derivative is to be computed and all of its neighbours. If hexahedral control volumes are used, the resulting equation system is then overdetermined, as there are six pairs of nodes and only three Cartesian derivatives. This problem can be overcome by using a least-squares fit, as was done by Muzaferija and Gosman [76]. The advantage of this approach is that it generalizes readily to unstructured grids containing control volumes of arbitrary shape (as no assumption is made regarding the number of neighbours).

However, the approach adopted was the more conventional one of using central differences, and assuming the derivative to be constant over each control volume (which approximation is anyway implicit in the second-order cell-centered discretization). It is then a reasonable approximation to use the differences between the three pairs of neighbouring cells, which yields a closed equation system. For the east and west nodes, for instance:

$$\phi_E - \phi_W = (\nabla \phi)_P \cdot (r_E - r_W).$$

(3.123)

This equation is then solved for $\nabla \phi$. 
3.8.3 Solution of Linear Equation System

The discretizations described in the sections above all result in systems of linear equations. These are solved in sequence by a suitable method; the coefficients are then all reevaluated and the process repeated, until all the linear equations satisfy, to some specified tolerance, the (generally nonlinear) transport equations they approximate. These iterations are called 'outer iterations' because the solver may itself be iterative (its iterations are then called 'inner iterations').

The strongly implicit procedure of Stone [105] was used to solve the linear equation systems. This is an iterative procedure, which proceeds on the assumption that the solution will be smooth; this is generally a good assumption for elliptic partial differential equations. Discontinuities are not expected in the problems to be considered, particularly as a turbulence closure model will be used.
3.9 Application to Stratified Flow over Hills

3.9.1 Hypotheses to be Tested

The following hypotheses are to be tested:

1. The hypothesis of Smith [101], that low Froude number flows over hills can be treated within the context of his theory by approximating the blocked flow upstream of the hill as a solid surface.

2. That a constant turbulent Schmidt number is not appropriate for stably stratified flows.

3. That, in stably stratified flows, a value of zero is more appropriate for the constant $C_\alpha$ than one of unity.

4. That stratification in the form of discrete layers reproduces the effect of linear stratification.

5. That the size of the domain has no effect on the local solution before the arrival of internal waves reflected from the domain boundaries.

6. That effects of blocking increase with the aspect ratio of the hill, and that they are consistent with the theory of Smith for high aspect ratios.

7. That the wave-breaking envelope in the $Fh-\alpha$ plane is as described by Castro and Snyder.

8. That merging of the low-level rotor and the breaking wave zone occurs for the COS3 hill, at a Froude number of 0.6; also, that it does not occur for any of the other cases investigated.

9. That the Reynolds-averaged simulation of the flow is symmetrical about the channel centreline.

The first hypothesis can be tested by simulating flows over hills that are expected to result in wave breaking. The heights of the dividing streamlines $H_0$ and $d$ can then found, and compared with the theory. The depth of the mixed region $\delta_C$ can also be compared with predictions.

Since the theory is two-dimensional, wide hills are expected to provide results in rather better agreement with it than are narrow hills.

Hypothesis 2 will be tested by performing simulations of the same stratified flow using both a constant turbulent Schmidt number, and equation 2.197 instead. A third simulation will be
performed using both equation 2.197 for the Schmidt number and equation 2.198 for the constant $C_D$, in order to assess the importance of the value of $C_D$ relative to that of $\sigma_L$.

Hypothesis 3 will be tested by performing simulations of the same stratified flow using values of $C_{e3}$ of zero and of unity. The results will be compared to experiment.

Hypothesis 4 is often invoked in analytical models of lee wave phenomena (such as that of Peltier and Clark [86]). Experimental work is also often performed using discrete layers of various depths. This hypothesis is being tested in order to investigate the influence of layering on such parameters as the height at which the lee wave overturns and breaks. The results are also expected to be of some interest from a turbulence modelling point of view. Since regions of fluid will exist in which there is no stratification (and thus less suppression of turbulence), it is reasoned that it might be possible for the turbulence model to give different results from those obtained in the linearly stratified flow, even though the mean flow ought not to be much affected. Hypothesis 4 is to be tested by approximating the stratification in a stepwise fashion, using layers of constant density instead of linear variation. Layers of three different depths will be tested.

Hypothesis 5 is to be tested by performing additional simulations in extended domains. One will be extended in all three coordinate directions, one in width and length only, and one only in the lengthwise direction.

Hypothesis 6 is to be tested by comparing the upstream height of the dividing streamline for different hill aspect ratios.

Hypotheses 7 and 8 will be tested by performing simulations for a number of hills and Froude numbers, and comparing the results with the experiments of Castro and Snyder [17]. These hypotheses (number 8 in particular) are a test of the adequacy of the turbulence model.

Hypothesis 9 will be tested by performing simulations including the full width of the channel and comparing the results to those employing a symmetry condition on the centreline.

In order to test for the occurrence or otherwise of wave breaking, some criterion must be decided upon. Two possibilities present themselves. The first and most obvious one is to test for the presence of a vertical streamline not connected with the ground. In particular, a closed streamline aloft would qualify. We would expect to find such a streamline somewhere in between the hill and the first lee wave crest, some distance above the lower surface (a recirculation zone is expected beneath the crest of the first lee wave, but this is merely a low level rotor).

It is arguable, perhaps, that requiring the horizontal velocity $U$ to reverse sign is overly restrictive, and that flows in which $U - \overline{U}$ is negative are, in a sense, breaking. The flow will be overturning turbulently, but the mean flow may still be positive. However, overturning in this sense may happen whenever there is sufficiently strong turbulence. If the lee waves break down through
Kelvin-Helmholtz instability, for instance, this criterion might well be satisfied, but it would not mean wave breaking in the usual sense of that phrase.

This does not mean, however, that such cases are not of interest (for instance, reversal of the flow will create a critical level preventing the upward propagation of wave energy). Both criteria will therefore be employed below. The first criterion will in practice be applied by examining the streamlines of the solution obtained. The second, weaker criterion will be implemented by plotting

$$U - \bar{u}' = U - \sqrt{\frac{2}{3} k},$$  \hspace{1cm} (3.124)

where $k$ is the turbulence kinetic energy. Equation 3.124 follows from the definition of turbulence kinetic energy in the $k-\varepsilon$ model, which assumes isotropy of normal Reynolds stresses. If the sign of 3.124 is negative, we can conclude that turbulent instabilities are strong enough to reverse the flow intermittently.

### 3.9.2 Flow and Fluid Properties

Almost all simulations to be described below were performed at a Reynolds number of $10^4$ (except for those performed for the specific purpose of assessing Reynolds number independence). A value of 675 was assumed for the molecular Schmidt number, and 0.93 for the turbulent Schmidt number. The high value of the molecular Schmidt number is appropriate for stratification induced by salinity, which was the method employed in the experimental work. It is not strictly appropriate for thermally stratified air, where the (equivalent) Prandtl number is approximately three orders of magnitude lower. However, provided the Reynolds number is sufficiently high, viscous diffusive fluxes will be sufficiently insignificant to make any results applicable to atmospheric flows also.

### 3.9.3 Nondimensionalization

All physical quantities were nondimensionalized using the following mass, length, and time scales:

- **Length:** $h = 0.1 \text{ m}$
- **Time:** $h/U_0 = 1.0 \text{ s}$

where $h$ is the obstacle height and $U_0$ the velocity at which it is towed. Since the Boussinesq approximation is used, the mass does not appear in any quantity except the coefficient $\beta$. In order to simulate a Froude number $F_k$, we require

$$F_k^2 = \frac{U^2}{N^2 h^2} = \frac{U^2}{\frac{\partial e}{\partial z} h^2},$$  \hspace{1cm} (3.125)
where \( \rho_0 \) is the (constant) background density. We can replace \( \rho \) with a density-determining scalar \( \theta \), if we ensure that
\[
\frac{1}{\rho} \frac{\partial \rho}{\partial z} = \beta \frac{\partial \theta}{\partial z},
\]
where \( \beta \) is
\[
\beta = \frac{-\theta}{F_h} \left( \frac{\partial \theta}{\partial z} \right). \tag{3.127}
\]

If we choose a value of 10 for \( \partial \theta / \partial z \), we obtain the following values for \( \beta \)

<table>
<thead>
<tr>
<th>( F_h )</th>
<th>( \beta )</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.55</td>
<td>0.331</td>
</tr>
<tr>
<td>0.6</td>
<td>0.278</td>
</tr>
<tr>
<td>0.7</td>
<td>0.204</td>
</tr>
<tr>
<td>0.75</td>
<td>0.178</td>
</tr>
<tr>
<td>0.8</td>
<td>0.156</td>
</tr>
</tbody>
</table>

which were the values (of both quantities) used in the simulations performed.

### 3.9.4 Definition of Computational Domain

The experimental flows to be simulated were carried out in a towing tank 1 metre deep, 2.4 metres wide, and 25 metres long.

The computational domain is defined as a short section of the tank, moving with the hill, which is considered stationary. The lengthwise dimension, in the direction of the upstream flow, will be referred to as \( x \), the spanwise dimension, \( y \), and the vertical dimension, \( z \).

Most results were obtained using a symmetry on the lateral centreline (\( y = \text{constant} \)). The validity of the implied assumption of flow symmetry was tested by performing some simulations including the full channel width. The hill was mounted on a base plate extending eight hill heights both up and downstream. The standard boundary conditions used are, therefore, in full
The standard domain (extended versions of which will also be tested), is shown below in figure 3.9.

![Computational Domain](image)

Figure 3.9: Computational Domain

All dimensions are given in hill heights $h$. For all the cases to be considered, $h = 10$ cm in dimensional units.

The dimensions of the computational domain correspond exactly to those of the experiment, with the exception of the domain length, which is somewhat shorter (15m in total, while the actual length was 25m). The influence of the domain length was investigated by performing simulations in an extended domain. Domains extended in the spanwise and vertical dimensions were also used, in order to try to determine the influence of finite domain size in general.

Three grids were used to subdivide the domain. The high and low resolution grids were only used to assess the grid dependence of the solution; all other results presented below were calculated using the medium resolution grids. The number of grid points in each direction is summarized in the following table:

<table>
<thead>
<tr>
<th></th>
<th>Low Resolution</th>
<th>Medium Resolution</th>
<th>High Resolution</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid points in x</td>
<td>72</td>
<td>108</td>
<td>144</td>
</tr>
<tr>
<td>Grid points in y</td>
<td>24</td>
<td>32</td>
<td>48</td>
</tr>
<tr>
<td>Grid points in z</td>
<td>32</td>
<td>48</td>
<td>64</td>
</tr>
</tbody>
</table>
All of the above grids were stretched and non-orthogonal. The grid in the immediate vicinity of the hill was kept uniform. As an example, the uniformly gridded region of the medium grid for the COS3 hill is given in the table below.

<table>
<thead>
<tr>
<th>Minimum</th>
<th>Maximum</th>
</tr>
</thead>
<tbody>
<tr>
<td>Uniform grid in x</td>
<td>-1</td>
</tr>
<tr>
<td>Uniform grid in y</td>
<td>0</td>
</tr>
<tr>
<td>Uniform grid in z</td>
<td>0</td>
</tr>
</tbody>
</table>

For the low resolution grid, the maximum z value was lowered to 1.5, while it was increased to 2.0 for the high resolution grid. Outside this region, the grid was stretched by applying a geometric progression. The ratios used were 1.2 in x and y and 1.15 in z.

The $k$-$\epsilon$ model assumes a logarithmic variation of the velocity in the control volumes adjoining a wall. This can make grid independence hard to attain or prove, since this law is only ever correct for a particular range of values of the (nondimensional) grid spacing. For flows at low to moderate Reynolds numbers, this range can be small. Decreasing the grid spacing near the wall can, therefore, effectively result in the application of the wall function in a region where it is not valid, although the unrefined grid might not have had this effect. It is therefore quite possible for fine grids to yield unphysical results. This is partially dealt with by fixing the depth of the control volumes adjoining the surface of the hill so as to be the same for all grids. The value chosen was 0.09375 in nondimensional units, a little larger than the vertical extent of the control volumes above it; the resolution above the near-wall region is kept has high as possible in order to adequately resolve that portion of the boundary layer not contained in the first cell.

Grid generation was carried out using simple programs written for the purpose. The method used will be briefly described:

For a given geometric grid expansion ratio $\gamma$, it is possible to relate the size of the smallest grid cell to the size of the domain as follows:

$$\Delta x = \frac{x_{\text{max}} - x_{\text{min}}}{\sum_{i=0}^{n-1} \gamma^i},$$

(3.128)

where $n$ is the number of nodes allocated to that particular dimension. The points required to resolve the region with a uniform grid can be calculated iteratively by requiring the grid resolution in that region to match the value of $\Delta x$ just outside it. The latter is calculated using equation 3.128 above, and progressively more nodes allocated to the uniform region until convergence is satisfactory.

Grid expansion in two directions, and a fixed size for the near-wall grid cell, can be dealt with in the same framework with little more difficulty.

It is also necessary to ensure that grid vertices were present on the lower boundary at the precise
edges of the lower wall boundary, so that any given cell contains only one boundary condition (attempts were made to try to apply two, but this proved difficult). The wall boundary begins and ends at 8 hill heights up and downstream of the hill respectively. The necessary adjustments were made using a text editor, by finding nodes close to ±8, and adjusting their location slightly. An iterative method could also perform this task, but since all nodes were arranged in vertical columns, moving all those at any particular location in x and y was trivial.

The medium grid is shown in figures 3.10, 3.11 and 3.12 below.

Figure 3.10: Grid in the y and z plane
Figure 3.11: Close up of grid in the $x$ and $z$ plane near the hill
The resolution used for the time dimension was usually the same; 0.1 nondimensional time units (equal to the hill height divided by the upstream velocity).

This is partly because no less was thought to be guaranteed to be adequate, on the grounds that the Courant number would grow much larger than unity, so that the accuracy of the advection scheme could not be guaranteed. However, the physical oscillation period inherent in the system \((2\pi/N, \text{where } N \text{ is the Brunt-Väisälä frequency})\) has been shown to be adequately resolved by a time step of this size (see section 3.4.1 above). There is thus reason to believe the time step to be sufficiently small; this was tested by carrying out one simulation with the size of the time step doubled.

