ADAPTIVE SUBGRID-SCALE MODELLING AND MULTIPLE MESH SIMULATION OF LOW-REYNOLDS-NUMBER CHANNEL FLOW

A Thesis Presented to The University of Surrey for The Degree of Doctor of Philosophy

by

ZHAO Hong

Department of Mechanical Engineering
University of Surrey
U.K.

October 1994
Acknowledgements

I would like to express my sincere gratitude to Dr. P. R. Voke for his supervision, continuous encouragement and many helpful suggestions during the study. The financial sponsorship provided by SERC is very gratefully acknowledged.

I am also grateful to my colleagues and many staff in the Department for their advice and assistance.
# Contents

1 Introduction .......................................................... 1
   1.1 Turbulence .......................................................... 1
   1.2 Numerical simulation of turbulence ....................... 2
   1.3 The objective of the present study ......................... 8
   1.4 Outline of the thesis .............................................. 9

2 Literature Survey .................................................. 11
   2.1 Large-eddy simulation ........................................... 11
      2.1.1 Introduction .................................................. 11
      2.1.2 Fundamentals ................................................ 12
      2.1.3 Equations of large-scale motions ..................... 13
   2.2 Subgrid-scale modelling ....................................... 15
      2.2.1 Introduction .................................................. 15
      2.2.2 Subgrid-scale models .................................... 16
   2.3 Near-wall model .................................................. 36
   2.4 Subgrid-scale model with $k_e$ not in the $k^{-5/3}$ range .................................................. 39
   2.5 The economy of large-eddy simulations .................. 47
   2.6 Summary ........................................................... 48
5 Simulations With Varying Model Constant  85
5.1 Problem description ................................. 85
5.2 The test runs ........................................... 86
5.3 Results and discussion ............................. 88

6 Simulations With Varying Numerical Resolution  96
6.1 Problem description ................................. 96
6.2 The test runs ........................................... 97
6.3 Results and discussion ............................. 101

7 Simulations With Low-Reynolds-Number Modification  124
7.1 Problem description ................................. 124
7.2 The test runs ........................................... 125
7.3 Results and discussion ............................. 126

8 Simulations Using Dynamic Subgrid-Scale Model  134
8.1 Problem description ................................. 134
8.2 The test runs ........................................... 135
8.3 Results and discussion ............................. 136
9 Simulations Using Multiple Mesh Method

9.1 Problem description ................................................................. 150
9.2 The test runs .................................................................................................. 150
9.2.1 Simulations testing the theoretical foundation of multiple mesh
    method ........................................................................................................ 150
9.2.2 Multiple mesh simulations .............................................................. 153
9.3 Results and discussion ................................................................................ 154

10 Concluding Remarks

10.1 A review of the thesis ................................................................................... 165
    10.1.1 Preliminary study .............................................................................. 165
    10.1.2 Improvement in subgrid-scale modelling ....................................... 167
    10.1.3 Improvement on the economy of the LES method ..................... 168
10.2 Suggestions for future work ....................................................................... 169
    10.2.1 Subgrid-scale modelling ................................................................. 169
    10.2.2 Multiple mesh method ................................................................. 173

References
## List of Tables

3.1 The Pressure ................................................... 62
3.2 The Velocity ..................................................... 62
5.1 The Computational Region and Grid .............................. 87
5.2 Numerical parameters of the simulations with varying model constant 88
6.1 The characteristics of the simulations with varying numerical resolution 100
7.1 The characteristics of the simulations with low Reynolds number modification ..................................................... 126
8.1 The characteristics of the simulations using dynamic SGS model 137
8.2 The dynamically computed model coefficient $C$ ................. 142
9.1 The characteristics of the simulation testing multimesh method 152
9.2 The characteristics of the multiple mesh simulation ............... 155
## List of Figures

3.1 The flow geometry ................................................................. 65

4.1 A coarse mesh computational cell with eight fine mesh subcells ...... 81

4.2 A coarse mesh computational cell with four fine mesh subcells ...... 82

4.3 A coarse mesh computational cells with fine mesh subcells .......... 84

5.1 The history of total volume-averaged $u'^2$ .............................. 90

5.2 The history of total volume-averaged $v'^2$ .............................. 90

5.3 The history of total volume-averaged $w'^2$ .............................. 91

5.4 The history of total volume-averaged kinetic energy $k$ .............. 91

5.5 Streamwise one-dimensional energy spectra at $t = 9\delta/u_r$ .......... 92

5.6 Cross-stream one-dimensional energy spectra at $t = 9\delta/u_r$ .......... 92

5.7 Spanwise one-dimensional energy spectra at $t = 9\delta/u_r$ .......... 93

5.8 LES-10 streamwise one-dimensional energy spectra at $t = 40\delta/u_r$ .......... 94

5.9 LES-10 cross-stream one-dimensional energy spectra at $t = 40\delta/u_r$ .......... 94

5.10 LES-10 spanwise one-dimensional energy spectra at $t = 40\delta/u_r$ .......... 95

6.1 A LESc mesh computational cell with LESm mesh subcells .......... 111

6.2 A LESm mesh computational cell with LESf mesh subcells .......... 111

6.3 The history of the simulation .................................................... 112

6.4 Mean streamwise velocity $u_{mean}$ versus $y^+$ ........................ 113
6.5 Streamwise velocity fluctuation $u_{rms}$ versus $y^+$ ........................................ 114
6.6 Spanwise velocity fluctuation $w_{rms}$ versus $y^+$ ........................................ 115
6.7 Cross-stream velocity fluctuation $v_{rms}$ versus $y^+$ ........................................ 116
6.8 Principal Reynolds stress component $u'v$ ........................................ 117
6.9 Skewness of the streamwise velocity fluctuations ........................................ 118
6.10 Flatness of the streamwise velocity fluctuations ........................................ 119
6.11 Skewness of the cross-stream velocity fluctuations ........................................ 120
6.12 Flatness of the cross-stream velocity fluctuations ........................................ 121
6.13 Skewness of the spanwise velocity fluctuations ........................................ 122
6.14 Flatness of the spanwise velocity fluctuations ........................................ 123
7.1 Mean streamwise velocity $u_{mean}$ versus $y^+$ ........................................ 129
7.2 Streamwise velocity fluctuation $u_{rms}$ versus $y^+$ ........................................ 130
7.3 Spanwise velocity fluctuation $w_{rms}$ versus $y^+$ ........................................ 131
7.4 Cross-stream velocity fluctuation $v_{rms}$ versus $y^+$ ........................................ 132
7.5 Principal Reynolds stress component $u'v$ ........................................ 133
8.1 The subgrid length scale $C$ versus $y^+$ ........................................ 144
8.2 Mean streamwise velocity $u_{mean}$ versus $y^+$ ........................................ 145
8.3 Streamwise velocity fluctuation $u_{rms}$ versus $y^+$ ........................................ 146
8.4 Spanwise velocity fluctuation $w_{rms}$ versus $y^+$ ........................................ 147
8.5 Cross-stream velocity fluctuation $v_{rms}$ versus $y^+$ ........................................ 148
8.6 Principal Reynolds stress component $u'v$ ........................................ 149

9.1 The history of the simulation ......................................................... 158

9.2 The history of mean shear stresses, $uv$, $u'v'$ and $uv + u'v'$ .......... 160

9.3 Mean streamwise velocity $u_{mean}$ versus $y^+$ .......................... 161

9.4 Streamwise velocity fluctuation $u_{rms}$ versus $y^+$ ..................... 162

9.5 Cross-stream velocity fluctuation $v_{rms}$ versus $y^+$ .................... 163

9.6 Spanwise velocity fluctuation $w_{rms}$ versus $y^+$ ....................... 164
Abstract

The present study is aimed at enhancing the effectiveness of the large eddy simulation (LES) approach to the computation of turbulent flows by these two methods: i) developing a superior subgrid scale (SGS) model and ii) improving the economy of LES.

First of all, the various existing SGS models are extensively investigated, and their advantages and disadvantages are addressed to highlight the areas requiring improvements. This study leads to the construction of a modified SGS dynamic model. In addition, a detailed derivation of the second-order velocity structure function SGS model is made, correcting an error found in that model. A new multiple mesh method is also designed to accelerate LES.

After the above theoretical studies, several low-Reynolds-number channel flow simulations have been performed. Firstly, simulations with varying model constants are carried out, and the results agree with those of Deardorff [14], showing that a model constant of about 0.1 is optimum for channel flows.

Secondly, simulations with varying numerical resolutions have been carried out. They reveal that the refinement of the mesh in the direction normal to the wall improves all the turbulence statistics, both higher- and lower-order statistics, over the whole channel, while the refinement of the resolution in the streamwise and spanwise directions improves lower-order statistics over the whole channel, but only improves higher-order turbulence statistics in the central region of the channel.

Thirdly, a dissipation-range SGS model (i.e. the Smagorinsky model with low-Reynolds-number modification [67]) is, for the first time, tested and compared with the standard Smagorinsky model. The results obtained show some promise for automatically adjusting the SGS model with Reynolds number.

Fourthly, the performance of the modified dynamic SGS model is assessed through a comparison of length scales computed respectively by this modified model, the Germano-Lilly dynamic SGS model and two empirical wall damping functions in conjunction with an optimum model coefficient, which have been successfully used in many simulations of channel flows. Two values of the ratio of filter widths are set for each of the dynamic models. The results have confirmed that the modified dynamic SGS model can be successfully extended to simulate low-Reynolds-number channel flows. Of great promise is that the modified SGS dynamic model gives the correct behaviour of the subgrid eddy viscosity in the region of a plane wall to an accuracy that exceeds the best-tuned wall damping function, and almost collapses with the theoretical behaviour of the length scale near the wall without any tuning and adjustment. In addition, the impact of the choice of the ratio of filter widths on the modified dynamic SGS model is much less than that on the Germano-Lilly model.