**3.9.5 Inflow and Initial Conditions**

The following values were prescribed at the inflow boundary:

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>(U)</td>
<td>1.0</td>
</tr>
<tr>
<td>(V)</td>
<td>0.0</td>
</tr>
<tr>
<td>(W)</td>
<td>0.0</td>
</tr>
<tr>
<td>(\theta)</td>
<td>100-10z</td>
</tr>
<tr>
<td>(k)</td>
<td>1.0E-8</td>
</tr>
<tr>
<td>(\epsilon)</td>
<td>1.0E-8</td>
</tr>
</tbody>
</table>

where \(\theta\) denotes the density determining scalar, which increases (in most cases) linearly with depth. All values are nondimensional.
Initial conditions were identical to the inlet conditions, corresponding to an impulsive start.
4. Results

4.1 Summary of some Validation Tests

Before applying the method described above to the problems that are the subject of this work, some tests were carried out in order to validate the implementation of the method. Some of these will be briefly described here.

One set of test cases designed specifically for the validation of methods using skewed grids is that described by Demirdzić et al [23]. These also have the advantage of exercising the scalar transport equation, so that most features of the method are tested simultaneously. The test cases detailed in [23] include, amongst others, shear driven and buoyancy driven cavity flows. Both of these were simulated using the current method, and in both cases the results of Demirdzić et al were replicated. However, the following description will focus on the buoyancy-driven flows, since the shear-driven flows contain no physical or numerical aspects not present in the buoyancy-driven cases (with the relatively trivial exception of moving boundaries).

The two cases considered are flows within two-dimensional cavities skewed at an angle of 45 degrees, such that the left and right hand walls are inclined at this angle to the vertical, and the top and bottom walls are horizontal. Adiabatic wall boundary conditions are imposed on the top and bottom walls, and isothermal boundary conditions on the left and right hand walls. The left wall is 'hot', and the right wall 'cold'. Heat transfer takes place by natural convection. The problem parameters were as follows:

- Rayleigh number: \(10^6\)
- Length of horizontal boundaries: 1.707
- Length of inclined boundaries: 1.000
- Temperature of hot wall: 1.000
- Temperature of cold wall: 0.000
- Density: 1.000

Two test cases were considered. These varied in the following respects:

<table>
<thead>
<tr>
<th>Case 1</th>
<th>Case 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Viscosity:</td>
<td>1.0E-03</td>
</tr>
<tr>
<td>Prandtl number:</td>
<td>0.1</td>
</tr>
</tbody>
</table>

Streamline and scalar contour plots are included at the end of this section.

These tests did not exercise the turbulence model implementation. One test that that did was the
simulation of a turbulent Couette flow. The results were compared to experimentally obtained ones (see [97]) which are reproduced in figure 4.9 below.

Turbulent plane Couette flow at a Reynolds number of 34000 was simulated using a long domain with inlet and outlet boundaries. When a solution was obtained, the solution at the outlet boundary was copied onto the inlet, and the process repeated until no further change was evident.

This was done partly because reliable periodic boundary conditions were not available, but also because the inlet and outlet boundary conditions would be the ones actually used to simulate flow over hills, and thus results obtained using periodic boundary conditions would be less useful for validation purposes.

Results are shown below in figures 4.9 and 4.10, and are believed to be satisfactory.

The above test cases have the disadvantage of being two-dimensional. This was addressed by simulating a three-dimensional shear-driven cavity flow at a Reynolds number of 100, and the results obtained compared to those of Deng et al. [24]. The cavity was cubic, each dimension equal to unity, as was the velocity imposed on the top wall.

Plots of horizontal velocity $U$ and vertical velocity $V$ on the geometrical symmetry plane of the cavity are shown below in figures 4.11 and 4.12. The extreme points of these quantities are also compared in the table below; differences amount to approximately 1 or 2 %, which is felt to be acceptable. Note that a slightly lower grid resolution was used to reproduce them, as well as a different advection scheme (the current results were obtained using the van Leer scheme, while Deng et al employed a scheme described in their paper).

<table>
<thead>
<tr>
<th></th>
<th>Deng et al</th>
<th>Current work</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Minimum value of U</td>
<td>-0.21488</td>
<td>-0.21168</td>
<td>1.5 %</td>
</tr>
<tr>
<td>Maximum value of V</td>
<td>0.15228</td>
<td>0.14932</td>
<td>1.9 %</td>
</tr>
<tr>
<td>Minimum value of V</td>
<td>-0.24846</td>
<td>0.24261</td>
<td>2.4 %</td>
</tr>
</tbody>
</table>
Figure 4.1: Streamlines, for Pr = 0.1 Reproduced from Demirdzić et al [23].

Figure 4.2: Streamlines, for Pr = 0.1, obtained using the current method.
Figure 4.3: Streamlines, for $Pr = 10$. Reproduced from Demirdzić et al [23].

Figure 4.4: Streamlines, for $Pr = 10$, obtained using the current method.
Figure 4.5: Temperature Contours, for Pr = 0.1 Reproduced from Demirdžić et al [23].

Figure 4.6: Temperature Contours, obtained using the current method.
Figure 4.7: Temperature Contours, for Pr = 10 Reproduced from Demirdžić et al [23].

Figure 4.8: Temperature Contours, obtained using the current method.
Figure 4.9: Turbulent Couette Flow. Reproduced from Schlichting [97].

Figure 4.10: Turbulent Couette Flow, simulated with current method.
Figure 4.11: Shear-driven three-dimensional cavity flow, from Deng et al [24]. Horizontal and Vertical Velocity profiles on the symmetry plane.

Figure 4.12: Shear-driven three-dimensional cavity flow, current method. Horizontal and Vertical Velocity profiles on the symmetry plane.
4.2 COS1 Hill

This is the axisymmetric cosine hill used by Castro and Snyder [17]. Stratified flows at three Froude numbers (0.55, 0.6 and 0.7) were simulated. As with all the results to be presented in this section, the medium resolution grid (described in section 3.9.4 above) was used. The turbulence model used was the standard high Reynolds number \( k\varepsilon \) model, modified by the use of equation 2.193 for \( \nu_t \).

Wave breaking seems to be present for this flow at the lowest Froude number. This fact is not in agreement with observations made in the experimental work, where wave breaking was observed only at \( F_h = 0.3 \). It is, however, somewhat debateable whether the flow (shown in 4.14) really does involve breaking waves. No wave breaking was observed in the simulations at the two higher Froude numbers.

Castro and Snyder did not present photographs of dye visualizations (at these particular Froude numbers - some were presented for lower \( F_h \)), so comparisons with such cannot be made. However, the results may be compared with simulations on different hills, and are therefore presented in the same way as these. Figures 4.14, 4.15, and 4.16 show streamlines obtained by tracking the movement of massless particles. This is felt to be the most appropriate way to compare the
simulation results with dye visualizations.

The development of the flow (for $F_h = 0.6$) in time is illustrated in figure 4.17. A steady state is soon seen to be reached.

For further details of these flows, the reader is referred to sections 4.6 and 4.7 below, where some integral flow quantities are presented graphically.

Figure 4.14: Simulated streamlines for the COS1 Hill, at a Froude number of 0.55. These results are from $t=35$. 
Figure 4.15: Simulated streamlines for the COS1 Hill, at a Froude number of 0.6. These results are from $t=35$.

Figure 4.16: Simulated streamlines for the COS1 Hill, at a Froude number of 0.7. These results are from $t=45$. 

122
Figure 4.17: Streamlines, COS1 Hill, $P_a=0.6$. Plots are of solutions at multiples of 5 time units, starting at $t=5$. 
4.3 COS2 Hill

This is the COS2 hill used by Castro and Snyder [17]. Stratified flows at three Froude numbers (0.6, 0.7, and 0.75) were simulated. The medium resolution grid (described in section 3.9.4 above) was used. The turbulence model used was the standard high Reynolds number $k$-$\varepsilon$ model, modified by the use of equation 2.193 for $\nu_t$.

Wave breaking was simulated for this hill at Froude numbers of 0.6 and 0.7, but not 0.75. Agreement with experiment is mixed. At a Froude number of 0.6, photographs of the experimental flow visualized with dye streamers (figure 4.19 overleaf) are compared with streamline plots (obtained by tracing the paths of massless particles, which presumably would be transported in roughly the same way as dye). In the experimental flow, an isolated dark area area is visible aloft in the immediate lee of the hill. If this is a rotor, it is not simulated; this particular flow does not appear to be particularly well reproduced.

At a Froude number of 0.7, Castro and Snyder reported marginal wave breaking (figure 4.21). The simulated solution, shown in figure 4.22, is in good agreement with this.

No wave overturning is found in the solutions for $F_h = 0.75$; this is also in good agreement with
the results of Castro and Snyder.

Further results for this hill at $F_h = 0.6$ are shown in figures 4.24, as streamlines at various time steps.
Figure 4.19: The flow over the COS2 Hill at a Froude number of 0.6. This is Castro and Snyder's figure 3d [17].

Figure 4.20: Simulated streamlines for the COS2 Hill, at a Froude number of 0.6. These results are from $t=35$. 

Figure 4.21: The flow over the COS2 Hill at a Froude number of 0.7. This is Castro and Snyder's figure 3c [17].

Figure 4.22: Simulated streamlines for the COS2 Hill, at a Froude number of 0.7. These results are from t=45.
Figure 4.23: Simulated streamlines for the COS2 Hill, at a Froude number of 0.75. These results are from $t=45$. 
Figure 4.24: Streamlines, COS2 Hill, $F_s=0.6$. Plots are of solutions at multiples of 5 time units, starting at $t=5$. 
4.4 COS3 Hill

This is the COS3 hill used by Castro and Snyder [17]. Stratified flows at five Froude numbers (0.55, 0.6, 0.7, 0.75, and 0.8) were simulated. The medium resolution grid (described in section 3.9.4 above) was used. The turbulence model used was the standard high Reynolds number $ke$ model, modified by the use of equation 2.193 for $\nu_t$.

Wave breaking was found in the simulation results for $F_h = 0.6$ and 0.7, while no wave breaking was found for $F_h = 0.8$. In this respect, the simulations are in good agreement with experiment. Visual comparison of streamline plots (figures 4.27, 4.29 and 4.31 below) with experimental flow visualizations (figures 4.26, 4.28 and 4.30 below) is also encouraging.

Figures 4.40, 4.43, 4.46 at the end of this chapter show streamline plots at various times. Figure 4.44, also shows contours of the density determining scalar, while figures 4.41 and 4.45 show contours of the turbulence kinetic energy for the cases at $F_h = 0.6$ and $F_h = 0.7$.

At a Froude number of 0.6, wave overturning takes place at around $t=10$ (see figure 4.40). The flow in the lee of the hill subsequently evolves into a deep, turbulent (see figure 4.41) and well mixed region.
This region may correspond to the ‘merging’ flow, observed by Castro and Snyder for this hill and Froude number. The streamline plot (see figure 4.27) closely resembles the experimental flow, but, as figure 4.42 shows, the breaking and recirculation regions are in fact distinct, and there is a region dividing the two in which the flow does not appear to reverse, even intermittently. Some true merging can be seen earlier, at t=20, but this is not sustained.

Hypothesis 8 (merging takes place at \( F_h = 0.6 \) for the COS3 hill, and not for any of the other flows investigated) is partially supported by these results. The wave breaking and recirculation regions appear connected, but on closer inspection, we find that they are distinct to some extent. Moreover, the flow downstream of the COS2 hill at \( F_h = 0.6 \) appears somewhat similar (see figure 4.24), so the second part of the hypothesis is not fully supported.

At a Froude number of 0.7, wave overturning takes place at around t=15 (see figure 4.43); the flow then undergoes a transition (during which turbulence appears, see figure 4.45) at around t=25.

It is worth noting that until after t=20, there is essentially no turbulence anywhere. Contour plots of turbulence kinetic energy are shown in figure 4.45, for the COS3 hill at a Froude number of 0.7. These plots begin at t=25, for at t=20 \( k \) is still approximately zero everywhere. Only after t=20 does turbulence develop. A transition period then follows, as a sudden growth in turbulence kinetic energy temporarily erodes the breaking wave. Eventually the wave adjusts, and the breaking region reforms, this time turbulently. Thereafter, the flow appears to approach a steady state, although integration to higher t would be required to show this to be the case beyond doubt. Certainly, the turbulence kinetic energy contours of figure 4.45 appear to be approaching a state resembling that postulated in the theory of Smith [101]. By about t=40, a steady state seems to have been reached.

This pattern of laminar overturning followed by a transition to turbulence is found in all the simulation results that exhibit wave breaking, although the time at which transition takes place varies.

Some comment should also be made regarding the recirculation zones in contact with the lower surface. There are in fact three flow reversal zones at \( z = 0 \), visible in figure 4.36. The third of these appears at \( x=10 \) just downstream of the trailing edge of the plate (which is at \( x=8 \)). Such a feature was not commented on by Paisley and Castro, but was noted by Castro and Snyder in their experimental work, albeit for \( F_h=0.8 \) only (neither its presence nor absence were noted for any other \( F_h \)).

The butterfly-shaped main flow reversal region, centred around \( x=6 \), appears to be displaced downstream from the location of the same feature in the results of Paisley and Castro [82]. It is evident from figures 4.37 and 4.38 that considerable advection into this region takes place; a low-level stream of reversed flow can be seen at \( y=1.5 \), which impinges on the edge of the jump. It is seen to be at least in part supplied by fluid from higher \( y \), with (rather weak) contra-rotating
vortices visible, one centered around $x=4$, $y=2.5$, and the other at approximately $x=4.9$, $y=0.9$.

Figure 4.37 also shows that the edge of the jump is still quite sharp, in spite of the turbulent flow in and around it. Further downstream, in figure 4.38 the third recirculation zone is apparent, as is the relatively sharp edge of the wake even far downstream.