Finally, simulations using the new and old multiple mesh methods are performed. The instantaneous results just after the interpolation of the coarse mesh velocity field onto the fine mesh show that the fine mesh velocity field created by the new multiple mesh method contains the information of the residual field. In contrast, there is no difference between the fine mesh results obtained by the old method and those from a simulation on the coarse mesh.
1 Introduction

1.1 Turbulence

Turbulence originates in instabilities of laminar flow. It is essentially time dependent and three-dimensional in space. Turbulence is so prevalent in nature and engineering that the practical significance of understanding its fundamental mechanisms and prediction of turbulent flows is evident, and increasing.

A real turbulent motion consists of eddies of various sizes and vorticities. Between them there is a strong interaction due to the nonlinear vortex stretching [26]. Although the range of length scales of all these eddies is significantly large, distinguishable upper and lower limits still exist. The upper size limit is set by the width of the flow, and the lower limit by viscosity effects. In addition, all these various sized eddies have a certain kinetic energy determined by the intensity of the velocity fluctuation of the corresponding spatial frequency. Such a distribution of the energy between the frequencies is called an energy spectrum which represents the characteristic spectral dynamic behaviour of a turbulent flow. In turbulent flows, the largest eddies are produced by imposed flow and maintain themselves by continually absorbing energy from mean flow. Through nonlinear and spatial interactions arising from the nonlinear nature of turbulence, the larger eddies excite smaller eddies and transfer some of their energy to the smaller eddies. This energy cascade process will continue until eventually a scale is reached where nearly all the energy extracted from larger eddies is drained out by viscous dissipation, with none left to pass on down the cascade to smaller scales. This picture of turbulent flows shows that the larger structures are responsible for most of the property transport in turbulent flows, while the smaller structures mainly play a role to dissipate the energy provided by the larger ones [63, 17].
1.2 Numerical simulation of turbulence

With advances in numerical methods and the availability of modern supercomputers, computational fluid dynamics has developed enormously over the last two or three decades. In comparison to experimental investigation, computational predication has the advantages of low cost and greater speed. In addition, a computational prediction has the ability to give detailed information of all the relevant flow variables, such as velocity, pressure, temperature, viscosity, throughout the domain of interest, whereas some primary flow quantities remain inaccessible to direct measurement. Moreover, a computational prediction encounters few limitations on the values of Reynolds number, the size of domain, etc. This advantage allows computational methods to simulate realistic flows. On the other hand, the study of a basic phenomenon, rather than a complex engineering application, may be needed at any time. Such an ideal flow phenomenon of interest can be much more easily and exactly simulated in a computation than in an experiment.

Even though computational fluid dynamics is very promising with all the foregoing advantages, and its potential has been continually enhanced due to the increasing development of computer technology, limitations still remain, and it is not expected that computation will completely replace experimentation in the foreseeable future. One of the most outstanding examples is turbulence, which is one of the remaining unsolved problems in the area of physical science. In spite of the fact that the Navier-Stokes equations can correctly describe the properties of turbulent fluid flows [6], the computation of such an extremely complex and strongly nonlinear flow phenomena in all its time-dependent details (referred to as direct simulation) is still restricted to low Reynolds number. A number of successful simulations of this type have been reported [44, 55, 25, 45]. In a direct simulation, turbulent motion is simulated by numerically integrating the Navier-Stokes equations in three-dimensional
1.2 Numerical simulation of turbulence

space and as a function of time so that all physical scales up to the viscous scale are completely resolved. Therefore the flow fields resulting from a direct simulation are undoubtedly true realizations of turbulent flow fields, and can be analysed to answer questions on the basic behaviour of turbulence. Unfortunately, such a direct simulation of turbulent flows is hopeless in the sense of real engineering application. For a simulation at a Reynolds number $R_e = 10^5$ which is common in most flows of engineering interest, a grid of $10^{11}$ discrete points, which is generally proportional to $R_e^{9/4}$ [28], would be needed in a finite-difference computation to simulate all the turbulent eddies down to and including those with a dissipation length scale, whereas the largest grid that can be handled in present-day computers has $10^7$ mesh points. Moreover, if a decrease in the dissipation scale by one order of magnitude occurs, $10^3$ times more computer storage and $10^4$ times as much computational time would be required due to the four-dimensional nature of the turbulence. On the other hand, the decrease of the dissipation length scale is proportional to $R_e^{-3/4}$, approximately inversely proportional to the increase of $R_e$. Considering this rapid decrease of the dissipation scale with increasing Reynolds number, some authors have therefore speculated that the direct simulation of all scales of high Reynolds number flows is totally impractical because the information carried in such a direct simulation is far more than can be manipulated in a reasonable amount of time by the largest computers currently available [11]. The most ambitious simulation reported to date in terms of total number of mesh points is a study by Spalart of a three-dimensional turbulent boundary layer using a spectral method with up to about $10^7$ grid points [59].

In summary, a direct simulation of turbulent flows of engineering interest requires that viscosity be strong enough to damp out the unresolved scales, or mean velocity be low enough to make the energy cascade process end at resolved scales. It is therefore believed that the computational and memory requirements of a direct
1.2 Numerical simulation of turbulence

simulation of flows of practical importance render such a simulation unfeasible, and that a direct simulation will always be a tool to investigate turbulent motions for a finite band of scales only. Hence, further achievement in turbulent flow simulation is only possible by some techniques incorporating various mathematical models.

Almost all of the approaches to the simulation of turbulent flows are based on some form of averaged Navier-Stokes equations. The averaging method of any type is aimed at reducing the amount of information to be manipulated in a simulation. As a result of this simplification, the Navier-Stokes equations obtained are accordingly no longer closed due to the nonlinear terms in the primitive Navier-Stokes equations. It is therefore concluded that all the turbulence models developed so far are essentially making closure assumptions for the unknowns arising from the averaging process. At present two techniques are widely used. One is based on Reynolds' idea to obtain the time-averaged Navier-Stokes equations for the mean velocity [52]. The other one is to obtain some form of space-averaged Navier-Stokes equations.

Consider first the time-averaged approach, which is also called the Reynolds averaged Navier-Stokes (RANS) method. After the time-averaging technique is applied to the turbulent field, the turbulence variables are smoothly varying because of the removal of high frequency turbulent fluctuations from the primitive equations. It is therefore expected that the solutions become stable and static. This achievement is gained at the price of the introduction of the Reynolds stresses as additional unknowns arising from the averaged nonlinear terms into the averaged equations of motion, resulting in more variables to solve for than there are equations of motion. To deal with the problem we must establish a closure model possessing the following abilities: first of all, the unknowns should be modelled to be able to represent an influence involving the unsteady turbulent motions on the time-averaged equations; secondly, the unknowns should be modeled only by those which are known (or are being computed explicitly). Thus, the solution of the equations is equivalent to the
laminar solution, modified by the Reynolds stresses. Although the technique has been developed extensively, the closure models involved in this technique seem flow dependent, and only successful in the simulations of simple flows, excluding flows involving large mean shear, recirculation, complex geometry of boundaries and intermittency.

It has been speculated that the origin of the severe limitation of the RANS method lay with a major deficiency, namely the necessity of modelling all of the structures of turbulence. Unfortunately, this deficiency is inherent in the time-averaging approach because the average is made over all of the scales of the turbulent motions simultaneously, even the largest ones. It is well known that the large structures are highly inhomogeneous and anisotropic, and accordingly vary greatly from flow to flow. Moreover matters are further complicated in the presence of the foregoing various complex factors, such as large mean shear, recirculation etc. Various attempts have been made to overcome the difficulty either by creating closure models involving higher order correlations modelled in terms of lower order ones, or by adjusting the model to match the experiment for each situation it is applied to. Because higher order statistical quantities adjust themselves more slowly to local conditions than the lower order ones, the first effort to truncate the former is obviously against the aim that the quantities should be modelled in terms of known variables that have a longer response time [53]. In addition, the second effort seems not to hold a better chance than the former to make any achievement because experimentation is often not able to supply sufficient data to adjust the closure models; needless to say, it can be a costly effort. There is therefore little real hope of finding a universal closure model. Obviously, something should be done to the averaging method, which averages away even the largest scales, before the impasse encountered by the RANS method can be circumvented.

The second approach to the computation of the properties of high Reynolds
number turbulent flows employs a different averaging method. In this case, the Navier-Stokes equations are averaged over a small spatial region in space, rather than over all the turbulence scales, to rule out the details of the small scales, and then generate the equations of large scale motions. For obvious reasons it is called large-eddy simulation (LES). Such a truncation at the small scales, which adjust themselves faster to local conditions than the large ones used to model the former (to be discussed later), is preferable to the truncation made in the RANS method of the higher order statistical quantities. The time scales of the modelled terms should be smaller than those of known variables. Unfortunately, the equations obtained for large scale motions are not closed either. The closure models involved in this technique are called subgrid-scale (SGS) models. In a large-eddy simulation, the detailed time-dependent motions of large eddies are computed explicitly. In contrast, the small eddies have to be modelled by a SGS model which can be viewed as an effect of the truncated small scales on the large ones. This idea, from which LES was developed, is based on two experimental observations: first, the large-scale turbulent structures are quite variable from flow to flow, therefore it is impossible to model in a general way; second, the small-scale turbulent structures are more universal due to the effect of the cascade, and hence much more amenable to general modelling. Such a closure model should act as a communicative channel between the explicit and the implicit dynamics of turbulence. Generally, unless the statistical dynamics of the turbulence are understood well enough, the communication cannot be described correctly. Fortunately we are quite knowledgeable about the roles played by large- and small-scale eddies respectively [26, 36]. The large-scale eddies dominate the turbulent flow properties, produce turbulent transport which depends on the gross character of the flow, and have a very direct relationship to the local mean-flow structure. They are the energy-containing eddies and involved in turbulence energy production. By contrast, the small- scale eddies mainly play a more general role of the acceptance and dissipation of the energy extracted from those resolved scales
1.2 Numerical simulation of turbulence

which lie at the border of resolution to avoid "damming up" of the turbulence energy in the large scales. They are the energy-cascade and energy-dissipation eddies. Therefore, provided that the large scales of motion are resolved directly, the LES technique should be able to describe turbulent motions more precisely. There are two reasons contributing to the optimism about such an approach. First of all, as most of the energy-containing eddies are resolved, the results might be anticipated to be insensitive to the details of how the small-scale motions are dealt with. Secondly, it is observed that, while large-scale eddies differ considerably between flows, the small-scale motions hardly change in character, and may thus be parameterized more rationally.