At a Froude number of 0.8, the wave initially develops as it does for $F_h = 0.7$, but appears to lag behind it, so that, for instance, the streamline plot for $F_h = 0.8$ at $t=15$ looks rather like the one for $F_h = 0.7$ at $t=10$. Eventually, the flow undergoes a turbulent transition, during which some turbulent flow reversal does take place (at $t=25$, see figure 4.35). This quickly disappears again, as the turbulence generated smoothes out the instability. The lee waves at higher $t$ do appear a little less steep than those observed experimentally; the earlier purely laminar flow appears to be closer. This suggests that the turbulence model is not quite satisfactory for the very intermittent turbulence seen in this flow.
Figure 4.26: The flow over the COS3 Hill at a Froude number of 0.6. This is Castro and Snyder's figure 2c [17].

Figure 4.27: Simulated streamlines for the COS3 Hill, at a Froude number of 0.6. These results are from t=25.
Figure 4.28: The flow over the COS3 Hill at a Froude number of 0.7. This is Castro and Snyder's figure 2b [17].

Figure 4.29: Simulated streamlines for the COS3 Hill, at a Froude number of 0.7. These results are from t=45.
Figure 4.30: The flow over the COS3 Hill at a Froude number of 0.8. This is Castro and Snyder's figure 2a [17].

Figure 4.31: Simulated streamlines for the COS3 Hill, at a Froude number of 0.8. These results are from t=45.
Figure 4.32: Simulated streamlines for the COS3 Hill, at a Froude number of 0.55. These results are from $t=35$.

Figure 4.33: Simulated streamlines for the COS3 Hill, at a Froude number of 0.75. These results are from $t=45$. 
Momentum Vectors in the Windward Separation Region
COS3 Hill, Fh = 0.7, at z = 0.047, t = 40

Figure 4.34: Vectors Plot of Windward Recirculation Zone, COS3 Hill, Fh = 0.7

Contours of $U - \bar{u}'$ on y=0, t=25

Figure 4.35: Contours of $U - \bar{u}'$, COS3 Hill, Fh = 0.8.
Contours are drawn in increments of 0.1 on a base of -0.5. The solid line is the contour at zero.
The fluctuating velocity $\bar{u}'$ is given by $\sqrt{2/3} k$. 

137
Figure 4.36: Contours of $U$ near the wall, showing the recirculation zones, COS3 Hill, $F_h = 0.7$. The solid line is the contour at zero $U$. This contour is taken at $z = 0.047$ above the surface.

Figure 4.37: Vector plot of lee recirculation zone, COS3 Hill, $F_h = 0.7$. 

138
Figure 4.38: Vector plot of obstacle wake, COS3 Hill, $F_h = 0.7$.

Figure 4.39: Contour plot of vertical velocity aloft, COS3 Hill, $F_h = 0.7$

Contours are drawn in increments of 0.075 on a base of -0.3. The strong solid line denotes zero velocity, light solid lines positive velocity, and dashed lines negative velocity.
Figure 4.40: Streamlines, COS3 Hill, $F_h=0.6$. Plots are of solutions at multiples of 5 time units, starting at $t=5$. 
Figure 4.41: Contours of Turbulence Kinetic Energy, COS3 Hill, $F_k=0.6$ Contours are drawn in increments of 0.005 on a base of 0.
Figure 4.42: Contours of $U - \bar{u}$, COS3 Hill, $F_k = 0.6$. Contours are drawn in increments of 0.1 on a base of -0.5. The solid line is the contour at zero. The fluctuating velocity $\bar{u}$ is given by $\sqrt{2/3 \, k}$. 
Figure 4.43: Streamlines, COS3 Hill, $F_h=0.7$. Plots are of solutions at multiples of 5 time units, starting at $t=5$. 
Figure 4.44: Scalar Contours, COS3 Hill, $F_r=0.7$ Contours are drawn in increments of 5.0 on base of 0. Plotted are solutions separated by 5 nondimensional time units, starting at $t=20$. 
Figure 4.45: Contours of Turbulence Kinetic Energy, COS3 Hill, $F_h=0.7$. Contours are drawn in increments of 0.005 on a base of 0.
Figure 4.46: Streamlines, COS3 Hill, $F_h=0.8$. Plots are of solutions at multiples of 5 time units, starting at $t=5$. 
4.5 Wave Drag and Comparison with Neutral Flow

A simulation of neutrally stratified flow over the COS3 Hill was carried out, although no experimental data are available for comparison. A streamline plot (generated by tracing the paths of massless particles released far upstream, as in the stratified cases described above) of the flow over and in the lee of this hill is shown below in figure 4.47; note the separation from the hill surface, which is suppressed by sufficiently stable stratification.

The availability of results for the neutrally stratified case makes it possible to make an estimate of the wave drag, which is assumed to be that portion of the drag due to stable stratification. This may not be entirely accurate, as the effects of the latter may, for instance, reduce frictional drag by suppressing turbulence. However, since the obstacle is steep, and the Reynolds number high enough to induce separated flow, it may safely be assumed that the pressure component of the drag predominates. This was in fact found to be the case, as the viscous component was in general several orders of magnitude smaller.

Plotted below in figure 4.48 are the ratios of the drag for various stably stratified cases to that of the neutral case, against time. This drag appears to increase monotonically with increasing stratification over the range considered, but the behaviour of the drag with time is not monotonic. The $F_h = 0.6$ and $0.7$ cases, in which wave breaking appears, appear to show (strongly damped) oscillating drag (although integration to higher $t$ would be really be required to show this in the $0.6$ case), while the $F_h 0.8$ case does not.

This may simply indicate that the oscillation period for $F_h = 0.8$ is so long that the simulation time is less than half of an oscillation. Certainly, it might be expected that the frequency of such oscillations would be related to the Brunt-Väisälä frequency, and therefore increase with increasing stratification. The relative behaviour of the $F_h = 0.6$ and $0.7$ cases supports this interpretation. However, the available evidence does not exclude the possibility that the drag oscillations are related to the presence of wave breaking, which is absent from the $F_h = 0.8$ simulation. This might be tested simply by integrating the latter to higher $t$.  

147
Figure 4.47: Simulated streamlines for the COS3 Hill, for neutral flow.

Figure 4.48: Wave drag plots at Froude numbers 0.6, 0.7, and 0.8.
4.6 Critical Froude number as a function of Aspect Ratio

Castro and Snyder [17] sketched a Froude number versus obstacle aspect ratio regime diagram showing the envelope within which wave breaking was found to take place in their experiments. This diagram is reproduced in figure 4.49 below. Simulation results are plotted, as well as the experimental curve. The simulation results are mostly consistent with the experimental curve. The fairly good agreement supports hypothesis 7, and thus the supposition that the numerical method and turbulence model used are adequate.

![Figure 4.49: Aspect ratio versus Froude number regime diagram. Solid symbols denote wave breaking, open symbols no wave breaking. Line is the (approximate) critical Froude number found experimentally [17].](image)

However, it is evident that agreement between experimental results and those obtained by numerical simulation becomes poorer for the lowest aspect ratio hill. This might simply be because the COS1 hill lies on a part of the envelope with vertical slope (Castro and Snyder found wave breaking at only a single Froude number), and that this would presumably render it very sensitive to the slightest error. Another possibility is that the decreasing quality of agreement is in fact related to the Froude number and not the aspect ratio; since the quantity plotted in figure 4.49 is a binary one (whether or not there is wave breaking), we might only expect poor agreement at low Froude numbers to show up in it when the aspect ratio became low enough to inhibit wave breaking (at low Froude numbers) in either experiment or simulation.

As noted in section 4.2 above, however, the presence of wave breaking at $F_h = 0.55$ in the COS1 case is somewhat open to question in any case.
4.7 Comparison with the Theory of Smith

This section describes the results of tests carried out to verify hypotheses 1 and 6, which relate to the nonlinear theory of Smith [102]. Figure 4.50 shows the height of the dividing streamline \( \hat{H}_0 \), as discussed in section 2.1 above. The line is the theory of Smith, in which \( \hat{H}_0 \) is given by

\[
\hat{H}_0 = \frac{3\pi}{2} + \hat{d} = \frac{3\pi}{2} - 0.985 + \frac{1}{F_h}.\]  

In other words, \( \hat{H}_0 \) is assumed to be constant with respect to the level of the blocked layer upstream of the obstacle, so that \( \hat{H}_0 \) increases with stability. The solid symbols are the simulation results from the COS3 hill. These were obtained by assuming that Smith’s hypothesis of no disturbance at \( \hat{H}_0 \) is correct. The maximum height attained by the dividing streamline was therefore used to define \( \hat{H}_0 \). It is evident the value of \( \hat{H}_0 \) obtained from the simulations is rather lower than that predicted by theory. This may be due to the fact that the hypothesis that there is no disturbance at \( \hat{H}_0 \) is clearly not correct (see streamline plots at the end of this chapter, and also figure 4.39, showing contours of vertical velocity above \( H_0 \)).

The streamline bounding the recirculating flow originates from a much lower height upstream. The deflection of the upper branch of the dividing streamline implies that the mixed region does not perfectly absorb the wave energy radiated from the lower boundary. Instead, some propagates
As might be expected, some disturbances are also seen aloft, no doubt due to the wave energy that has been transmitted there. Figures 4.53 to 4.55 at the end of this section show vorticity plots of apparently large-scale instabilities near the upper boundary. As the grid there is fairly coarse, it is not believed that these are captured with any accuracy. However, they do appear, at least superficially, to resemble the three-dimensional instabilities observed by Afanasyev and Peltier [1]. Simulations with a much finer grid would be required to properly investigate them, however.

A second possible explanation is that the flows at $F_h = 0.6$ and 0.55 (which deviates from theory much more than that those at higher Froude number) no longer resemble the flow configuration assumed by Smith as closely as the flow at $F_h = 0.7$ does. Castro and Snyder [17] find that this is the case experimentally, particularly for the flow at $F_h = 0.6$, where the turbulent zone due to the breaking wave was observed to merge with the low level rotor below. However, simulation results are less clear. Plots of turbulence kinetic energy (figure 4.45) at the higher Froude number show a structure resembling the mixed region in figure 2.1 above. The resemblance is marginally less clear in the contour plots for $F_h = 0.6$, shown in figure 4.45. There the turbulent region extends to the ground, and downstream of the breaking wave region. Although it is not completely clear whether this configuration corresponds exactly to the 'merged flow' observed in the experiments, it is evident that turbulence is not confined to a well defined region aloft, as assumed by Smith. This may well affect the validity of his theory, since the height of the lower branch of the dividing streamline is now difficult to define. If it is required to bound the mixed region in which $\rho$ is constant, then it will touch the wall. Since Smith obtains the condition that the velocity at $H_0$ equals that on the lower branch (equation 2.98 above), this creates an incompatibility between his theory and the no-slip condition on the wall.

The open symbols in figure 4.50 represent the experimental results of Castro and Snyder [17]. These are slightly higher than the simulation results, but in rather better agreement with them than either is with the theory of Smith. Note that the qualitative trend in the experimentally observed $H_0$ is rather indeterminate at the Froude numbers computed. It is not entirely clear how Castro and Snyder measured $H_0$. Possibly the behaviour of the dye streamers was observed, and $H_0$ inferred on that basis. That would introduce a degree of uncertainty, since the streamers were separated by a finite distance. A degree of judgment would also be involved in the method used in this work - figures 4.56 and 4.57 show how this quantity was determined; the higher of the two dotted lines is $H_0$ in each case. Some subjective judgment is clearly involved, and so the lack of a clear trend can perhaps not be regarded as particularly significant.

It is observed, however, that the simulation results lie slightly further from the theoretical line than do the experiments.

The theory of Smith essentially implies a blocked region upstream of the hill, of height $d$, given
by equation 2.115 above. This has been measured by following streamlines (strictly paths of hypothetical massless particles) that were observed to stagnate on the hilltop back upstream to evaluate their original height. Hence this upstream height is that of fluid that would have just barely failed to surmount the hill.

Results for all hills and Froude numbers are shown in figure 4.51 below.

Figure 4.51: COS3 Hill, Lower Dividing streamline height $\bar{d}$ as a function of $F_h$ and Hill aspect ratio.
Line is the theory of Smith [101]

Theoretical predictions are, for the most part, in very good agreement with simulation results. The exceptions are for the COS1 hill, which is axisymmetric and thus not very wide; it might be expected that a significant fraction of the fluid possessing sufficient energy to surmount the hill instead simply flows around it. Indeed, this does also appear to happen eventually for fluid trapped upstream of the wider hills (see figure 4.34 showing the windward recirculation zone for the COS3 hill at $F_h = 0.7$).

Hypothesis 6 is supported by these results - increasing obstacle ratio leads to increased blocking of the upstream flow.

Castro and Snyder did not present results for $\bar{d}$. However, Paisley and Castro [82] did present such results obtained using mixing-length and $k-l$ turbulence models. These were all lower than or on the theoretical line.
Using the theory of Smith [101], it is also possible to predict the depth of the mixed region \( \delta_C \), by solving equation 2.110 numerically \(^1\). The results are shown in figure 4.52 above. A similar comparison was not made by Castro and Snyder [17], so that comparison with experiment is not possible. However, as discussed above, visual comparison of the simulation results with photographs taken during the experiment is encouraging. There is, however, no clear trend of \( \delta_C \) with respect to \( F_h \).

\(^1\)The method of false position was used for this purpose
Figure 4.53:

Figure 4.54:
Contours of x-vorticity, COS3 Hill, Fh = 0.6 
for x = 15 at t = 35

Figure 4.55:
Figure 4.56: Streamlines, COS3 Hill, \( F_h = 0.6 \).
The higher of the two dashed lines is \( H_q \), the dividing streamline height in Smith's model. The lower is \( H_i \). Their heights are 2.05 and 0.35 respectively.

Figure 4.57: Streamlines, COS3 Hill, \( F_h = 0.7 \).
The higher of the two dashed lines is \( H_q \), the dividing streamline height in Smith's model. The lower is \( H_i \). Their heights are 2.55 and 0.35 respectively.
Variations in three turbulence model parameters were initially investigated, namely the effective values of the constants $C_\mu$, $C_s$, and $\sigma_l$. The results already presented were obtained using the first of these, and need not be discussed further here. The remaining results are tests of hypotheses 2 and 3, which relate to the respective values these two constants should assume in stably stratified flow.