In brief, the SGS-motion carries only a fraction of the total turbulent kinetic energy, and it plays a more passive role in various turbulent flows of removing kinetic energy from the large scales and dissipating it. Thus the approximation errors in SGS models are expected to be of less importance than those in a time-averaging method, though the space-averaged turbulent flow field, like time- or ensemble-averaged ones, ignores some information of turbulent flows, and SGS models arising from the process of the space-averaging are formally very similar to those commonly used to close the time- or ensemble-averaged equations of turbulent motions. Furthermore it can be concluded (optimistically) that LES involving SGS models can be applied to computing a wide range of turbulent flows as the best substitute so far for direct simulation, but at much lower cost of computer time if the grid is fine enough to make the fraction of energy resident in the small eddies small and therefore make LES relatively insensitive to the quality of the subgrid-scale model. From this point of view, large eddy simulation is more promising and has a greater potential than time-averaging closure method in engineering applications. It is, however, necessary to be aware that there are two areas requiring improvements before large-eddy simulation can be used as an engineering tool. First, there is a need to create a superior
1.3 The objective of the present study

The objectives of the present work are as follows:

1) Although most recent results obtained by the workers in the field of large-eddy simulation have proved that LES is indeed a valuable research tool, the further development of subgrid-scale modelling to make LES applicable to practical engineering is still central to many studies on large-eddy simulation. One facet of the present study was to construct a better SGS model to enhance the effectiveness of LES. There are several factors that can improve SGS modelling of LES, such as the behaviour in the near-wall region, the dependence on the flows, and the ability to represent the Reynolds stresses on a local basis besides the correct mean energy balance of large-scale flow field and to adjust itself automatically with Reynolds number, etc. An extensive investigation of the existing SGS models was therefore made, and the advantages and disadvantages were addressed to highlight the areas requiring improvements. Also a nearly full simulation of the channel turbulent flow was performed at low Reynolds number to obtain further insight and understanding into the mechanism of turbulent flow, which is essential for a marked improvement in the efficiency of LES. On the basis of those investigations, a modified dynamic SGS model was proposed.
2) One of the main disadvantages of LES is that it is significantly expensive in terms of computing time, which greatly defers the application of LES in practical engineering. Even with the expected advances in computers coming from the introduction of massively parallel machines and larger memory chips, the cost of such simulations will remain high. On the other hand, an improvement in SGS modelling may come at increased cost. It is, therefore, speculated that the technique will have an excellent chance of becoming a computational tool for the solutions of the problems of engineering interest rather than only for those of a limited range of well-chosen flows as long as the cost of computation decreases. The other main aim of the present study is to develop a multiple mesh method to improve the economy of LES, to attempt to fulfil its promise as an instrument for engineering design and analysis.

1.4 Outline of the thesis

Chapter 2 serves as a literature survey, in which the concept of LES and its valid scope are stated; in the second place, the various subgrid-scale models are reviewed in detail. In addition, a comparative study of all the existing SGS models is made to expound their advantages and disadvantages.

In chapter 3, the governing equations which describe turbulent flows are presented. Then the equations of large-scale motions are introduced, meanwhile, the involved space-averaging method which introduces more unknowns is discussed, followed by a detailed description of the closure assumptions actually used in the present study for these unknowns (known as SGS models), with an great emphasis placed on a new dynamic SGS model. Finally, details of the numerical methods are given. The coupling between the continuity and the momentum equations together with a discussion on the finite-difference scheme for the pressure gradient is
1.4 Outline of the thesis

described.

Chapter 4 is devoted to improving the economy of the LES method. Firstly the concept of the multiple mesh method is introduced, followed by a detailed derivation of the formulation and a description of the implementation of this method.

The following five chapters, 5, 6, 7, 8 and 9, concentrate on testing the LES technique. Here the turbulent channel flow is selected so that the study can focus on fundamental issues of LES and SGS modelling. From the results obtained, the effects of the values of the model constant, grid refinement, low-Reynolds-number modification and a modified SGS dynamic model on the results are investigated. In addition, the results of the present large eddy simulations are compared with experimental results and, when available, with the results of direct simulations of NASA. Finally, the theoretical foundation of the multiple mesh method is tested, and real multiple mesh simulations are carried out to show their economic advantage.

In chapter 10, a brief review of the present study, concluding remarks on the results and suggestions for future work are given.
2 Literature Survey

2.1 Large-eddy simulation

2.1.1 Introduction

As described in the previous chapter, turbulent flows contain eddies with a wide range of scales. But such a vast number of eddies can be simply divided into two characteristically different groups according to their behaviour: i.e. large eddies, which absorb energy from the mean flow and perform most of the turbulent transport, and small eddies, which accept the energy transferred from the large ones and dissipate it. Mathematically, there should exist two length scales to represent them. It is very straightforward to set the length scale of the largest eddies by the integral length scale of the velocity correlations because it is determined mainly by the geometry that encloses the flow or produces the turbulence, and limits the size of possible eddies. The length scale of the smallest is regarded as the Kolmogorov length \( \eta \) which is related to the kinematic viscosity \( \nu \) and the rate of dissipation of energy \( \epsilon \) (per unit mass) in the following form [62]:

\[
\eta^4 = \nu^3 / \epsilon .
\]  

(2.1)

Equation (2.1) shows that the smallest eddies adjust themselves in each flow so that the amount of energy passing down the cascade may be dissipated, i.e. the more energy to be dissipated or the smaller viscosity, the smaller \( \eta \); the less energy to be dissipated or the larger viscosity, the larger \( \eta \). Accordingly, in a LES the large and small scales are treated mathematically in two different ways: the large-scale motions are computed explicitly in detail, while the small ones are modelled by some form of SGS model. This approach to the computation of turbulent flows is generally defined as LES [17].
2.1 Large-eddy simulation

2.1.2 Fundamentals

Some workers have claimed that the realization of LES in a practical implementation requires the existence of an intermediate zone, called an inertial subrange, to separate large scales and small scales, corresponding to the low wavenumber and high wavenumber ranges respectively in the wavenumber space. An ideal inertial subrange is a regime which should only extract energy from the larger eddies and lose it to the smaller ones and be free from any sources and sinks of kinetic energy, i.e. the kinetic energy migrates from low to higher wavenumbers purely by the nonlinear interactions of the inertial forces, and the total kinetic energy of this subrange is conserved. In such an inertial subrange, the functional form of the energy spectrum can be determined uniquely by the wavenumber $k$ and the amount of energy passing through it. Dimensional analysis gives [57]

$$E(k, t) = C_k e^{2/3} k^{-5/3}, \quad C_k \approx 1.4.$$  \hspace{1cm} (2.2)

For real turbulent flows, a subrange in which negligible dissipation occurs and the transfer of energy by inertia forces is the dominant process may be considered as an inertial subrange. Obviously, with continuing decreases in the value of Reynolds number this range will shrink, vanish, and eventually an overlap could occur between the energy production range and the dissipative range. A distinct inertial subrange therefore only exists if the mean flow Reynolds number is sufficiently large. An inertial subrange is very desirable for LES, but fortunately some authors [18] have stated that the existence of an inertial subrange is an unnecessarily strict condition. However, a less strict requirement that the subgrid scales be entirely in the equilibrium range is required for LES to work. The notion of an equilibrium range implies a zone in which the amount of energy transferred through the eddies is large compared with the rate of change of their energy; in such an equilibrium range, the structure of the small scale turbulence depends only on the energy flux through this
range and the rate of dissipation. Generally, the grid-scale cutoff should lie in this subrange to separate the resolved and subgrid scales. This separation of large and small scales is the first concern in a LES.

### 2.1.3 Equations of large-scale motions

Owing to the widely accepted definition that a large-eddy simulation is any simulation of a turbulent flow in which the large-scale motions are explicitly resolved while the small-scale motions are represented approximately by SGS models, the first step in applying the LES technique to any turbulent flow is to separate what is to be computed (large-scale motions) and what must be modelled (subgrid-scale motions). The most straightforward means of doing so is to decompose the velocity into the grid scales which are sufficiently large to be represented in the calculation, and the subgrid scales whose information is unknown. As has already been presented in the previous chapter, the decomposition can be done by some form of space-averaging method. Mathematically the decomposition is via filtering. A general filtering operation has been proposed by Leonard [31]:

\[
\bar{f}(x_i) = \int G(x_i - x'_i) f(x'_i) dx'_i
\]

in which \(f(x_i)\) is a function containing all the scales, \(\bar{f}(x_i)\) the large-scale or resolvable-scale component of \(f\) and \(G\) the filter function. The integral in the equation (2.3) is extended over the entire domain.

Filter functions commonly used include the Gaussian and the sharp Fourier cutoff. They are defined respectively in physical space as [31]

\[
G_i(x_i - x'_i) = (6/\pi)^{1/2}/\Delta_i \exp[-6(x_i - x'_i)^2/\Delta_i^2], \quad (i = 1, 3)
\]

\[
G_i(x_i - x'_i) = 2 \sin[\pi(x_i - x'_i)/\Delta_i]/[\pi(x_i - x'_i)], \quad (i = 1, 3)
\]
2.1 Large-eddy simulation

Some authors apply a sectionally continuous top hat filter with variable width, to the
direction normal to the walls to account for variation of turbulence length scale in
that direction [41]. Also the discretization of the differential equations is considered
as a kind of filter.

When the filter (2.3) is applied to the Navier-Stokes and continuity equations,
the filtered equations are given, in dimensionless form, by

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0,$$

(2.6)

$$\frac{\partial \overline{u}_i}{\partial t} + \frac{\partial}{\partial x_j}(\overline{u}_i \overline{u}_j) = -\frac{\partial \overline{p}}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} + \frac{1}{Re} \frac{\partial^2 \overline{u}_i}{\partial x_i \partial x_j}. $$

(2.7)

Equations (2.6) and (2.7) govern the evolution of the large scales, but with the
following additional term arising from the filtering process, the operation of the
artificial truncation at small scales,

$$\tau_{ij} = \overline{u}_i \overline{u}_j - \overline{u}_i \overline{u}_j$$

(2.8)

where $\tau_{ij}$ are the SGS stresses which must be modelled in a LES to represent the ef­
fect of the small scales on the large ones. If we decompose $u_i$, the velocity containing
all scales, into its resolved-scale and subgrid-scale components:

$$u_i = \overline{u}_i + u_i' ,$$

(2.9)

then the SGS stresses are therefore decomposed into three parts: the resolvable part,
$L_{ij} = \overline{u}_i \overline{u}_j - \overline{u}_i \overline{u}_j$ (also called Leonard term), the cross term, $C_{ij} = \overline{u}_i u_j' + \overline{u}_j u_i'$, and
the SGS Reynolds stresses, $R_{ij} = u_i' u_j'$. The various SGS models arise from the
different approximations of these three terms which are either treated together or
separately.
2.2 Subgrid-scale modelling

Since the first true LES involving a SGS model was performed by Deardorff [12], construction of better SGS models has always been the essential for improving the efficiency of LES. New models have been continuously developed and tested. This section will give a brief introduction to SGS modelling, followed by an exhaustive review of the current SGS models.

2.2.1 Introduction

Most of the existing SGS models use an eddy viscosity to account for the influence of the subgrid-scale motions on the large-scale motions. That means that the energy cascade is viewed solely as an energy loss of the large-scale motions due to an artificial viscosity arising from subgrid-scale motions. There are two reasons for doing so. First of all, the predominant role of the small eddies is to act as acceptors of energy from the large eddies and to dissipate it. Secondly, the transfer of energy from the large to small eddies is largely a one-way process and appears to the large eddies as a dissipative effect. Thus, the basis of SGS models of the eddy viscosity type is to produce an eddy viscosity in partial analogy to the molecular case. With the eddy viscosity $\nu$, as the proportional factor, the anisotropic part of the SGS stresses of this type are assumed proportional to the large-scale strain rate tensor $\overline{S}_{ij}$, i.e.,

$$\tau_{ij} = -2\nu \overline{S}_{ij}$$

(2.10)

where $\overline{S}_{ij} = \frac{1}{2}(\partial u_i/\partial x_j + \partial u_j/\partial x_i)$ is the large-scale strain rate tensor or the large-scale deformation tensor. The eddy viscosity hypothesis is typically valid for most free flows, in particular at high Reynolds numbers. However, the opposite, energy transport from small scales to large scales (backscatter), can occur intermittently. In calculations made by Piomelli [46], it was found that half the grid points in
direct numerical simulations of transitional and fully turbulent channel flows were experiencing backscatter, and that the backscatter was even more important in the buffer layer or in the transitional stage. Concerning backscatter, Germano et al. [20] proposed a dynamic SGS model which allows for temporal and spatial variations of the model parameter. A negative model parameter corresponds to backscatter. There is also another type of SGS model called the scale similarity model [2, 3], which is aimed at limiting the correlation between $\tau_{ij}$ and $S_{ij}$ imposed by the SGS models of the eddy viscosity type because it is speculated that the imposed high correlation is responsible for the underestimated $\tau_{ij}$ variance. An exhaustive review of the various existing SGS models will be given in the following sections.

2.2.2 Subgrid-scale models

Smagorinsky model

The Smagorinsky model, first introduced by Smagorinsky [56] and further developed by Lilly [33], is one of the first SGS models, and is still widely used. This model is absolutely dissipative in nature, which will be seen shortly. In the case of the Smagorinsky model, the eddy viscosity involved in the equation (2.10) is defined by

$$\nu_t = (C_s \Delta)^2 \overline{S}$$

with

$$\overline{S}^2 = 2\overline{S_{ij}S_{ij}}$$

where $C_s$ is a constant, $\overline{S}$ the amplitude of the large-scale strain rate tensor (or velocity deformation), and $\Delta$ the mesh separation distance.

There has been an argument about the value of $C_s$, since the constant was first given a value of about 0.23 on the basis of the studies of the decay of homogeneous turbulence. Other researchers also deduced the value of the constant for
2.2 Subgrid-scale modelling

homogeneous isotropic turbulence in different way. For example, Lilly [34] obtained a value for \( C_s \) of about 0.17. Recently a value of 0.2 was determined by Métais and Lesieur [40] by assuming that the strain rate ratio \( \langle S^3 \rangle / \langle S^3 \rangle \) is unity. In the following, a value for \( C_s \) of about 0.17 is derived from a Kolmogorov spectrum [4] of homogenous turbulence on the basis of the usual local-equilibrium-turbulence arguments.

At high Reynolds number, the rate of resolved turbulent kinetic energy dissipation (which is assumed equal to the rate of the energy transfer from the resolved-scales to the subgrid-scales) is well approximated by [34]

\[
\epsilon \approx -\tau_{ij} \tilde{S}_{ij} .
\]  

(2.13)

Upon substitution of (2.10) and (2.11) into (2.13), the energy dissipation can be written as

\[
\epsilon = (C_s \Delta)^2 \overline{S}^3 .
\]  

(2.14)

On the other hand, for local-equilibrium turbulence at a high Reynolds number, the energy balance for the resolved-scales reads [4]

\[
\epsilon = \int_0^{k_c} 2\nu k^4 E(k, t) dk
\]  

(2.15)

where \( k_c \) is the largest wave number unambiguously representable on a finite difference mesh, whose relation with the mesh spacing \( \Delta \) is \( k_c = \pi / \Delta \). For a Kolmogorov spectrum

\[
E(k, t) = C_k \epsilon^{5/3} k^{-5/3}, \ C_k \approx 1.4 ,
\]  

(2.16)

with \( k_c \) falling in the inertial subrange, the integration of equation (2.15) gives \( \epsilon \) the form

\[
\epsilon = \frac{3}{2} \nu_4 C_k^3 (\pi / \Delta)^4 .
\]  

(2.17)

Finally, substitution of (2.11) and (2.14) into (2.17) yields the result

\[
C_s = 0.23 C_k^{-3/4} \approx 0.17 .
\]  

(2.18)
2.2 Subgrid-scale modelling

The Smagorinsky model with a value for $C_s$ of about 0.2 was successfully applied in the large eddy simulations of homogeneous turbulence [38] and buoyant convection [13]. In contrast to the simulations of homogeneous turbulence and buoyant convection, large eddy simulations of other flow configurations have proved more difficult to perform successfully. For example, in the presence of mean shear, Deardorff found the value of $C_s$ of about 0.2, appropriate to homogeneous turbulence, damps resolved motions. On the other hand, smaller values of $C_s$ gave excessive turbulence energy [14]. Also, McMillan, Ferziger and Roagillo [39] found that the value of $C_s$ must be lowered in the presence of strain or shear.

Besides the sensitivity of various turbulent flows to $C_s$, Clark et al. [9] found that eddy viscosity models of the Smagorinsky type predict the global energy transfer from large to small scales with acceptable accuracy, but fail to predict the local stresses.

There are also two other obvious deficiencies related to the Smagorinsky model. First, the absolutely positive value of the Smagorinsky eddy viscosity indicates that the model accounts for only transport of energy from large scales to the small ones. Second, the Smagorinsky eddy viscosity fails to vanish at the walls.

Despite all the shortcomings of the Smagorinsky model, it is still one of the most popular SGS models owing to its ability to predict the global energy transfer adequately. It therefore is either used alone in some simple flows or used as a base model in some form of combined models to simulate more complex flows. These will be presented later in this chapter.
2.2 Subgrid-scale modelling

Second-order velocity structure function model

The second-order velocity structure function model is also in the form of an eddy viscosity, i.e.

\[ \tau_{ij} = 2\nu_s \overline{S}_{ij} - \frac{2}{3} K_s \delta_{ij} \]  \hspace{1cm} (2.19)

where \( K_s \) is the subgrid kinetic energy which is defined as

\[ K_s = \int_{k_c=\pi/\Delta}^{\infty} E(k, t) dk . \]  \hspace{1cm} (2.20)

However, in contrast to the constant eddy viscosity coefficient used by Smagorinsky, the eddy viscosity coefficient in this model is computed in physical space at each point based upon a kinetic energy spectrum local in space, which is determined with the aid of a local second-order velocity structure function

\[ F_2(x, r, t) = < |u(x + r, t) - u(x, t)|^2 > |r|_r \]  \hspace{1cm} (2.21)

where the operator "< . >" stands for the spatial average over a sphere of radius \(|r| = r\) surrounding \(x\).