A further modification to the production term of the $\epsilon$ equation was also evaluated, prompted by analysis of the results obtained using a variable $\sigma_l$. This, it was hoped, would impose some global limit on the growth of the turbulence length scale, and is discussed in section 4.8.3 below.

### 4.8.1 The Turbulent Schmidt Number

This modification relied upon the suppression of the turbulent mixing efficiency at high stability, through the model given by equation 2.197. It might be anticipated that this would improve the modelling of stratified turbulence. However, this was not found to be the case - the modification was found, instead, to prevent lee wave overturning in flows where this was expected on the basis of experimental results.

Figures 4.61, 4.62 and 4.63 show streamlines, scalar contours, and contours of $U - \bar{u}'$. No wave breaking is evident, but a great deal of turbulent diffusion is. Figure 4.67 shows contours of turbulence kinetic energy, which may be compared to those of figure 4.45. Clearly, turbulence is not suppressed to a sufficient degree by stratification. This indicates that although $R_{fc} \approx 0.2$ physically [73], this value is not appropriate for the $k-\epsilon$ model without further modifications.

There are very many possible ways in which the turbulence model might be improved. However, investigation of the results obtained using a variable turbulent Schmidt number indicates that one aspect in particular deserves attention. This is the length and time scale associated with the modelled turbulence.

As stated above in section 2.6.1 above, the eddy viscosity is expressed as a function of a velocity scale, as well as either a length or a time scale. Thus, the constitutive relationship for the eddy viscosity in the $k-\epsilon$ model can be written as

$$\nu_t \sim \frac{k^2}{\epsilon} \sim \frac{u_l^2}{\tau}.$$

where the velocity scale is $u$, the length scale $l$ and the time scale $\tau$. Hence we can identify the following relationships:
As mentioned previously in sections 2.6, a clear separation of length and time scales is required between turbulence to be modelled, and unsteady effect to be simulated. In this case, the requirement must be that the turbulent time scale should not become comparable with the inverse of the Brunt-Väisälä frequency; if this should happen, the turbulence model might be expected to smooth out internal waves by treating them as turbulence.

As equation 4.5 above indicates, it is possible to relate the time scale of turbulent motions to the ratio of $k$ and $\epsilon$. Figures 4.59 and 4.60 below show contours of this ratio, for simulations of the COS3 Hill at a Froude number of 0.7, performed using a fixed and a variable turbulent Schmidt number.

It is clear that the turbulent time scale is several orders of magnitude higher when a variable Schmidt number is used. This is not really surprising, since the time scale $\tau$ varies linearly with $k$, and one purpose of the modification is to permit turbulence to exist at higher stability, other things being equal. Hence, it might be expected that $k$ would be higher in general.

Although the magnitude of the effect is clearly substantial, a separation of scales might still be possible. The constant of proportionality in equation 4.5 might be very small. In order to assess if this is so, it is necessary to examine equation 4.2 more closely.

The velocity scale is based on the square root of the turbulence kinetic energy, which is defined as

$$u \sim k^{1/2},$$

$$l \sim k^{3/2}/\epsilon,$$

$$\tau \sim k/\epsilon.$$  \hspace{1cm} (4.5)

As equation 4.5 above indicates, it is possible to relate the time scale of turbulent motions to the ratio of $k$ and $\epsilon$. Figures 4.59 and 4.60 below show contours of this ratio, for simulations of the COS3 Hill at a Froude number of 0.7, performed using a fixed and a variable turbulent Schmidt number.

The constant of proportionality relating $\nu_i$ to $k$ and $\epsilon$ is 0.09, roughly $10^{-1}$; hence it seems reasonable to conclude that the constant of proportionality in equation 4.5 above should be very roughly between $10^{-1}$ and $10^3$.

The period $\tau_w = 2\pi/N$ implied by the Brunt-Väisälä frequency $N$ should give some indication of the timescales of internal wave motion; in the cases considered here, this is roughly of order unity or $10^3$. Hence, if we require that

$$\frac{\tau_w}{\tau} \sim \frac{\epsilon}{kN} \gg 1.$$  \hspace{1cm} (4.7)

then the ratio of $k$ to $\epsilon$ may be no larger than, very approximately, $10^2$, and preferably very much lower.

158
This condition is only barely fulfilled when the constant turbulent Schmidt number is employed. Perhaps critically, the time scale appears to be quite low in the immediate lee of the hill, where wave breaking takes place. Elsewhere, particularly just upstream and far downstream of the hill, it assumes values that might be sufficiently high to violate condition 4.7 above. This is not necessarily a problem; sufficiently far from the hill, internal waves may be absorbed without error, since the intention is to model open boundaries. Such absorption could even be considered desirable.

When a variable turbulent Schmidt number is employed, however, the ratio of $k$ to $\epsilon$ seems to be roughly two orders of magnitude greater than with constant $\sigma_t$. Clearly, adequate separation of scales is then not present.

In physical flows, the turbulence lengthscale is suppressed by stratification, particularly in the vertical direction. This effect is ignored in the model, which may account for its poor performance.

Results obtained using the modified $f_p$ function 2.198 show little improvement (see figures 4.64, and 4.65). It is concluded that merely compensating for the effect of the reduced turbulent Schmidt number in equation 2.193 does not suppress the growth in turbulence lengthscale sufficiently.

It is not concluded that it is inappropriate to modify the turbulent Schmidt number at high $R_f$, however. That it is, in reality, a function of stability is not seriously in doubt [73]. The results do clearly show that its effect on the turbulence model is very strong, much stronger than that of equations 2.193 and 2.198. If the rest of the model were to remain unmodified, a higher value for $R_f$ is clearly required. However, it would be preferable instead to balance the effect by incorporating other effects of stability, currently neglected in the model as presented above.

This could be done by incorporating physical effects, like the suppression of the lengthscale in the vertical direction. However, the hypothesis, advanced above, that the poor performance of the variable Schmidt number can be explained in terms of the growth in the turbulence length and time scales, can to some extent be tested with a much simpler modification to the turbulence time scale. The results are described below in section 4.8.3.

Nevertheless, it is clear that hypothesis 2 (that a constant turbulent Schmidt number is inadequate) is falsified. Results obtained without equation 2.197 have already been shown to be in good agreement with experiment; even if use of equation 2.197 led to improved results, this would not necessarily suffice to show that approximating $\sigma_t$ as a constant was unsatisfactory. As already discussed, the poor results obtained using equation 2.197 do not in themselves indicate that some modification of this kind is inappropriate. However, in the light of the relatively good results obtained without it, it is clear that it is not necessary for the particular flows considered.

Figure 4.58 below shows the velocity profiles obtained using the modified and unmodified models; the use of the modified model is seen to result in significantly enhanced forward shear (horizontal
velocity increasing with height), and the disappearance of flow reversal aloft. Note the very high near-wall velocity obtained using the standard model (this is in fact greatly reduced from a maximum value of about 2.6 before the appearance of turbulence).

Figure 4.58: Horizontal velocity profiles through the wave breaking region, using \( f_\sigma = \text{constant} \), and \( f_\sigma \) given by equation 2.197. COS3 Hill, \( F_h = 0.7 \).

The results are still interesting with regard to atmospheric flows. The experiments took place with zero upstream turbulence and a constant background velocity (no mean shear or particularly thick boundary layer). Clearly, the results obtained using 2.197 are not in good accord with experiment. It is possible, however, that they may be pertinent to the atmospheric boundary layer, in which the gradient Richardson number is reduced by forward shear. Smith [102] states that for uniform shear upstream, a gradient Richardson number below 20 will prevent wave breaking entirely. Although uniform shear is not typically found in the atmosphere, a low Richardson number is likely if the overturning streamline is within the atmospheric boundary layer.

If sufficient shear or turbulence will suppress the jump, and thus the high drag configuration, then it might be possible to explain the rarity of such flows in the atmosphere. Moderate downslope winds are common, but severe storms are rare. Contrary to this, experimental and numerical investigations have often found that such conditions are easily generated [119]. The maximum wind speeds calculated numerically also frequently exceed those observed [21, 98].

We are aware of no numerical investigations that employed anything other than a constant Schmidt or Prandtl number. Most also used a free-slip lower boundary [20, 86, 87, 21, 22, 98, 26, 27, 6]. Although the importance of the lower boundary is believed to be secondary to that of the breaking wave [8, p 331], that of course assumes the wave actually does break. Experimental investigations of low Froude number flow over hills are generally carried out in towing tanks, so that the boundary layer approaching the obstacle is usually of negligible size; the work on which
the current simulations are based is slightly unusual inasmuch as the baseplate on which the models were mounted provided some kind of approaching boundary layer, if perhaps not very substantial.
Figure 4.59: Contours of $k/e$ (assumed to vary as the turbulent time scale), obtained using $f_*$ = constant. COS3 Hill, $F_A = 0.7$.

Figure 4.60: Contours of $k/e$ (assumed to vary as the turbulent time scale), obtained using $f_*$ given by equation 2.197. COS3 Hill, $F_A = 0.7$. 

162
Figure 4.61: Streamlines, COS3 Hill, $F_h=0.7$, $f_*$ given by equation 2.197. Plots are of solutions at multiples of 5 time units, starting at $t=0$. 

163
Figure 4.62: Scalar Contours, COS3 Hill, $F_a=0.7$, $L_c$ given by equation 2.197. Contours are drawn in increments of 0.0 on base of 0. Plotted are solutions separated by 5 nondimensional time units, starting at $t=20$. 
Figure 4.63: Contours of $U - \bar{u}'$, COSS Hill, $P_0=0.7$, $f_*$ given by equation 2.197
Figure 4.64: Streamlines, COS2 Hill, $F_0=0.7, f_r$ given by equation 2.197. $f_r$ given by equation 2.188. Plots are of solutions at multiples of 5 time units, starting at $t=0$. 

166
Figure 4.65: Scalar Contours, CQ3 Hill, $P_r=0.7$, $f_0$ given by equation 2.197, $f_r$ given by equation 2.198. Contours are drawn in increments of 5.0 on base of 0. Plotted are solutions separated by 5 nondimensional time units, starting at $t=20$. 

167
Figure 4.66: Contours of $U - \bar{u}^2$, COS3 HII, $F_h=0.7$, $f_e$ given by equation 2.197, $f_{er}$ given by equation 2.198.
Figure 4.67: Contours of Turbulence Kinetic Energy, COS3 Hill, $F_h=0.7$, given by equation 2.197. Contours are drawn in increments of 0.005 on a base of 0.

Figure 4.68: Contours of Turbulence Kinetic Energy, COS3 Hill, $F_h=0.7$, $f_0$ given by equation 2.197, $f_0$ given by equation 2.198. Contours are drawn in increments of 0.005 on a base of 0.
4.8.2 The $C_{t3}$ Constant

Hypothesis 3 (that $C_{t3}$ should be zero rather than unity for stable stratification), is, however, supported by these results. Figures 4.70 and 4.71, showing streamline and scalar contour plots for $C_{t3} = 1$, are clearly at variance with experimental results for high $t$. Although wave breaking is present initially, this disappears as turbulence develops.

This finding further supports the supposition that increased turbulence can remove the high drag configuration, since the effect of raising $C_{t3}$ is to reduce the level of the dissipation rate of turbulence kinetic energy. This is because in stable flows, the buoyancy production term $G_k$, which is multiplied by $C_{t3}$, is negative.

There is somewhat less evidence of a growth in the turbulence length and time scales than was observed for the variable turbulent Schmidt number. Figure 4.69 at the end of this section shows the ratio of $k$ to $\epsilon$ for the case discussed above; this appears, on the whole, to be intermediate in magnitude between the $C_{t3} = 0$ case and the results obtained using a variable turbulent Schmidt number.

![Figure 4.69: Contours of $k/\epsilon$ (assumed to vary as the turbulent time scale), obtained using $C_{t3} = 1$. COS3 Hill, $Fr_h = 0.7$.](image)

Figure 4.69: Contours of $k/\epsilon$ (assumed to vary as the turbulent time scale), obtained using $C_{t3} = 1$. COS3 Hill, $Fr_h = 0.7$. 

170
Figure 4.70: Streamlines, COS3 Hill, $F_{h}=0.7$, $C_{t}=1$. Plots are at solutions at multiples of 5 time units, starting at t=5.
Figure 4.71: Scalar Contours, COS3 Hill, $F_a=0.7$, $C_{D3}=1$ Contours are drawn in increments of 5.0 on base of 0. Plotted are solutions separated by 5 nondimensional time units, starting at $t=15$. 

172
4.8.3 Limitation of the Turbulence Time Scale

In order to prevent interference of the turbulence model with the internal wave motion, it was considered desirable to prevent the growth of the turbulence time scales approximated by it from growing to include the period defined by the Brunt-Väisälä frequency as follows:

\[ \tau_w = \frac{2\pi}{N} \]  

(4.8)

It was found, in section 4.8.1 above, that, provided the ratio of \( k \) to \( \epsilon \) remained about two orders of magnitude lower than this period, satisfactory results could still be obtained. Accordingly, a modification to effect such a limitation was implemented in the \( k-\epsilon \) model. This was done in essentially the same way as the length scale limitation proposed by Apsley and Castro [4]. For a limitation on the time rather than the length scale, the modification has a particularly simple form. In the original model, the production term for the production of \( \epsilon \) is:

\[ P_\epsilon = C_{e1}P_k + C_{e2}G_k \quad \text{where} \quad P_k \text{ and } G_k \text{ are the production of turbulence kinetic energy by shear and buoyancy respectively. Identifying } k/\epsilon \text{ as a turbulence time scale } \tau, \text{ this may be written as} \]

\[ P_\epsilon = C_{e1}P_k + C_{e2}G_k \frac{C_{e}}{\tau} \]

This may be modified to prevent excessive growth of \( \tau \), as follows:

\[ P_\epsilon = \frac{C_{e1}P_k + C_{e2}G_k \frac{C_{e}}{\tau}}{\tau} + \frac{(C_{e2} - 1)(C_{e1}P_k - C_{e2}G_k)}{\tau_0} \]

(4.11)

This modification has negligible effect when \( \tau_0 \gg \tau \); when \( \tau_0 = \tau \), the production of \( \epsilon \) becomes, in the case of local equilibrium, equal to its dissipation term \( (C_{e2}P_k/\tau = C_{e2}/\tau) \). The presence of the term in \( \tau_0 \) tends to act to prevent a further fall in \( \epsilon \) for \( \tau \gg \tau_0 \). This is because it introduces an element of negative feedback. Note that the production terms \( P_k \) and \( G_k \) in equation 4.11 contain the eddy viscosity, which varies linearly with \( \tau \), so that for a given velocity and density field, the \( \epsilon \) production term varies as

\[ P_\epsilon \sim k \]

(4.12)

for the unmodified \( k-\epsilon \) model, and as

\[ P_\epsilon \sim k \left( 1 + (C_{e2} - 1) \frac{\tau}{\tau_0} \right) \]

(4.13)

for the modified model. Increasing \( \tau \), therefore, increases the \( \epsilon \) production term, and this in turn suppresses \( \tau \) by increasing \( \epsilon \) and also by (indirectly) decreasing \( k \).