The eddy viscosity based upon the second-order velocity structure function [40] was developed from the spectral eddy viscosity [8], which depends only on time and is in the following form

\[ \nu_t(k/k_c, t) = \nu_t^+(k/k_c) \sqrt{E(k_c, t) / k_c} . \]  \hspace{1cm} (2.22)

Basically, (2.22) is the modification with respect to Kraichnan's eddy viscosity [30] through normalization with the aid of the kinetic energy spectrum at the cutoff wavenumber \(k_c\).

For a Kolmogorov spectrum, assuming \(\nu_t\) dependent only on time, the integration of (2.15) yields

\[ \epsilon = 2\nu_t C_k \epsilon^{3/2} \left( \frac{3}{4} \right)^{1/3} k_c^{4/3} . \]  \hspace{1cm} (2.23)
Rewriting (2.23), we can obtain
\[ \frac{3}{2} \nu_t C_k^{3/2} (C_k e^{2/3} k_c^{-5/3} / k_c)^{-1/2} = 1. \] (2.24)

The term in the brackets on the left side of (2.24) is equal to \( E(k_c, t) / k_c \), so the substitution of (2.22) into (2.24) yields
\[ \nu_t^+ = \frac{2}{3} C_k^{-3/2}. \] (2.25)

The advantage of (2.22) is that no energy is extracted from the system in the early stage of the energy cascade, when no energy has reached the cutoff yet. This behaviour is desirable for the linear and early nonlinear stages of transition where the Smagorinsky eddy viscosity fails. However, the eddy viscosity defined by (2.22) with \( \nu_t^+ = \frac{2}{3} C_k^{-3/2} \) is uniform in physical space, which is not reasonable because turbulence is highly intermittent and \( \nu_t \) is therefore expected to be different at each spatial point. In order to account for the intermittency, an eddy viscosity varying in space was introduced by Métails and Lesieur [40]
\[ \nu_t(x, \Delta, t) = \frac{2}{3} C_k^{-3/2} \sqrt{\frac{E(x, k_c, t)}{k_c}} \] (2.26)

where \( \Delta \) is the computational grid mesh in the physical space and \( E(x, k_c, t) \) is a local kinetic energy spectrum at \( x \). For isotropic turbulence, the local kinetic energy spectrum can be calculated by using the second-order velocity structure function in physical space through the following relation [4],
\[ F_2(r, t) = 4 \int_0^\infty E(k, t) \left[ 1 - \frac{\sin(kr)}{kr} \right] dk. \] (2.27)

In both the studies of Métails and Lesieur [40], and Comte et al. [10], the formulation of the second-order velocity structure function has been derived. However, some error was found in both their derivations. In the following, a correct formulation is deduced.
2.2 Subgrid-scale modelling

In order to integrate (2.27), we assume that $E(k, t)$ corresponds to an inertial-range Kolmogorov spectrum extending from $k = 0$ to infinity. The integration of (2.27) is therefore

$$F_2(r, t) = 4 \int_0^\infty C_k \epsilon^{2/3} k^{-5/3} \left[ 1 - \frac{\sin(kr)}{kr} \right] dk$$

$$= 4C_k(\epsilon r)^{2/3} \int_0^\infty (kr)^{-8/3} [kr - \sin(kr)] d(kr)$$

$$= 4C_k(\epsilon r)^{2/3} \left\{ -\frac{3}{5} \left[ \frac{kr - \sin(kr)}{(kr)^{5/3}} \right]_0^\infty + \frac{3}{5} \int_0^\infty (kr)^{-5/3} [1 - \cos(kr)] d(kr) \right\}$$

$$= 4C_k(\epsilon r)^{2/3} \left\{ -\frac{9}{10} \left[ \frac{1 - \cos(kr)}{(kr)^{2/3}} \right]_0^\infty + \frac{9}{10} \int_0^\infty (kr)^{-2/3} \sin(kr) d(kr) \right\}$$

$$= 4C_k(\epsilon r)^{2/3} \frac{9}{10} \int_0^\infty (kr)^{1/3 - 1} \sin(kr) d(kr)$$

$$= \frac{9}{5} \Gamma(\frac{1}{3}) C_k(\epsilon r)^{2/3}$$

$$\approx 4.82 C_k(\epsilon r)^{2/3} \quad (2.28)$$

Assuming that $k_c$ lies in the inertial subrange, substituting (2.28) into (2.26) wherever the right hand of (2.28) appears, and putting $k_c = \frac{r}{\Delta}$, then we can derive the eddy viscosity based upon the second-order velocity structure function

$$\nu_t = \frac{2}{3} C_k^{-3/2} \sqrt{\frac{C_k^{2/3} k_c^{-5/3}}{k_c}}$$

$$= \frac{2}{3} C_k^{-3/2} \sqrt{\frac{\frac{9}{5} \Gamma(\frac{1}{3}) C_k(\epsilon r)^{2/3} k_c^{-8/3}}{\frac{9}{5} \Gamma(\frac{1}{3}) r^{2/3}}}$$

$$= \frac{2}{3} C_k^{-3/2} \sqrt{\frac{F_2(r, t)}{\frac{9}{5} \Gamma(\frac{1}{3}) k_c^{3/3} r^{3/3}}}$$

$$= \frac{2}{3} C_k^{-3/2} k_c^{-4/3} \frac{\frac{9}{5} \Gamma(\frac{1}{3})^{-1/2}}{r^{-1/3} \sqrt{F_2(r, t)}}$$

$$= \frac{2}{3} C_k^{-3/2} \frac{9}{5} \Gamma(\frac{1}{3})^{-1/2} \left( \frac{r}{\Delta} \right)^{-4/3} \sqrt{F_2(r, t)}$$

$$= B \left( \frac{r}{\Delta} \right) r \sqrt{F_2(r, t)} \quad (2.29)$$
where \( B(\frac{r}{\Delta}) = \frac{2}{3} C_k^{-3/2}(\frac{r}{\Delta})^{-4/3} \pi^{-4/3} \left[ \frac{\pi}{6} \Gamma(\frac{1}{3}) \right]^{-1/2} \). If setting \( r = \Delta \), (2.29) is therefore

\[
u_i = 0.067 C_k^{-3/2} \Delta \sqrt{F_2(r, t)}.
\] (2.30)

Note that in LES, only resolved velocity can be calculated, so \( F_2(r, t) \) has to be determined by the following formula

\[
F_2(r, t) = F_2(r, t) + C_0
\] (2.31)

where \( F_2(r, t) \) is the second-order velocity structure function contributed by the resolved velocity (called the filtered second-order velocity structure function), and \( C_0 \) is a correction for the subgrid contribution, i.e. the contribution from the eddies with wavenumber higher than \( k_c \). The expression for \( C_0 \) can be given by analogy to (2.27)

\[
C_0(r, t) = 4 \int_{k_c}^{\infty} E(k, t) \left[ 1 - \frac{\sin(kr)}{kr} \right] dk.
\] (2.32)

Assuming again an inertial-range extending to infinity, we integrate (2.32) with \( \zeta = r/\Delta \) and \( k_c = \pi/\Delta \)

\[
C_0(r, t) = 4 \int_{k_c}^{\infty} C_k \varepsilon^{2/3} k^{-8/3} \left[ 1 - \frac{\sin(kr)}{kr} \right] dk
\]

\[
= 4 C_k (er)^{2/3} \int_{kr}^{\infty} (kr)^{-8/3} [kr - \sin(kr)] d(kr)
\]

\[
= 4 C_k (er)^{2/3} \left[ \int_{kr}^{\infty} (kr)^{-8/3} d(kr) - \int_{\pi}^{\infty} (kr)^{-8/3} \sin(kr) d(kr) \right]
\]

\[
= 4 C_k (er)^{2/3} \left\{ \left( -\frac{3}{2} \right) [(kr)^{-2/3}]_{\pi}^{\infty} + \frac{3}{5} [(kr)^{-8/3} \sin(kr)]_{\pi}^{\infty} \right\}
\]

\[
= 4 \left( \frac{3}{2} \pi \zeta \right)^{-2/3} - \frac{3}{5} (\pi \zeta)^{-8/3} \sin(\pi \zeta)
\]

\[
+ \frac{9}{10} [(kr)^{-2/3} \cos(kr)]_{\pi \zeta}^{\infty} + \frac{9}{10} \int_{\pi \zeta}^{\infty} (kr)^{-2/3} \sin(kr) d(kr) \right\}. \quad (2.33)
\]

From Gradshteyn and Ryzhik [24], formula 3.761.2:

\[
\int_{-\infty}^{\infty} x^{n-1} \sin x \, dx = \frac{i}{2} \left[ e^{-\pi i \mu} \Gamma(\mu, iu) - e^{\pi i \mu} \Gamma(\mu, -iu) \right]
\] (2.34)
for the real part of $\mu \mathcal{R}_e(\mu) > -1$, the above equation (2.33) yields

$$C_0(r,t) = 4C_e(cr)^{2/3} \left\{ \frac{3}{2}(\pi \zeta)^{-2/3} - \frac{3}{5}(\pi \zeta)^{-5/3} \sin(\pi \zeta) - \frac{9}{10}(\pi \zeta)^{-2/3} \cos(\pi \zeta) \right\}$$

$$+ \frac{9i}{20} \left\{ e^{-i\pi/6} \Gamma\left(\frac{1}{3}, i\pi \zeta\right) - e^{i\pi/6} \Gamma\left(\frac{1}{3}, -i\pi \zeta\right) \right\}$$

$$= \frac{9}{5} \Gamma\left(\frac{1}{3}\right) C_e(cr)^{2/3} H\left(\frac{r}{\Delta}\right)$$