Some value needs to be set for \( \tau_0 \); on the basis of previous results described in section 4.8.1 above, the value 100\( \tau_w \) was chosen. Ideally, \( \tau_w \) should be calculated on the basis of local conditions.
However, this was found to result in numerical instability, and a fixed value was used instead, based on the global value of $N$.

Several simulations were carried out using this modified model. One used the otherwise unmodified $k-\varepsilon$ model, and two used the variable turbulent Schmidt number modification described above. All simulations were of the COS3 Hill at a Froude number of 0.7. The simulation using a constant turbulent Schmidt number was carried out at a Reynolds number of $10^5$; those using the variable Schmidt number at $Re = 10^4$ and $10^5$.

The results for the constant Schmidt number did not differ noticeably from previous results obtained without the modification to the $\varepsilon$ production term.

The results obtained for the variable turbulent Schmidt number, however, are very different from those obtained previously without limiting the turbulent time scale. Instead, they closely resemble those obtained with the constant turbulent Schmidt number, and are in reasonably good accord with experimental results.

Figures 4.72 and 4.73 below show streamline plots at $t = 45$ for the low and high Reynolds number cases. They may be compared with figures 4.29 and 5.24. That these improved results were indeed accompanied by a decrease in the turbulent time scale is shown by figure 4.74.
Figure 4.72: Simulated streamlines for the COS3 Hill, at a Froude number of 0.7 and a Reynolds number of $10^4$. These results are from $t=45$, and were obtained using a variable turbulent Schmidt number ($f_s$ given by equation 2.197) and the limited turbulence time scale modification.

Figure 4.73: Simulated streamlines for the COS3 Hill, at a Froude number of 0.7 and a Reynolds number of $10^5$. These results are from $t=45$, and were obtained using a variable turbulent Schmidt number ($f_s$ given by equation 2.197) and the limited turbulence time scale modification.
Figure 4.74: Contours of $k/\epsilon$ (assumed to vary as the turbulent time scale), obtained using a variable turbulent Schmidt number ($f_\alpha$ given by equation 2.197) and the limited turbulence time scale modification. COS3 Hill, $F_h = 0.7$, $Re=10^5$. 
4.9 Nonlinear Stratification

Hypothesis 4 states that linear stratification may be represented approximately by piecewise constant stratification. This hypothesis was tested by replacing the linear stratification profile used elsewhere in this work by stepwise profiles of the same mean gradient as the linear stratification. It is hoped that insight may be gained into the importance of perturbations from linearity in the background density profile.

Three different profiles were considered. In each case, the density-determining scalar \( \theta \) was kept constant within a layer of thickness \( \lambda \), to be succeeded by another such layer above it, and so on. In each layer, the value of \( \theta \) was set so that the mean gradient in \( z \) would be identical to that due to the linear profile. Values of 0.5, 0.25, and 0.125 hill heights were used for \( \lambda \). The case chosen for simulation was the COS3 hill at a Froude number of 0.7.

The results are shown in figures 4.75, 4.76 and 4.77 in terms of contours of \( \theta \). The 'coarse' profile corresponds to \( \lambda = 0.5 \), 'medium' to \( \lambda = 0.25 \), and 'fine' to \( \lambda = 0.125 \).

Comparison of these results with those for linear stratification indicates that the state at high time resembles the linear result increasingly closely with decreasing \( \lambda \). Note that with the possible exception of the coarsest profile, all results reproduce the essential features of the flow reasonably well. This observation appears to indicate that the critical Froude number is not greatly affected by small scale structure in the density field.

Additional simulations using a nonlinearly stratified fluid (weak quadratic departures from linearity, see figure 4.78 at the end of this section) were undertaken for the COS3 hill at a Froude number of 0.75, at which combination of parameters wave breaking is marginally present with linear stratification. Hence it was hoped that any effect due to the nonlinear stratification would be relatively obvious.

This was in fact the case. The results obtained with a density scalar \( \theta \) profile such that \( \partial^2 \theta / \partial z^2 < 0 \) showed no wave breaking, while that with \( \partial^2 \theta / \partial z^2 > 0 \) did (see figures 4.80 and 4.79 at the end of this section). This would appear to indicate that stratification increasing in strength with height does not have the same effect as a linear profile with the same mean density gradient.

Although the average of \( \partial^2 \theta / \partial z \) is the same over the whole of the domain depth, all the fluid involved in the wave overturning events is confined to the lower half of the domain. Averaged only over the domain half-depth, the mean stratification of the three cases is of course quite different, the cases with zero and positive second derivatives being more strongly stratified. Hence this result is perhaps not very surprising.

The appearance of the flow in the lee for the \( \partial^2 \theta / \partial z^2 < 0 \) case, with a strikingly discontinuous
jump, does, however, resemble the more strongly stratified cases (such as $F_h = 0.55$, see figure 4.32), and arguably the kind of 'hydraulic jump' seen in shallow water flow over topography (which, of course, implies $\partial^2 \theta / \partial z^2 \to -\infty$, locally).

Further investigations with a shallow layer of more stratified fluid near the surface, or with an inversions aloft, might be more informative. A layer of increased stratification (large $\partial \theta / \partial z$) might be inserted into the $\theta$ profile at various $z$ and the results examined.
Figure 4.75: Scalar Contours for the Coarse Stepwise Scalar Profile, COS3 Hill, $F_h=0.7$. Contours are drawn in increments of 5.0 on base of 0. Plotted are solutions separated by 5 nondimensional time units, starting at $t=20$. 
Figure 4.76: Scalar Contours for the Medium Stepwise Scalar Profile, COS3 Hill, $F_{ex}=0.7$. Contours are drawn in increments of 0.5 on base of 0. Plotted are solutions separated by 5 nondimensional time units, starting at $t=20$. 
Figure 4.77: Scalar Contours for the Fine Stepwise Scalar Profile, COS3 Hill, $P_s=0.7$. Contours are drawn in increments of 5.0 on base of 0. Plotted are solutions separated by 5 nondimensional time units, starting at $t=20$. 

181
Figure 4.78: Initial density scalar profiles for linear and nonlinear continuous stratification.
Figure 4.79: Simulated streamlines for the COS3 Hill, at a Froude number of 0.75. Stratification increasing with height, $\partial^2\theta/\partial z^2 < 0$. These results are from $t=45$.

Figure 4.80: Simulated streamlines for the COS3 Hill, at a Froude number of 0.75. Stratification decreasing with height, $\partial^2\theta/\partial z^2 > 0$. These results are from $t=45$. 
4.10 Influence of Domain Extent

We shall now attempt to determine whether the size of the domain used was adequate. Typically, such discussions relate only to the adequacy of a numerical simulation; however, in this case, the influence (or rather the lack thereof) of the location of boundaries is also of some physical interest, and has therefore been treated as a hypothesis to be tested (hypothesis 5).

This was done by performing three additional simulations for the COS3 hill at a Froude number of 0.7. The most critical issue is whether or not the length of the domain was adequate, since the simulation differs from the experimental work of Castro and Snyder [17] in this regard. If effects are detected due to the spanwise width or vertical depth of the domain, however, these would likely carry over to the experimental work also, since the depth and width used in the simulations was the same. The dimensions of the extended grids are shown in the table below, along with those of the standard grid for comparison.

<table>
<thead>
<tr>
<th>Standard</th>
<th>Long Domain</th>
<th>Long and Wide Domain</th>
<th>Long, Wide, and Deep Domain</th>
</tr>
</thead>
<tbody>
<tr>
<td>High x</td>
<td>50</td>
<td>75</td>
<td>75</td>
</tr>
<tr>
<td>Low x</td>
<td>-100</td>
<td>-150</td>
<td>-150</td>
</tr>
<tr>
<td>High y</td>
<td>12</td>
<td>12</td>
<td>18</td>
</tr>
<tr>
<td>Low y</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>High z</td>
<td>10</td>
<td>10</td>
<td>10</td>
</tr>
<tr>
<td>Low z</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

Results from the domain extended in the streamwise direction only are shown in figures 4.86 and 4.87. The corresponding results obtained using the standard domain are shown in figures 4.43 and 4.44. It is obvious that differences are visible in streamline plots at all time steps.

The differences appear immediately - discrepancies are visible in the very first plots, from t=5. This cannot be due to wave reflection from the boundaries, since the maximum group velocity of a columnar mode in this domain is limited to

\[ c_g = \frac{ND}{\pi U} \tag{4.14} \]

and waves of larger wavenumber are slower (see figure 2.9. Note that in an infinite domain, the maximum group velocity would be infinite (as \( D = \infty \), see also [26]). Even for waves propagating downstream, the maximum velocity is

\[ \frac{ND}{\pi U} + U \approx 5.5. \tag{4.15} \]

At this velocity, a wave would reach the downstream boundary at \( t=9 \); it would take until about \( t=20 \) for it to propagate back up again. It has been shown above in section 2.3 that reflections from the upstream boundary would take about 44 time units to arrive. Hence it is not possible for internal wave reflections from either boundary to cause the discrepancy at \( t=5 \).
An alternative explanation is that the 'squashing' phenomenon is involved, rather than wave reflection. This might not require a finite time to appear, since it is a consequence of continuity rather than wave radiation. The relevant wave speed to consider is then that of sound, which, given that the fluid is incompressible, is infinite.

The interpretation would be that the stratification within the 'slab' of fluid upstream of the obstacle is distorted by its motion; since the fluid upstream is compressed by the motion of the obstacle, an incompressible fluid would be forced to pass over or around the obstacle. If the latter is not possible, then the fluid forced out of the way must be at approximately the same height as the obstacle, as fluid from lower down possesses insufficient energy to rise over the obstacle. This would distort the stratification upstream of the obstacle, since the space upstream of the obstacle is now occupied by fluid that had earlier occupied a smaller vertical extent.

Figure 4.83 which shows the profiles of the density-determining scalar upstream of the obstacle (in the standard domain), shows that the stratification is substantially unaltered at $t=5$. This is not quite the case at $t=40$; we may conclude that the distortion of the scalar profile at this later time might represent limited evidence of squashing, but that this phenomenon does not account for the discrepancy at $t=5$, and that its effects, if present, do not propagate at infinite speed. The distortion seen at $t=40$ is in fact likely to be due to columnar modes; the appearance of the scalar perturbation profile shown in figure 4.82 is consistent with this hypothesis.

Figures 4.84 and 4.85 show contours of the pressure at $t=5$ for lengthwise extended and standard domains. The solution from the extended domain shows that the pressure field of the obstacle is limited to approximately $-10 < z < 10$. However, the plots for the standard domain show that the influence of the obstacle extends all the way to the inflow boundary. The pressures far up and downstream of the obstacle are roughly constant (with respect to all three coordinate directions) in both cases, but assume very different values. The following table shows the up and downstream pressures at $t=5$, for both cases.

<table>
<thead>
<tr>
<th>Standard Domain</th>
<th>Long Domain</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pressure at $x=-100$, $y=0$, $z=0$</td>
<td>0.237</td>
</tr>
<tr>
<td>Pressure at $x=50$, $y=0$, $z=0$</td>
<td>0.172</td>
</tr>
</tbody>
</table>

On the basis of these results, the discrepancy between the standard and extended domains is attributed to pressure reflection from the upstream boundary, leading to an increase in the pressure gradient across the obstacle. This is more severe for the standard domain.

The question arises whether this is a transient phenomenon due to the impulsive start, or whether it persists. Streamline plots 4.43 and 4.86 show differences between the solutions at all time steps. However, except for the last plots, at $t=50$, the results appear to be becoming rather similar with increasing $t$. As we have already shown, by $t=50$, internal waves reflected from the upstream boundary are expected to have returned to the vicinity of the obstacle. No such reflections are
expected at this time in the extended domain; there, the first columnar modes are expected at around $t=65$. This might account for the differences seen in the last plots at $t=50$, which are rather greater than those from $t=40$ or 45.

However, visual examination of streamline plots is not an exact science. Figure 4.81 below shows the height of the dividing streamline $H_0$, which is treated as an integral measure of the development of the flow. The standard solution undergoes a turbulent transition at around $t=25$, in which the stagnation zone aloft temporarily collapses, then reforms. Thereafter, $H_0$ remains relatively constant, although integration to much higher $t$ would be required to show that very low frequency unsteadiness is absent. The results from the extended domain appear to lag by about 10 nondimensional time units; a similar transition appears to take place at around $t=35$. Despite the different behaviour of the two solutions with respect to time, the value of $H_0$ converged upon is apparently much the same.

Figure 4.81:
The dividing streamline height $H_0$ plotted as a function of time for both the standard and lengthwise extended domains.