(2.35)

where

$$H\left(\frac{r}{\Delta}\right) = \frac{4}{5} \left\{ \left\{ \frac{3}{2} - \frac{9}{10} \cos(\pi \zeta) \right\} \pi^{-2/3} \left(\frac{r}{\Delta}\right)^{-2/3} - \frac{3}{5} \pi^{-5/3} \left(\frac{r}{\Delta}\right)^{-5/3} \sin(\pi \zeta) \right\}$$

$$+ \frac{9i}{20} \left\{ e^{-i\pi/6} \Gamma\left(\frac{1}{3}, i\pi \zeta\right) - e^{i\pi/6} \Gamma\left(\frac{1}{3}, -i\pi \zeta\right) \right\} .$$

(2.36)

Note that this solution of $C_0(r,t)$ is a correction to the formulations derived by Métais et al. [40] and Comte et al. [10]. They applied the formula (2.34), for $\mathcal{R}_e(\mu) > -1$, with $\mathcal{R}_e(\mu) = -\frac{5}{3}$ which is obviously incompatible with the condition $\mathcal{R}_e(\mu) > -1$. The relation between $C_0(r,t)$ and $F_2(r,t)$ is easily found by substituting (2.28) into (2.35)

$$C_0(r,t) = F_2(r,t) H\left(\frac{r}{\Delta}\right) .$$

(2.37)

Now we can obtain $F_2(r,t)$ expressed as a function of $\bar{F}_2(r,t)$ by substituting (2.37) into (2.31):

$$F_2(r,t) = \frac{1}{1 - H\left(\frac{r}{\Delta}\right)} \bar{F}_2(r,t) .$$

(2.38)

Finally the expression for the eddy viscosity based on the second-order structure function becomes

$$\nu_t(x,\Delta,t) = B\left(\frac{r}{\Delta}\right) r \sqrt{\frac{1}{1 - H\left(\frac{r}{\Delta}\right)}} \sqrt{\bar{F}_2(r,t)} .$$

(2.39)

Now consider the subgrid kinetic energy $K_s$ which is defined as

$$K_s = \int_{k_s}^{\infty} E(k,t) \, dk .$$

(2.40)
2.2 Subgrid-scale modelling

For a Kolmogorov spectrum, the subgrid kinetic energy is found from (2.40) to be

\[ K_s = \frac{3}{2} C_k (cr)^{2/3} \pi^{-2/3} \left( \frac{r}{\Delta} \right)^{-2/3} \]

\[ = \frac{9}{5} \Gamma(\frac{1}{3}) C_k (cr)^{2/3} \frac{3}{2} \pi^{-2/3} \left( \frac{r}{\Delta} \right)^{-2/3}. \]  

(2.41)

From (2.28), (2.38) and (2.41), \( K_s \) is expressed as

\[ K_s = \frac{\pi^{-2/3}}{\frac{9}{5} \Gamma(\frac{1}{3})} \frac{3}{2} \left( \frac{r}{\Delta} \right)^{-2/3} \frac{1}{1 - H \left( \frac{r}{\Delta} \right)} F_2(r, t). \]  

(2.42)

Substituting (2.39) and (2.42) into (2.19), we can obtain the subgrid Reynolds stresses

\[ \tau_{ij} = 2B \left( \frac{r}{\Delta} \right) r \left[ 1 - H \left( \frac{r}{\Delta} \right) \right]^{-1/2} \sqrt{F_2(r, t)} \delta_{ij} \]

\[ - \frac{\pi^{-2/3}}{\frac{9}{5} \Gamma(\frac{1}{3})} \left[ 1 - H \left( \frac{r}{\Delta} \right) \right]^{-1} F_2(r, t) \delta_{ij}. \]  

(2.43)

When the SGS model based on second-order velocity structure function is actually to be used in LES, clearly the definition of the structure function \( F_2 \) should be made first. In addition, the form of \( F_2 \) will affect near wall behaviour of the model. Comte et al. [10] have given three options for calculating the structure function \( F_2 \) corresponding to turbulent flows with three, two or one homogenous direction, and found that all of them are capable of giving a correct near wall behaviour of SGS eddy viscosity without requiring any damping function.

Firstly, for three-dimensional homogeneous and isotropic small scale turbulence, such as free turbulence, a three-dimensional (3-D) resolved structure function with \( \Delta x = \Delta y = \Delta z \) can be used, which takes the following form

\[ F_2(\Delta, M) = \frac{1}{6} \left[ |\bar{u}(M) - \bar{u}(M_B)|^2 + |\bar{u}(M) - \bar{u}(M_W)|^2 \right. \]

\[ + |\bar{u}(M) - \bar{u}(M_N)|^2 + |\bar{u}(M) - \bar{u}(M_S)|^2 \]

\[ + |\bar{u}(M) - \bar{u}(M_T)|^2 + \left. |\bar{u}(M) - \bar{u}(M_E)|^2 \right] \]  

(2.44)
where $\Delta = (\Delta x \Delta y \Delta z)^{1/3}$. $M_N$, $M_S$, $M_W$, and $M_E$ are the four neighbouring points around $M$ in the same $xz$ plane, and $M_T$ and $M_B$ are the two neighbours of $M$ which have the same horizontal coordinates $(x, z)$.

Secondly, in turbulent flows with two homogeneous directions, such as the $x$ and $z$ directions in a channel flow, the small scales are homogeneous and isotropic in all $xz$ planes. A two-dimensional (2-D) resolved structure function can be employed [10],

$$
\bar{F}_2(\Delta, M) = \frac{1}{4} \left[ \left( \frac{\Delta}{\Delta x} \right)^2 (|\bar{u}(M) - \bar{u}(M_S)|^2 + |\bar{u}(M) - \bar{u}(M_W)|^2) + \left( \frac{\Delta}{\Delta z} \right)^2 (|\bar{u}(M) - \bar{u}(M_T)|^2 + |\bar{u}(M) - \bar{u}(M_B)|^2) \right]^{(2.45)}
$$

where $\Delta = \min(\Delta x, \Delta z)$.

Thirdly, if the smallest resolved scale is homogeneous and isotropic only in one direction, the corresponding structure function defined is in one-dimensional (1-D) form [10]

$$
\bar{F}_2(\Delta, M) = \frac{1}{2} \left[ \left( \frac{\Delta}{\Delta y_T} \right)^2 |\bar{u}(M) - \bar{u}(M_T)|^2 + \left( \frac{\Delta}{\Delta y_B} \right)^2 |\bar{u}(M) - \bar{u}(M_B)|^2 \right]^{(2.46)}
$$

where $\Delta y_T = |y_{M_T} - y_M|$, $\Delta y_B = |y_{M_B} - y_M|$, and $\Delta = \min(\Delta y_T, \Delta y_B)$.

This model has been applied in various flow configurations. For example, Silveira et al. [58] simulated the flow past a backward-facing step. It was also utilized by Normand [43] to simulate weakly-compressible isotropic turbulence and compressible boundary layer over a flat plate at an external Mach number of 5. Later Comte et al. [10] performed a priori tests in the case of an incompressible direct turbulent channel flow simulation. He found that the eddy viscosity and dissipation went practically to zero at the walls by using his specially designed one-, two- and three-dimensional resolved structure functions, without requiring any
damping function. Comte et al. [10] also applied this model in the simulations of compressible sheared turbulence at higher Mach number than before and suggested that the model enables the simulation of realistic compressible flows, provided the turbulent Mach number and the density gradients are not too high, for example, initial r.m.s. Mach numbers are not larger than 0.8.

In summary, the second-order velocity function model has the advantage of being derived from a local isotropy assumption and designed to transfer energy from grid scales to subgrid scales along a Kolmogorov cascade at a rate corresponding to the local kinetic energy dissipation. In addition, the model takes intermittency of flows into account better, since $\nu_t$ is defined locally. Also, all of the three formulations of the model (3-D (2.44), 2-D (2.45) and 1-D (2.46)) make the eddy viscosity go to zero at the walls, without using any damping function. However, it should be noted that the second-order velocity structure function model is based upon the assumption that turbulence is fully developed at small scales, with a $k^{-5/3}$ energy spectrum. For turbulence at a moderate or low Reynolds number this is certainly not the case. On the other hand, there is no guide-line for the calculation of the second-order velocity structure function for turbulent flows where no homogeneous direction exists. Under these circumstances, this model loses its advantage of not requiring any damping function with respect to the Smagorinsky model. Furthermore, the second-order velocity structure function model is basically of the eddy viscosity type. Its absolute positive value of eddy viscosity, like the Smagorinsky eddy viscosity, fails to account for energy transfer from the smaller to the larger eddies, i.e. backscatter. Taking all these facts into consideration, the present study will not use this model in the LES being performed later.
Bardina’s model

As the understanding of turbulence mechanism grows, new SGS models are constantly emerging. Every effort made is to provide a way of circumventing the existing deficiencies of SGS modelling. Here is presented the mixed model suggested by Bardina et al. [2, 3]. This model was developed to limit the stress-strain correlation imposed by the Smagorinsky model, or other models of eddy viscosity type.

The mixed model is a linear combination of the Smagorinsky model and scale-similarity model. A basic assumption for deriving the scale-similarity model is that the net rate of energy transfer from the filtered flow field to the SGS flow field is determined by the eddies whose scales lie just below the filter width. It is these eddies, the largest SGS ones, which require modelling.

By analogy to the method used to decompose the full flow field, Bardina et al. decomposed the filtered and SGS flow fields and found that the largest scales of the SGS flow field are similar in structure to the smallest resolved scales.

Upon applying a second filter, which is coarser than the first one, to the resolved flow field, we can obtain the larger flow field \( \overline{u}_i \), which contains the larger eddies of the resolved flow field. Then subtraction of \( \overline{u}_i \) from \( u_i \) results in the smallest eddies of resolved flow field \( u_i - \overline{u}_i \).

On the other hand, filtering the SGS flow field gives the largest SGS eddies \( \overline{u}'_i \). According to the definition of the decomposition (2.9), \( u'_i \) can be expressed as

\[
u'_i = u_i - \overline{u}_i . \tag{2.47}\]

Then \( \overline{u}'_i \) becomes

\[
\overline{u}'_i = u_i - \overline{u}_i = u_i - \overline{u}_i . \tag{2.48}\]

The identity of the equations (2.48) and the definition of the smallest eddies of resolved flow field \( u_i - \overline{u}_i \) shows that the smallest resolved eddies are simultaneously
2.2 Subgrid-scale modelling

the largest SGS eddies. Both of them can be represented by \( \overline{u}_i - \overline{u} \), which was defined as the transfer flow field by Bardina. Assuming the main interaction between resolved and SGS eddies takes place in the transfer flow field, Bardina et al. modelled each term of SGS stress tensor in terms of the transfer and the larger velocity fields as follows

\[
R_{ij} = \overline{u}'_i \overline{u}'_j \approx \overline{u}'_i \overline{u}'_j = (\overline{u}_i - \overline{u}_i) (\overline{u}_j - \overline{u}_j),
\]

\[
C_{ij} = \overline{u}'_i \overline{u}_j + \overline{u}_i \overline{u}'_j \approx \overline{u}'_i \overline{u}_j + \overline{u}'_i \overline{u}_j = (\overline{u}_i - \overline{u}_i) \overline{u}_j + \overline{u}_i (\overline{u}_j - \overline{u}_j),
\]

\[
R_{ij} + C_{ij} \approx \overline{u}_i \overline{u}_j - \overline{u}_i \overline{u}_j.
\]

By comparing the average correlation coefficients between the “exact” and model values of the SGS Reynolds stresses from eddy viscosity and the scale-similarity models for homogeneous isotropic turbulence and for homogeneous turbulence in the presence of mean shear, Bardina et al. [3] found that the scale-similarity model correlates well with the SGS Reynolds stresses locally, but does not dissipate sufficient energy, while the Smagorinsky model gives poor representation of Reynolds stresses on a local basis, though it is able to maintain the correct mean energy balance of the large-scale flow field. On the other hand, their results also show that the correlation between the scale-similarity and the Smagorinsky model is almost zero at all levels (i.e. tensor, vector and scalar levels). So Bardina et al. [3] speculated that the linear combination might be a desirable SGS model

\[
\tau_{ij} = -2 \nu \overline{\tau}_{ij} + C_r (\overline{u}_i \overline{u}_j - \overline{u}_i \overline{u}_j) - \frac{1}{3} \delta_{ij} (\overline{u}_n \overline{u}_n - \overline{u}_n \overline{u}_n)
\]

where \( C_r = 1.1 \) and \( \nu \) is the Smagorinsky eddy viscosity. More recent work [60] has shown that the constant \( C_r \) must be unity to ensure Galilean invariance (further details will be given in the next section). The model obtained is called the mixed model. In the mixed model the major function of the scale-similarity component is to transfer energy from the smaller resolved scales to the larger ones, whereas
the eddy-viscosity component is to account for the proper energy dissipation. The performance of the mixed model was examined in the simulations of homogeneous isotropic turbulence in a rotating coordinate frame, and of sheared turbulence [3]. The results show that the mixed model predicts turbulence statistics better than eddy viscosity models. Also, in the investigation of the relationship between the filter and the SGS model, Piomelli, Ferziger and Moin [48] found that the mixed model gives more accurate results than the Smagorinsky model when a Gaussian filter is used, while the Smagorinsky model is fairly accurate when coupled with a sharp Fourier cutoff filter.

Although the modelled SGS stresses calculated by using this model demonstrated a better representation of SGS stresses on a local basis [3], there are some problems related to this model to be addressed. Firstly, this model was not Galilean-invariant in its original form. Secondly, the value of model constant \( C_s \), obtained by a combination of least squares fitting and small adjustments to make LES fit experimental data, was found to be smaller than that of the pure Smagorinsky model [3]. The first problem has been overcome completely by setting the scale-similarity model constant \( C_s = 1 \), but no such easy solution for the second problem has been found. The reduction of the value of \( C_s \) involved in this model is considered as the result of the low subgrid drain produced by the scale-similarity component of the model. Owing to the difficulties in determining the amount of the subgrid drain resulting from the scale-similarity component of the mixed model for various turbulent flows, we therefore speculate that it will be even more difficult to adjust \( C_s \) involved in the mixed model from flow to flow than that in the Smagorinsky model; thus this model shares the same limitation as the Smagorinsky model. For this reason, the work on the mixed model is not taken further in the present study.
Generalized turbulent stresses

An examination of various models for the SGS stresses showed [60] that the SGS cross-stresses are not Galilean-invariant. In that case, if the resolved terms of the SGS stresses are calculated directly while the SGS cross-stresses and Reynolds stresses are parameterized by a Smagorinsky model which is Galilean-invariant, the equations of motion for the large eddies are not Galilean-invariant. That means this model is inconsistent with the basic physics of the problem, which requires that the description of the turbulence be the same in all inertial frames of reference. To avoid the problem, Germano [21, 22, 23] proposed a new definition of the SGS stresses based on a general statement defining a turbulent stress. The turbulent stress \( \tau(f, g) \) related to two generic functions \( f \) and \( g \) was defined as the difference between the filtered value of their product and the product of their filtered values

\[
\tau(f, g) = \bar{f} \bar{g} - \bar{f} \bar{g} .
\] (2.53)

It is easy to show that the turbulent stress so defined is Galilean-invariant, since

\[
\tau(f + \alpha, g + \beta) = \tau(f, g) , \quad \alpha, \beta = \text{const.}
\] (2.54)

Usually in LES, the SGS stress \( \tau_{ij} \) is given by

\[
\tau_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j .
\] (2.55)

According to the definition (2.53), \( \tau_{ij} \) is the turbulent stress related to the velocity components \( u_i \) and \( u_j \), and is Galilean-invariant. But the individual terms forming the turbulent stress (resolved term, \( L_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j \); cross term, \( C_{ij} = \bar{u}_i \bar{u}_j + \bar{u}_i \bar{u}_j \); and Reynolds stress, \( R_{ij} = \bar{u}_i \bar{u}_j \)) are not turbulent stresses any more. Therefore, Germano redefined these terms as

\[
L_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j
\] (2.56)

\[
C_{ij} = \bar{u}_i \bar{u}_j + \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j
\] (2.57)
\[ \mathcal{R}_{ij} = \overline{u'_i u'_j} - \overline{u'_i u'_j} \]  
\hspace{1cm} (2.58)

where \( \mathcal{L}_{ij} \), \( C_{ij} \) and \( \mathcal{R}_{ij} \) are the modified resolved stress, the modified cross stress and the modified Reynolds stress respectively. It is easy to prove that

\[ \tau_{ij} = \mathcal{L}_{ij} + C_{ij} + \mathcal{R}_{ij} \]  
\hspace{1cm} (2.59)

and as stated earlier, it is Galilean-invariant.

**Dynamic subgrid-scale eddy viscosity model**

Several problems related to the Smagorinsky model have received attention over the years:

1) The Smagorinsky model coefficient is input *a priori*. This single universal constant is incapable of representing correctly various turbulent flows.

2) An *ad hoc* damping function is used to obtain the predicted SGS stresses with proper asymptotic near wall behaviour.

3) The length scale to be used with an anisotropic grid is unclear.

4) The eddy viscosity does not vanish in the laminar regime.

5) The backscatter of energy transfer upgrid is ruled out completely.

It is believed that the dynamic SGS model recently developed by Germano *et al.* [20] is able to provide a way of circumventing most of the deficiencies just addressed above, thereby eliminating the corresponding difficulties.

The most promising feature of the new model to be presented shortly is that the model coefficient is computed by using the information of the smallest resolved scales, in principle at each point in space and at each instant in the time integration,
rather than input \textit{a priori}. This dynamically determined model coefficient is able to adjust itself to flow conditions without any \textit{ad hoc} modifications like damping functions near walls. The idea of considering the smallest resolved scales as the principle scales in constructing the SGS model is the same as that emerging from the mixed model, and is consistent with the concept of energy cascade.