It is therefore suggested that the use of a domain extending 100 hill heights upstream of the hill causes the development of the flow to be accelerated artificially. The result is some anomalous transient behaviour. It is nonetheless believed that the gross features of the flow are not materially affected, provided steady state conditions have been attained. This conclusion is conditional upon the attainment of steady state conditions at 40-50 nondimensional time units. It should be said that the present results do not entirely suffice to prove that this is the case.

At later times, columnar modes appear in the solution upstream of the obstacle. These, however, are apparently weaker than those found by Paisley and Castro [82], who simulated the same flows using mixing length and $k - l$ turbulence models. No clear evidence of columnar modes could be found in the velocity field. This is likely to be in part because of the relative weakness of these
modes, but also, at later times, due to the fact that the wave interferes with itself after reflection. This interference is destructive in the velocity field [8, p 180], so that the disturbances cancel, but is constructive in the density field. Figure 4.82 shows the perturbation of the density determining scalar $\theta$ from its initial value, at a Froude number of 0.7.

Figure 4.82: Scalar perturbation profile upstream of the COS3 Hill, $F_r=0.7$

The results obtained using the domain extended in both spanwise and streamwise directions are substantially identical to those from the lengthened domain discussed above. A third domain was extended in all three directions; this also produced identical results. This appears to indicate that, somewhat surprisingly, the width and depth of the domain do not materially affect the flow. Results are shown in the form of streamlines and scalar contours in figures 4.88, 4.89, 4.90 and 4.91. It should be noted that the computations in the extended domains were not continued long enough to permit reflected internal waves to return to the vicinity of the obstacle.

It is concluded that hypothesis 5 (that the standard domain size is adequate) is supported by these results only conditionally. It appears that the flow approaches the same steady state, whether or not an extended domain is used, but it is not the case that the transient behaviour is the same. It does appear that hypothesis 5 is correct for the larger domain, however.
Figure 4.83: Scalar Profiles for the Standard Domain, COS3 Hill, $F_h=0.7$, at $t=5$ and $t=40$

Figure 4.84: Pressure Contours for the Standard Domain, COS3 Hill, $F_h=0.7$, at $t=5$.
One contour every 0.1 pressure units.

Figure 4.85: Pressure Contours for the Long Domain, COS3 Hill, $F_h=0.7$, at $t=5$.
One contour every 0.1 pressure units.
Figure 4.86: Streamlines for Long Domain, COS3 Hill, $F_a=0.7$. Plots are of solutions at multiples of 5 time units, starting at $t=5$. 
Figure 4.87: Scalar Contours for Long Domain, COS3 Hill, $F_h=0.7$ Contours are drawn in increments of 5.0 on base of 0. Plotted are solutions separated by 5 nondimensional time units, starting at $t=10$. 
Figure 4.88: Streamlines for Long and Wide Domain, COS3 III, $P_h = 0.7$. Plots are of solutions at multiples of 3 time units, starting at $t=5$. 
Figure 4.89: Scalar Contours for Long and Wide Domain, COS3 Hill, \( F_a = 0.7 \) Contours are drawn in increments of 5.0 on base of 0. Plotted are solutions separated by 5 nondimensional time units, starting at \( t = 10 \).
Figure 4.90: Streamlines for the Long, Wide and Deep Domain, COS3 Hill, $F_h=0.7$. Plots are of solutions at multiples of 5 time units, starting at $t=5$. 
Figure 4.91: Scalar Contours for Long, Wide and Deep Domain, COS3 Hill, $F_x=0.7$. Contours are drawn in increments of 5.0 on base of 0. Plotted are solutions separated by 5 nondimensional time units, starting at $t=10$. 

194
4.11 Effect of Mid-Channel Symmetry Condition

This section describes tests carried out to determine the effect, if any, of enforcing symmetry in the spanwise (y) direction of the flow. It is therefore relevant to hypothesis 9.

Previous numerical studies of the experiments simulated here have employed a symmetry condition along the geometrical symmetry plane (y = 0) of the channel [81, 80, 82]. However, recent experimental work (Vosper et al, see [116]) has thrown the appropriateness of this approach into question. Strong vortex shedding was observed for Froude numbers of 0.4 and below; although no Froude numbers this low have been investigated here, weaker effects of this kind may also be present at slightly higher Froude numbers. In order to investigate the validity of the symmetry condition for numerical simulations, some simulations including the full width of the channel were carried out.

All simulations were carried out using (twice) the medium grid described above. All three hills were investigated, at a Froude number of 0.6, which is towards the lower end of the range investigated; it was expected that low Froude numbers would yield stronger asymmetrical effects.

A second set of simulations was carried out with initial conditions perturbed slightly from symmetry; the $U$ velocity component was varied linearly, by 0.2% across the channel width. One last simulation was carried out for the COS3 Hill only, with a perturbation of 2%.

It was found that simulating the full width of the domain had little effect on the results for any of the hills. No von Kármán vortex streets were observed for any hill, for instance, and only slight asymmetry was detectable in the results. The centreline solution appeared not greatly affected in all cases. Figures 4.104 to 4.104 show streamline plots on $y = 0$; these may be compared with figures 4.15, 4.20, and 4.27 above. Agreement, while not exact, is reasonable; the size of the mixed region aloft is not much changed.

The way in which the hill aspect ratio influenced the symmetry of the flow was somewhat surprising. It had been anticipated that the axisymmetric COS1 hill would show the greatest effect, and the wide COS3 hill the weakest. This was because the width of the latter might permit vortices shed from either (spanwise) end to remain unaffected by each other; this would not be possible in the axisymmetric case, where an alternating von Kármán vortex street might be expected. Contrary to these expectations, the simulations of the COS1 hill actually showed not only less asymmetric behaviour than the others, but virtually none at all.

Perturbations to the initial conditions influenced the results only slightly; even a variation in $U$ of 2% across the channel, on the COS3 Hill case, did not greatly affect the degree of symmetry. Slight differences were evident in the immediate lee of the hill, but these did not extend into the mixed turbulent region further downstream.
Figures 4.92 to 4.97 below show isosurfaces of z-vorticity for all three hills. Figures 4.98 to 4.101 show contours of x-momentum on various slices through the computational domain, all at constant z. These illustrate the case showing the strongest evidence of asymmetrical flow. Vector plots at two values of z are also shown, in figures 4.102 and 4.103. Although no asymmetrical vortex-shedding was observed experimentally at $F_h = 0.6$, the simulated asymmetry is rather weak; the question of whether the simulated flow has any physical counterpart is probably open, particularly since the obstacles used in the experiments were of different shapes, and the Reynolds number might also have been different.

It is concluded that the turbulence model used is sufficiently dissipative to remove all turbulence from the momentum solution, even very large scale vortices. Since it is a Reynolds-averaged model, this is not surprising - such models are derived (see section 2.6 above) by explicitly filtering out all length and time scales due to turbulent unsteadiness. These results merely demonstrate that the filtering remains thorough enough to do this, in spite of the turbulence-suppressing modifications that have been added to improve the modelling of stratified flow.

However, what asymmetry was observed was rather weak in all cases, and thus difficult to interpret, as well as indicating only a slight influence due to the symmetry condition with the current model. However, it might be worthwhile to make note of a few observations on what asymmetry is observable in the results on the wider hills.

The fact that some asymmetry was detectable in the results for the wider hills, but not the axisymmetric case, might most easily be explained by reference to effects that are present on the wider hills but weak or absent for COSI; these include wave breaking and blocking. The former seems a possible explanation, as the strong stratification and acceleration of the flow in the immediate lee of the hill suppresses the eddy viscosity there. This might permit some small numerical errors to amplify by stimulating physical instabilities, without being suppressed by the turbulence model. Indeed, it appeared that departures from symmetry were strongest at or near the interface between the high-speed flow in the lee of the hill and the highly turbulent, mixed, and stagnant zone downstream of it (the ‘hydraulic jump’, if that analogy is accepted). Downstream of the overturning region, asymmetrical flow features are visibly damped, until, sufficiently far downstream, none are apparent.

Unfortunately, while these results are very much consistent with the behaviour to be expected of a Reynolds-averaged turbulence model, they also indicate that such a model is possibly fundamentally unsuitable for strongly stratified flows. In view of the experimental evidence for vortex shedding at $F_h \leq 0.4$, it is clear that these flows contain turbulent flow at lengthscales comparable to internal waves. There would therefore be a scale separation problem; the unsteady time scales of the turbulence to be modelled would not necessarily be well separated from those of the lee waves that should be resolved.

At the Froude numbers examined, no clear evidence was found for significant flow asymmetry;
however, what asymmetry there was appeared to be damped by the turbulence model. It is possible that the simulation might otherwise predict vortex shedding, but none was found experimentally at these Froude numbers [116].

It is to be expected that this effect would manifest itself more strongly at low Froude numbers, for two reasons. Firstly, the Brunt-Väisälä frequency $N$ increases with stability, so that the possible time scales of internal wave motion are reduced further, towards the domain of turbulence. Secondly, experiments [116] indicate that the large vortices shed in von Kármán streets in the x-y plane begin to appear at Froude numbers of 0.4 and below. Although large vortices might be present in any plane, we would expect turbulent motions in z to be suppressed by the stratification, so that the von Kármán vortices are possibly those with the longest length and time scales in these flows. Vosper et al also found the frequency of vortex shedding to be lower than the Brunt-Väisälä frequency, so that it appears that a turbulence model that models the former correctly will probably fail to yield correct results for internal wave motion.

It is concluded that results obtained using the symmetry condition differ little from those obtained without it, and that the use of this condition is therefore appropriate whenever use of the numerical model (particularly the turbulence model) is also appropriate. Results obtained at low Froude numbers should, however, be regarded with caution. It is probably not appropriate to use this model to simulate flow at Froude numbers of 0.4 and below.

Instead, it might be appropriate to use a large eddy simulation, or a length-scale limited turbulence model, such as that proposed by Apeley and Castro [4]. These workers modified the $\epsilon$ production term of the standard $k-\epsilon$ model so as to cancel the dissipation term when the calculated mixing length ($\sim k^{3/2}/\epsilon$) became equal to some fixed value; for calculated mixing lengths exceeding this, the modification exerted some negative feedback by increasing the effective production of $\epsilon$. The modification was made with a view to better prediction of atmospheric boundary layers, but might have application for the flows in this work, particularly at low Froude numbers. There, it is expected that a limiting length scale would be imposed by the requirement that the scales of turbulence treated using the turbulence model be separated from those involving internal waves.

Some calculations obtained using a modification of this kind are described in section 4.8.1 above; these, however, do not include the full width of the channel.
Figure 4.92: Isosurfaces of z-velocity, COS1 Hill, $F_h = 0.6$

Figure 4.93: Isosurfaces of z-velocity, COS1 Hill, $F_h = 0.6$
Isosurfaces of z-vorticity, COS2 Hill, Fh = 0.6

Figure 4.94:

Isosurfaces of z-vorticity, COS2 Hill, Fh = 0.6

Figure 4.95:
Figure 4.96: Isosurfaces of z-vorticity, COS3 Hill, Fh = 0.6

Figure 4.97: Isosurfaces of z-vorticity, COS3 Hill, Fh = 0.6
Figure 4.98: Full channel width solution for the COS3 hill at $F_h = 0.6$.

Figure 4.99: Full channel width solution for the COS3 hill at $F_h = 0.6$. 

201
Figure 4.100: Full channel width solution for the COS3 hill at $F_h = 0.6$.

Figure 4.101: Full channel width solution for the COS3 hill at $F_h = 0.6$. 
Figure 4.102: Vectors on the $z=0.1$ plane. Colour is $x$-direction velocity component.

Figure 4.103: Vectors on the $z=1.0$ plane. Colour is $x$-direction velocity component.
Figure 4.104: Streamlines, full channel width results for the COS1 hill at a Froude number of 0.6.

Figure 4.105: Streamlines, full channel width results for the COS2 hill at a Froude number of 0.6.
Figure 4.106: Streamlines, full channel width results for the COS3 hill at a Froude number of 0.6.
5. Discussion

5.1 Grid and Time Step Dependence

The first issue that shall be addressed is whether the grid used resolved the flow sufficiently well. This was initially tested by running a simulation of the COS3 hill at a Froude number of 0.7 at a rather higher resolution, and comparing the results. Note that the high resolution simulation was not carried out with the standard turbulence model, but instead with the constant $C_d$ set to unity.\(^1\) Comparison is therefore made with the corresponding simulations at lower resolutions.

It was believed that grid independence for this particular flow likely implies grid independence for other hills, Froude numbers, and turbulence model modifications. It was not believed that any of the other cases (in particular, the narrower hills) represent more challenging problems from the point of view of grid resolution than the one considered. Nonetheless, further tests were deemed desirable, and an additional simulation was later carried out, which is described in detail below.

Figures 5.1, 5.2, 5.3 and 5.4 at the end of this chapter show solutions obtained using all three grids at four different time steps. Scalar contours were chosen instead of streamlines, as the latter need to be calculated by a procedure (such as the tracking of massless particles) that might add its own grid dependence, whereas contours require only interpolation. Agreement between all three is seen to be good. At $t = 20$, however, the low resolution solution ceases to be in good agreement with the finer grids, but there are no particularly significant differences between medium and high resolution solutions.

It was believed that the results obtained on the medium grid display an acceptable level of grid independence, and that this likely applies to other solutions (with different hill aspect ratios, Froude and Reynolds numbers, etc.) also.

In order to test this belief, a further high-resolution simulation was later carried out of the COS3 Hill case, with $F_r = 0.7$ and $Re = 10^5$, using a value of zero for $C_d$. The level of grid dependence was found to be less encouraging.

Results are shown in figures 5.6 to 5.15, in the form of density scalar contours at various time steps, for the high and medium resolution grid solutions. The initial development of the flow appears sufficiently well resolved in the medium resolution simulation, but differences appear at $t = 25$. This time corresponds to the beginning of the transition of the flow to turbulence; prior to it, the flow is essentially laminar. After $t = 35$, however, the two simulations again come into somewhat better agreement, at least with regard to the mixed region aloft in the immediate lee.

\(^1\)This was not intentional, but the difference was not thought to be critically important.
of the hill. The rotors downstream near the surface are apparently only well resolved in the high resolution case, however.

It appears that although the initial development of the flow is not grid dependent, and the solutions for sufficiently high $t$ similar (although integration to greater values still would be required to show this beyond any doubt), the period of transition from the laminar to turbulent flow may not be adequately resolved.

This is probably because the initial stages of the transition to turbulence occur without the involvement of the turbulence model, so that the physical instabilities are simulated directly. It certainly is the case that wave overturning occurs while the turbulence kinetic energy is still approximately zero; turbulence only develops after this point.

This explanation is consistent with the observation that the results obtained using a value of unity for $C_{e3}$ are apparently grid independent. Using a value of zero for this constant damps turbulence and would be expected to cause the turbulence model to remain quiescent for longer. This might well cause some of the initial turbulent breakdown to be simulated directly, for which purpose the grids used are certainly not adequate.

Some encouragement can be drawn from the fact that the maximum drag observed was similar for both the high and the low resolution grids (4.10 and 4.30 respectively, normalized with the drag for neutral flow), although at different times. However, it should be noted that this value was only reached on the fine grid at the end of the simulation, so might well have risen further were the calculation continued. Further, the rotors near the surface downstream of the high wave-breaking zone are clearly significantly better resolved on the high resolution grid.

The grid dependence of these results is therefore still open to some question. The time accuracy of the result, in particular, appears doubtful (see section 4.10 for a similar example of the turbulent transition occurring at different points in time), although the results appear not to be very sensitive to the size of the time step (see below in this section). The second of the tests described above was regrettably the very last for which results were obtained, so that no time was available for further testing. Such would certainly be desirable.

As noted above, a symmetry condition was used to represent the top boundary. A moving wall could also have been used, but the velocity then has to be prescribed in advance, and the logarithmic law used for turbulent flow. Both boundary conditions ensure that the normal velocity is zero, which prevents any waves leaving the domain. In order to test whether the choice of boundary condition is important, the COS3 hill case at $F_h = 0.7$ was simulated using both. The velocity of the wall boundary was assumed equal to the inflow velocity.

The results are shown in figures 5.16, for the wall boundary condition and 5.17, for symmetry. No differences are evident. Note that these simulations were performed using the low resolution grid,
owing to time constraints. However, the results are believed to be applicable to other resolutions.

In order to investigate the adequacy of the time resolution, a simulation of the COS3 Hill at a Froude number of 0.7 was performed with twice the time step size used in all other simulations described here. The flow eventually converged to seemed not to be too greatly affected in view of the large difference in the time step, but some effect certainly was evident. Figure 5.18 shows a streamline plot at $t = 40$, which may be compared to the equivalent figure 5.19 at normal time resolution.
Figure 5.1: Scalar contours on Low, Medium, and High Resolution Grids, COS3 Hill, $Fr=0.7$, at $t=15$. Results obtained with $C_{e3} = 1$. 
Figure 5.2: Scalar Contours on Low, Medium, and High Resolution Grids, COS3 Hill, Fr=0.7, at t=20 Results obtained with $C_{s,5} = 1$. 

210
Figure 5.3: Scalar Contours on Low, Medium, and High Resolution Grids, COS3 Hill, Fr=0.7, at t=25. Results obtained with $C_{a3} = 1$. 

211
Figure 5.4: Scalar Contours on Low, Medium, and High Resolution Grids, COS3 Hill, Fr=0.7, at t=30. Results obtained with C_{e3} = 1.
Figure 5.5: Streamlines, COS3 Hill, $F_h=0.7$, High Resolution Grid. Plots are of solutions at multiples of 1 time unit, starting at $t=1$. Results obtained with $C_{u3} = 1$. 
Figure 5.6: Scalar Contours on Low, Medium, and High Resolution Grids, COS3 Hill, Fr=0.7, at t=5.
Results obtained with $C_{f3}=0$, $Re=10^5$. 

214
Figure 5.7: Scalar Countours on Low, Medium, and High Resolution Grids, COS3 Hill, $F_r=0.7$, at $t=10$. Results obtained with $C_{\alpha}=0$, $Re=10^5$. 

215
Figure 5.8: Scalar Contours on Low, Medium, and High Resolution Grids, COS3 Hill, $F_r=0.7$, at $t=15$. Results obtained with $C_{e3}=0$, $Re=10^5$. 
Figure 5.9: Scalar Countours on Low, Medium, and High Resolution Grids, COS3 Hill, $F_r=0.7$, at $t=20$. Results obtained with $C_{s3} = 0$, $Re = 10^5$. 

217
Figure 5.10: Scalar Contours on Low, Medium, and High Resolution Grids, COS3 Hill, Fr=0.7, at t = 25. Results obtained with C_{c3} = 0, Re = 10^5.
Figure 5.11: Scalar Countours on Low, Medium, and High Resolution Grids, COS3 Hill, Fr=0.7, at $t = 30$. Results obtained with $C_{x3} = 0$, $Re = 10^5$. 

219
Figure 5.12: Scalar Countours on Low, Medium, and High Resolution Grids, COS3 Hill, $Fr=0.7$, at $t=35$. Results obtained with $C_{e3}=0$, $Re=10^5$. 

220
Figure 5.13: Scalar Contours on Low, Medium, and High Resolution Grids, COS3 Hill, $Fr=0.7$, at $t=40$. Results obtained with $C_{v3}=0$, $Re=10^6$. 
Figure 5.14: Scalar Contours on Low, Medium, and High Resolution Grids, COS3 Hill, $F_r=0.7$, at $t=45$. Results obtained with $C_{13}=0$, $Re=10^5$. 

222
Figure 5.15: Scalar Contours on Low, Medium, and High Resolution Grids, COS3 Hill, Fr=0.7, at t=50. Results obtained with $G_{13}=0$, $Re=10^3$. 

223
Figure 5.16: Streamlines for the low resolution grid, COS3 Hill, $F_L=0.7$. Plots are of solutions at multiples of 5 time units, starting at $t=0$. These results were obtained using a wall boundary condition on the upper boundary.
Figure 5.17: Streamlines for the low resolution grid, COS3 Hill, $f_x=0.7$. Plots are of solutions at multiples of 5 time units, starting at $t=5$. These results were obtained using a symmetry condition on the upper boundary.
Figure 5.18: Simulated streamlines for the COS3 Hill at $F_h = 0.7$. These results are from $t = 40$. Results obtained using $\Delta t = 0.2$.

Figure 5.19: Simulated streamlines for the COS3 Hill at $F_h = 0.7$. These results are from $t = 40$. Results obtained using $\Delta t = 0.1$. 
5.2 Effects of Wall Function Treatment and Near-Wall Grid

The grid-dependence in the near-wall region was investigated separately, for the reasons discussed above in section 3.9.4 - since moving the grid points nearest the surface amounts to changing the turbulence model there, such movement may not test merely whether the solution is grid-dependent, but also whether the turbulence model used in the near wall cells (the logarithmic wall function) is consistent with the model used elsewhere (\(k-e\)). Figure 5.20 shows contours of the nondimensional wall coordinate \(n^+\) (see equation 2.177) at the near wall grid point, for the COS3, \(F_h = 0.7\) case. The logarithmic wall function is generally held to be valid for values no smaller than \(n^+ = 20\) or 30. The results indicate that this lower limit is only just reached in some areas of the flow; elsewhere the value is lower, or the flow even completely laminar.

It was felt, however, that increasing the size of the near-wall cells would not be appropriate, since there would then be no hope of resolving the boundary layers, particularly since much of the flow was laminar and thus could not be treated correctly using the wall function regardless of the grid spacing.

Some tests were, however, carried out to investigate the consequences of reducing the size of the near wall cells further. This was done without changing the remainder of the grid in any way; an additional grid point was inserted midway between the previous near-wall point (at which the wall It was found that the size of near wall grid cells could not be reduced substantially, as the wall function was then very clearly applied outside its range of validity. The effect of halving the near wall cell height was to increase the production of turbulence to such an extent that, as a result of the increase in the turbulent kinetic energy, values of the nondimensional height \(n^+\) were not much lower than they had been in the original solution (although still not so high as to indicate satisfaction of the criterion that \(n^+\) be greater than 30). This was taken as an indication that the wall function was being applied in the linear sublayer below the logarithmic region in which it is valid, and that the effects of this included an unrealistic level of turbulence production there. As a result, the simulated flow ceased to exhibit any wave breaking, and did not agree well with the experimentally observed one.

Although this means that it would not be appropriate to attempt to substantially improve the resolution of the near-wall region, it regrettably does not mean that the original resolution was adequate, since no proof of this can be forthcoming.

It was felt to be futile to attempt to improve the modelling of the near-wall region, since turbulence was entirely absent from much of it, and where it was present it was often in combination with recirculation zones or large pressure gradients. All three of these cases are in any case not appropriately dealt with by the logarithmic wall function, regardless of resolution.

Instead, the problem was mitigated as far as possible by using the smallest affordable mesh size for the rest of the boundary layer. This was tested using the grid dependence investigation described
above. It is felt that the results indicate that the remainder of the boundary layer was adequately modelled. However, the modelling of the near wall region should be regarded as less than entirely satisfactory.

Recognizing this, the low-Reynolds number model of Launder and Sharma was implemented in the code, with the intention of using this for future simulations. Owing to the computational expense involved, however, this has not yet been done.
Figure 5.20: COS3 Hill at $F_h = 0.7$, medium grid; Contours of the wall coordinate $n^+$ at the centre of the near-wall cells. Contours range from 0 to 30 in increments of 5. The thick contour is that on which $n^+ = 30$.

Figure 5.21: COS3 Hill at $F_h = 0.7$ medium grid with small near wall cells; Contours of the wall coordinate $n^+$ at the centre of the near-wall cells. Contours range from 0 to 30 in increments of 5. The thick contour is that on which $n^+ = 30$. 
5.3 Influence of Advective Boundary Condition

The choice of the wave speed $c$ used in the advective boundary condition was discussed above in section 3.6.6. The two options investigated were the group and phase velocities of the fastest (columnar) mode, and a value of infinity, reflections arising from slower modes being eliminated by the virtue of the limited time the simulations were integrated to.

Figures 5.22 and 5.23 below show the development of the flow in the COS3, $F_h = 0.7$ case, with $c = \infty$. They may be compared with 4.43 and 4.44 in the results for the COS3 hill above, in section 4.4. Other cases were investigated, with apparently similar results. No difference is apparent until the time reaches about 40. Thereafter, slight differences do appear. The significance of the time is that this is when reflections of the slower $k = 1$ mode would be expected from the downstream boundary, if any were present. The fact that the solutions obtained with two different values of $c$ differ slightly is assumed to imply that such reflections must be present in at least one case, and also that they must be sufficiently weak not to greatly affect the results. The absence of any differences earlier on implies that there is no difference between the performance of the two values of $c$ with regard to the prevention of columnar mode reflection.

It is not really possible, on the basis of these results, to say which choice of $c$ is correct, only that it matters to some (not very great) extent. As previously discussed, the choice of a finite $c$ is preferred, both because it appears to make sense for the columnar modes, and also because it is a better approximation than infinity for other modes. The value of $c$ adopted for the remainder of the simulations reported on here was therefore the velocity (group and phase being the same) of the fastest possible columnar mode.

It should be noted, however, that the behaviour of the solution from $t = 40$ to $t = 50$ appears slightly more steady with $c = \infty$ than with $c = c_g$. The possibility that the former works better in practice cannot be excluded. Results were obtained using $c = \infty$ at Froude numbers of 0.6 and 0.7 on all three hills, and at also at 0.8 on the COS3 hill. All of these results closely resembled those obtained using $c = c_g$ (most more so than the results for COS3, $F_h = 0.7$ shown below). It is clear, therefore, that the influence of the advective boundary condition is weak at the times integrated to.
Figure 5.22: Streamlines, COS3 Hill, $F_a=0.7$, results obtained using zero gradient boundary condition. Plots are of solutions at multiples of 5 time units, starting at $t=5$. 
Figure 5.23: Scalar Contours, COS Hill, \( F_0 = 0.7 \), results obtained using zero gradient boundary condition. Contours are drawn in increments of 5.0 on base of 0. Plotted are solutions separated by 5 nondimensional time units, starting at \( t=20 \).
5.4 Reynolds-Number Sensitivity

The sensitivity of the results to the Reynolds number was investigated by carrying out one run at a Reynolds number of $10^5$ and one at $10^3$, to supplement the standard $10^4$. The COS3 hill at a Froude number of 0.7 was chosen for this purpose. It was hoped that this investigation would clarify to what extent the numerical solution was dependent on this parameter.

To a lesser extent, the results might also aid the interpretation of laboratory experiments, in which the Froude number was varied by varying the towing speed, and thus, simultaneously, the Reynolds number also. Consequently, results at low Froude numbers were obtained at relatively low Reynolds numbers. Independently varying the Reynolds number is possible in numerical simulations; unfortunately, a Reynolds number of $10^8$ already appears to be near the lower limit of the applicability of the wall boundary condition employed by the turbulence model, so that results at lower $Re$ are suspect.

The results obtained from the $Re = 10^5$ simulation (see figure 5.24 below) appear similar to those from $Re = 10^4$ (see figure 4.29 in section 4.4 above). In particular, both solutions clearly display wave breaking. The solution at the higher Reynolds number differs, however, in that the wave breaking zone appears to merge with the low-level recirculating flow, apparently in a similar manner to that observed by Castro and Snyder [17] for the $F_h = 0.6$ case.

The results obtained from the $Re = 10^3$ simulation show no wave breaking (see figure 5.25 below). As previously discussed, this does not necessarily indicate that the laboratory results obtained at a similar Reynolds number may have been affected by the value of this parameter, since the lower boundary condition used in the simulation is of very questionable validity at such a low Reynolds number.

It is concluded that the numerical results are sensitive to downward variations in the Reynolds number, but much less so to upward variations. Hence it is felt to be reasonable to regard the flow at $Re = 10^4$ as Reynolds-number independent.
Figure 5.24: Streamlines. COS3 Hill at $F_h = 0.7$, $Re = 10^3$.

Figure 5.25: Streamlines. COS3 Hill at $F_h = 0.7$, $Re = 10^3$. 
5.5 Sensitivity of Wave-breaking to Wall Function Formulation

All the results discussed above were obtained with the first variant of the wall function described in section 5.5 above. This imposes a turbulent flow wherever it is applied, by setting the near wall coordinate $n^+$ to some specified minimum value (in this case, $20$) if the value calculated would otherwise be lower. This procedure has the effect of ensuring that the use of the wall function is then supposedly valid; however, it does so by overriding the turbulence model, which might indicate laminar flow otherwise.

The cases investigated were half-channel simulations of COS3 at $F_h = 0.7$, $Re = 10^4$ and $10^5$, as well as full channel width simulations of COS1 at $F_h = 0.6$ and COS2 and COS3 at $F_h = 0.7$, all at $Re = 10^4$. Strikingly, none of these showed any evidence of wave breaking; most showed solutions similar to that shown in figure 5.26, with weak lee waves present but no wave breaking. Very thick turbulent boundary layers were observed; this would be expected to suppress wave breaking as forward shear is then imposed at higher altitude.

However, there is also the possibility that the form of the density profile near the ground (which would quickly be eroded by a wall function permitting too much turbulence) might be important. It is tempting to speculate that confusion between katabatic winds, which are driven by density gradients near the ground, and downslope winds such as the Chinook associated with breaking lee waves at high altitude (as has perhaps occurred in the case of the Adriatic Bora - see section 2.2 above), might not be entirely accidental. Indeed, Scorer [100] reports that katabatic flows can trigger lee waves by preventing separation from the lee slope of the hill, thus causing some observers to confuse the effects of the lee waves with the katabatic winds themselves. It also appears that downslope windstorms are of particularly great severity in polar areas, where cooling of the atmosphere by the ground also induces katabatic flow. Although most downslope winds are warm rather than cold, this is due to adiabatic warming of the air as it descends, and not heat transfer from the ground. In fact, since this is most likely colder than the air, the effect is probably also to intensify the strength of stratification near the ground. Such shallow stratified layers are also characteristic of katabatic flows.

It is possible that the reason for the surprisingly large effect of the lower boundary condition might not only be the shear resulting from it (caused by enhanced momentum exchange) but the erosion of the stable stratification of the part of the flow in contact with the ground (caused by enhanced mixing of the fluid). Moreover, the two effects are linked, since stability suppresses turbulence and with it drag and shear, effectively insulating the flow aloft from the surface below (see [34]). Hence it seems possible that the strength of stratification near the surface could perhaps be disproportionately important.

This hypothesis might be tested by investigating nonlinear density profiles that differ in the form of the profile as it approaches the ground.
Figure 5.26: Streamlines. COS3 Hill at $F_h = 0.7$, $Re = 10^5$, wall function imposing minimum $n^+$. 
6. Conclusions

6.1 Findings relating to the Hypotheses made

Conclusions related to the hypotheses tested are, in order:

Hypothesis 1: Applicability of the nonlinear theory of Smith [101].

The hydrostatic nonlinear theory of Smith [101] is found to have some predictive power. However, it agrees less well with experimental as well as numerical results for low Froude number flows, particularly where turbulence caused by wave breaking in the lee of the obstacle extends to the surface. This may be due to poor turbulence modelling for such flows, but also to the incompatibility between the no-slip condition on the surface and the velocity Smith's model predicts on the lower branch of the dividing streamline (equation 2.98).

It was also found to be the case that the simulated flow configuration differs from the postulated one, particularly as the upper branch of the dividing streamline does not remain undisturbed. Disturbances were found near the upper boundary, confirming that wave energy was being transmitted through the mixed region.

Hypothesis 2: Requirement for variable turbulent Schmidt number.

The modelling of turbulent diffusion of the density determining scalar was found to be a critical factor in determining whether a high drag flow configuration was maintained. A parameterization modelling the decrease in turbulent mixing efficiency with stability was tested, but found not to perform well with the \( k-\varepsilon \) model. In particular, too much turbulence was produced, and the breaking wave dissipated.

The reasons that might have given rise to the poor performance of the modified model were investigated; it was concluded that the modification caused the turbulent length and time scales to grow by several orders of magnitude. It was shown that this growth was sufficiently great to cause the turbulence model to interfere with the scales of motion associated with internal waves, which would explain its poor performance.

Hypothesis 3: Appropriate value of constant \( C_{13} \).

The appropriate value of the constant \( C_{13} \) was investigated; a value of zero was found to yield results in good agreement with experiment. A value of unity again led to too much turbulence production, and to the disappearance of wave breaking. It was also shown that this modification
occasioned a moderate growth in the turbulence length and time scales.

Hypothesis 4: Sensitivity of results to small perturbations in stratification.

Piecewise constant stratification is found to approximate linear stratification increasingly well as the layers of constant density are reduced in size. It is concluded that the solutions (with the models used) are reasonably insensitive to small perturbations of the local stratification away from linearity.

Investigations into the effect of weak quadratic departures from linear stratification were also undertaken. It was found that, for a case with marginal wave breaking, stratification increasing with height tended to suppress wave breaking aloft, and result in a flow somewhat resembling a 'hydraulic jump'.

Hypothesis 5: Independence of domain size.

Effects of finite domain extent are found to be important in the transient development of the flow. The steady state apparently converged to (pending the arrival of reflected internal waves) is not greatly affected; the possibility that the final state is not steady cannot, however, be excluded entirely. The fact that domain size is found to have an effect almost immediately suggests that effects other than internal waves may limit the minimum acceptable size of the domain. Pressure reflection from the upstream boundary was suggested as a factor.

The effect of the width and depth of the domain were also investigated. No effect was found at the times integrated to, which were chosen to be low enough to prevent internal waves reflected from the boundaries contaminating the solution.

Hypothesis 6: Consistency of upstream blocking with theory of Smith [101].

The theory of Smith was found to be in very good agreement with simulation results in respect of its predictions of the depth of the blocked fluid upstream of the hill, at least for the two wider hills. Simulations for the axisymmetric hill showed strong three-dimensional effects, however. The upstream-propagating columnar modes were found to be very weak; this may be due to the three-dimensional nature of the obstacles, the poor grid resolution near the inlet boundary, or the turbulence model used, which predicted large values of the turbulent length and time scale windward of the hill.

Hypothesis 7: Consistency of results with wave-breaking regime measured by Castro and Snyder [17].

The critical Froude number at which wave breaking takes place is found to be in broad agreement with experimental results. The aspect ratio versus Froude number diagram above (figure 4.49) shows simulations in which wave breaking was found to take place, and the experimental curve...
of Castro and Snyder. In both cases, the critical Froude number was found to increase with the aspect ratio of the obstacle.

The fact that the wave breaking regimes of the COS2 and COS3 hills are so much more similar than either is to that of COS1 suggests that the effect of hill aspect ratio rapidly diminishes for values much greater than unity. This is corroborated by the observations made with regard to blocking, which also show a similar trend. The theory of Smith suggests that the depth of the blocked layer upstream is strongly correlated with the wave breaking processes in the lee, and these results certainly seem consistent with this.

The turbulence modelling used for the simulations in question was based on the standard \( k-\varepsilon \) model, with a simple modification (equation 2.193) to the eddy viscosity. The standard high-Reynolds number wall function was used; however, this was formulated so as not to impose a minimum value of the nondimensional wall coordinate \( n^+ \) at the centre of the near-wall cell, since this practice was found to lead to results in very poor accord with experiment. In this case, as with the turbulence model modifications described above, it was found that those modifications tested which promoted the growth of turbulence (such as imposing a minimum \( n^+ \)) tended to eliminate the high drag configuration. Substantial forward shear was then observed downstream of the obstacle, and wave breaking was suppressed. This indicates that the lower boundary condition is important in determining whether wave breaking is sustained.

Hypothesis 8: Reproduction of 'merged flow' observed by Castro and Snyder.

This hypothesis appears to be correct inasmuch as the breaking wave is apparently seen to merge with the recirculation zone beneath. Close examination of the results for flow at \( Fr=0.6 \) over the COS3 hill does indicate that the two stagnant regions can still be distinguished to some extent. However, the entire depth of the fluid, from the surface to the upper branch of the dividing streamline, appears very well mixed and turbulent. It does not appear, however, that the simulated flow over the COS2 hill is in similarly good accord with experiment. The details and location of the recirculating flow aloft appear poorly reproduced.

Hypothesis 9: Applicability of symmetry condition on channel centreline.

The applicability of the channel centreline symmetry condition was investigated. It was found that the simulated flow was indeed symmetrical to reasonably good approximation. Some asymmetry was apparent in the lee of the COS3 hill, while flow over the COS1 hill was seen to be perfectly symmetrical. The asymmetry observed for the wider hills arose in regions of low turbulence, and disappeared again in regions of high turbulence. It was reasoned that while this confirms the adequacy of the boundary condition, it is less encouraging with regard to the turbulence model, since it constitutes good evidence that scale separation problems (such as were observed using the variable turbulent Schmidt number) would be expected to arise at lower Froude numbers.

239
6.2 Relevance to Atmospheric Flows

The flows simulated reproduced, as far as possible, the experiments of Castro and Snyder [17], and it appears that the reproduction is somewhat satisfactory. Insofar as the simulation results add to the understanding of the experimental work, their relevance to atmospheric flows is of the same character as that work.

It is believed that the towing tank experiments are meaningful, if idealized, analogues to realistic atmospheric flows. Differences between the two include the Reynolds number (which is several orders of magnitude higher in atmospheric flows), and the Schmidt number. The analogue of the latter in atmospheric flows (the Prandtl number) is near unity, while the Schmidt number for the salinity used to create the experimental stratification is much higher. However, this is largely irrelevant provided molecular diffusion is negligible, which will be the case if the Reynolds number is large enough. It is believed that this is the case (see, for instance, section 5.4 above).

The numerical method presented above has been shown to be capable of reproducing the experimentally observed flows somewhat satisfactorily. It is also able, however, to simulate properties of real atmospheric flows not easily replicated in a laboratory. For instance, it is possible to specify arbitrary inlet velocity and density profiles, simulate arbitrarily large spatial domains, and vary independently parameters that an experimenter might have to vary simultaneously (such as Froude and Reynolds number), in order to assess their effects.

All of these capabilities exist at the time of writing, and should permit numerical simulation of many of the phenomena described in section 2.2 above. There are, however, some potential capabilities that could usefully be implemented in the future, which will be discussed in the following section, as will some potential applications.

6.3 Suggestions for further Work

The following suggestions are made:

1. Further simulations ought to be carried out of the flows considered, using an extended domain as well as a finer grid. This would help eliminate the effects of the impulsive start, as well as allowing the simulations to be continued to for longer without the possibility of interference from reflected waves. This would be necessary to try to determine the nature of any grid dependence found, since the present results indicate that it may be confined to the transition of the flow to turbulence. If, however, it was found to carry over to the (quasi) steady state reached at sufficiently high
most of the work presented above would have to be revisited. Integration to higher $t$ would also help to clarify the nature of the oscillatory drag found in section 4.5 above. In particular, investigations at Froude numbers that do not result in wave-breaking could falsify the hypothesis that this phenomenon is related to the unsteadiness.

2. It has been determined that the $k$-$\varepsilon$ turbulence model is likely to be unsuitable as it stands for the simulation of flows at Froude numbers much lower than those considered above, and also that the inclusion of otherwise apparently physically realistic physical models may cause it to become unsuitable even for these. On the basis of some approximate estimates made in section 4.8.1 above, it is concluded that growth of the turbulent length and time scales is a likely problem. Limitation of the latter did indeed improve the results considerably (see section 4.8.3 above).

Hence, it is proposed that turbulence model modifications involving limitation of the turbulent length and time scales be further investigated for the flows considered above as well as others. At sufficiently low Froude number, turbulence scales larger than those of internal gravity waves are expected; simulations made using models that avoid averaging the latter will therefore essentially be large eddy simulations.

Some features useful for such simulations (e.g. the Smagorinsky subgrid model, and wall functions designed to be used with it) have already been implemented in the method described above. It is suggested, however, that an approach based on a length-scale limited version of the $k$-$\varepsilon$ model (such as that of Apsley and Castro [4]) might also be fruitful, as has already been found to be the case in the simulation described in section 4.8.3 above.

3. The work described above shows the modelling of the lower boundary to be important in the numerical simulations of flows with wave breaking. This suggests, firstly, that better wall models could be investigated, since those currently implemented are not very satisfactory at low Reynolds numbers. An (untested) implementation of the Launder-Sharma low Reynolds number $k$-$\varepsilon$ model is available for this purpose. However, it is expected that use of the ones currently implemented would be much more appropriate for real atmospheric flows, owing to their higher Reynolds numbers.

More interestingly, however, the strong influence of the numerical wall model suggests that the form of the near-wall flow may also be important physically. Most previous studies on similar flows appear to have neglected even the no-slip condition, however. There is therefore probably scope for investigations into the importance of wall roughness, the near-wall density gradient (particularly the effect of heating or cooling the wall), and approach-flow boundary layer thickness.
The effect of a cooled wall might be particularly interesting, since such knowledge might help to clarify the relationship between katabatic flows and downslope windstorms associated with wave breaking aloft.

4. All the work described above was carried out using uniform inlet velocity profiles (with the exception of some simulations aimed at investigating the applicability of the symmetry condition on the centreline), and with mostly linear stratification (some preliminary investigations have been made using nonlinear stratification, but these proved rather inconclusive). Since the method described above never assumes these, it would make sense to apply it to more realistic flows in which they are not present.

5. Further work could be carried out to clarify the role of the large scale vortices observed near the top boundary, which somewhat resemble similar structures remarked upon by Afanasyev and Peltier [1].

6. Realistic modelling of atmospheric flows would require incorporation of Coriolis effects, currently neglected. These would need to be added to the code implementing the method described above.

7. The performance of the code could stand improvement. Initial work has been carried out on the implementation of a multigrid model and parallelization using a 1-D domain decomposition. This should be completed; it is also the case that many possible optimizations (particularly with regard to memory use) have not yet been implemented.
Bibliography


243


249