Germano \textit{et al.} [20] derived the formulation of the dynamic SGS model by making judicious use of the nested grid, which is obtained through filtering the velocity field twice, first by a grid filter, and then by a coarser filter, called the "test" filter. For incompressible flow, the Navier-Stokes equations filtered on these two levels are given as

\[
\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j} (\bar{u}_i \bar{u}_j) = -\frac{\partial \bar{p}}{\partial x_i} - \frac{\partial \bar{T}_{ij}}{\partial x_j} + \nu \nabla^2 \bar{u}_i \tag{2.60}
\]

\[
\frac{\partial \hat{u}_i}{\partial t} + \frac{\partial}{\partial x_j} (\hat{u}_i \hat{u}_j) = -\frac{\partial \hat{p}}{\partial x_i} - \frac{\partial \hat{T}_{ij}}{\partial x_j} + \nu \nabla^2 \hat{u}_i \tag{2.62}
\]

where overbars represent the grid-filtered field, and caret over the overbars the test-filtered field; \(\bar{\tau}_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j\) (SGS stresses at the grid level) and \(\hat{\tau}_{ij} = \hat{u}_i \hat{u}_j - \hat{u}_i \hat{u}_j\) (SGS stresses at the "test" level). By assuming that the same functional form (the Smagorinsky model) can be used to model both \(\hat{T}_{ij}\) and \(\bar{T}_{ij}\), Germano wrote the subgrid-scale stresses in the following forms

\[
\tau_{ij} - \frac{1}{3} \delta_{ij} \tau_{kk} = 2C_A S \bar{S}_{ij} \tag{2.64}
\]

\[
T_{ij} - \frac{1}{3} \delta_{ij} T_{kk} = 2C_A \hat{S} \hat{S}_{ij} \tag{2.65}
\]

in which the model parameter \(C\) is equivalent to \(C_s^2\), where \(C_s\) is the original Smagorinsky constant; \(\Delta\) and \(\hat{\Delta}\) are the grid and the "test" filter widths respectively;
\[ \tilde{S}^2 = 2\hat{S}_{ij}\hat{S}_{ij} \]. Both \( \tau_{ij} \) and \( T_{ij} \) contain unresolved terms which require modelling involving an \textit{a priori} model constant in the previous methods, but their difference

\[ T_{ij} - \hat{\tau}_{ij} = \hat{\bar{u}}_i\hat{\bar{u}}_j - \hat{\bar{u}}_i\hat{\bar{u}}_j \]  

is computable from the filtered fields at the grid and "test" levels. Defining \( \bar{L}_{ij} \) as the resolved stresses, we have

\[ \bar{L}_{ij} = \hat{\bar{u}}_i\hat{\bar{u}}_j - \hat{\bar{u}}_i\hat{\bar{u}}_j = T_{ij} - \hat{\tau}_{ij} \].

The substitution of (2.64) and (2.65) into (2.67) yields

\[ \bar{L}_{ij} - \frac{1}{3} \delta_{ij} \bar{L}_{kk} = 2C\Delta^2\hat{\bar{S}}_{ij} - 2C\Delta^2\hat{\bar{S}}_{ij} \].

Upon contracting (2.68) with \( \overline{S}_{ij} \), Germano obtained the formulation for the model coefficient

\[ C = \frac{\bar{L}_{ij} \overline{S}_{ij}}{2\Delta^2\hat{\bar{S}}_{ij} - 2\Delta^2\hat{\bar{S}}_{ij}} \].

In order to deal with the problem of numerical instabilities encountered when using the local unaveraged version of (2.69), Germano averaged the numerator and the denominator over a plane parallel to the wall, assuming that \( C \) is only a function of the distance normal to the wall and time. Therefore (2.69) can be rewritten as

\[ C = \frac{\langle \bar{L}_{ij} \overline{S}_{ij} \rangle}{\langle 2\Delta^2\hat{\bar{S}}_{ij} - 2\Delta^2\hat{\bar{S}}_{ij} \rangle} \].

After the dynamic subgrid-scale model had been used in several large eddy simulations, some drawbacks of the Germano model have been recognized [54]. Firstly, the selection of \( \overline{S}_{ij} \) for contracting tensor to obtain a scalar equation of the model coefficient \( C \) is somehow arbitrary. Secondly, numerical instabilities are encountered because the denominator of the formulation of \( C \) could vanish or become very small. Lilly [35] has addressed these problems. He obtained a natural choice for...
the contracting tensor needed to derive a scalar version of the equation (2.68) by employing a least squares technique so that the problem of overspecification due to the tensorial nature of (2.68) was overcome. Defining \( Q \) to be the square of the error in (2.67) and still following the Smagorinsky formulation of the SGS model on both grid and “test” scale levels yields

\[
Q = \left[ \tilde{L}_{ij} - \frac{1}{3} \delta_{ij} \tilde{L}_{kk} - 2CM_{ij} \right]^2
\]

(2.71)

where \( M_{ij} = \left( \Delta^2 \tilde{S}_{ij} - \Delta^2 \tilde{S}_{ij} \right) \).

Since \( \partial^2 Q / \partial^2 C > 0 \),

\[
C = \frac{\tilde{L}_{kl} M_{kl}}{2M_{ij}M_{ij}}
\]

(2.72)

derived from \( \partial Q / \partial C = 0 \) represents the minimum of \( Q \). Now the denominator of (2.72) can vanish only if each of its five independent components vanish simultaneously, so the modified dynamic subgrid-scale model (hereafter referred to as the Germano-Lilly model) appears to have provided a guideline for removing a source of singularity. But the problem of computational instability remains if the local unaveraged version of equation (2.72) is used. It was found that the instability is attributed to high negative and positive values of \( C \) at some positions in the domain sustained over many time steps. To avoid this problem, Akselvoll and Moin [1] suggested that some form of averaging method should be applied to the equation (2.72)

\[
C = \frac{< \tilde{L}_{kl} M_{kl} >}{2 < M_{ij} M_{ij} >}
\]

(2.73)

where \(< \cdot >\) indicates that the quantities are averaged over a homogenous direction or a homogenous plane. In addition, a time-averaging method can also be used.

Germano et al. [20] applied the dynamic model to transitional and fully turbulent flow. Their results have verified that the model results in the correct asymptotic behaviour in the near-wall region without any ad hoc damping functions which are
often required when using standard subgrid scale models. In addition to this advantage, there is another favourable aspect of the model, i.e. the sign of \( C \) can locally become negative. This indicates the ability of the model to account for energy backscatter.

Considering the advantages of the dynamic SGS model, a lot of researchers have applied it to some complex flows where the Smagorinsky model does not work successfully. For example, backward facing step flows [1], turbulent cavity flows [69], stratified Ekman layers [5], turbulent recirculating flows [70] and rotating turbulent flows [61]. The dynamic model showed generally better agreement with direct numerical simulations. Some authors also extended the model to simulate compressible turbulence [42, 16] though the model was derived from incompressible turbulence. Their results showed that the dynamic model is able to provide good descriptions of highly compressible turbulence.

There is a further issue related to the dynamic model. As noted from the formulation presented earlier in this section, the ratio of “test” filter width to grid filter width remains an input to the dynamic model. In general, the ratio is expected to affect the results of the simulations because if its value is too small, the equation (2.73) contains little information and the resolved turbulent stresses can be contaminated by numerical errors, and if it is too large, important local information is averaged away. The results of transitional and turbulent channel flow simulations performed by Germano et al. [20] have shown insensitivity to the value of the ratio. However Cabot and Moin [7] found that low Reynolds number LES results for channel flow are more sensitive to the choice of the ratio. A modified dynamic model used in the simulations to be performed in the present study, which suits the low Reynolds number channel flows better, will be given in chapter 3.
2.3 Near-wall model

For a wall bounded turbulent flow, the turbulence length scales decrease as the wall is approached because the growth of turbulence structures is inhibited by the wall. This requires that the eddy viscosity and its derivatives must vanish at the wall.

Since the reduction of the turbulence length scales may cause them to become smaller than the filter size in the near-wall region, the definition of a length scale for the subgrid scales becomes less straightforward due to the anisotropies introduced by the solid wall than in isotropic flows. In this case, modelling of the flow in the vicinity of the wall (the near-wall model) becomes an issue of great importance in the simulation of high Reynolds number or complex flows when the Smagorinsky subgrid-scale model and the like are used.

Kim, Moin and Moser’s [29] results from a direct simulation of a turbulent channel flow showed that the normal velocity component $v$ of the fluctuation velocity approaches zero at the wall as $y^2$ ($y$ is the distance from the wall), whereas the components $u$ and $w$, parallel to the wall, have $y$ linear behaviour when the wall is approached. This is consistent with the observation that eddies near the wall stretch in the $x$- and $z$-directions, parallel to the wall. Consequently the turbulence shear stress $\tau_{12}$ should have $y^3$ behaviour. But the Smagorinsky model subgrid-scale stress is not capable of reflecting this limiting wall behaviour. For example, for a stretched grid spacing in the $y$-direction, and constant grid spacing in the $x$- and $z$-directions, the choice $\Delta = (\Delta_x \Delta_y \Delta_z)^{1/3}$ gives $\tau_{12} = \nu_t(\Delta) \bar{S}_{12} \sim y^{2/3}$, while the choice $\Delta = (\Delta_x^2 + \Delta_y^2 + \Delta_z^2)^{1/2}$ gives $\tau_{12}$ nearly constant in the very near-wall region. The failure of the Smagorinsky model to represent wall turbulence accurately in the near-wall region is not surprising because the assumption used in his model, namely that production and dissipation are in balance, is no longer valid near the wall, where
viscous diffusion becomes important. This deficiency of the Smagorinsky model can be easily recognized from its non-zero eddy viscosity \( \nu_t(\Delta) = (C_s \Delta)^2 S \) at the wall. To better predict turbulence structures of the near-wall region in simulations of channel flow, some workers introduced various empirical corrections into subgrid-scale modelling besides the stretched computational grid in the normal direction which is usually insufficient to account for the reduction of the turbulence length scales. When the length scale is denoted by \( l \), the eddy viscosity can be rewritten as

\[
\nu_t = l^2 \overline{S}.
\]  

(2.74)

The Smagorinsky model corresponds to \( l = C_s \Delta \). The various near-wall models are empirical formulations of \( l \). A brief review is given in the remainder of the present section.

Moin and Kim [41] used a Van Driest [15] damping function to reduce \( \Delta \) as the wall is approached. This leads to

\[
l = C_s \left[ 1 - \exp \left( -y^+ / A^+ \right) \right] (\Delta_x \Delta_y \Delta_z)^{1/3}.
\]  

(2.75)

Another near-wall model was constructed by Bardina, Forswear and Reynolds [2] as

\[
l = C_s \left[ 1 - \exp \left( -y^+ / A^+ \right) \right] \sqrt{\Delta_x^2 + \Delta_y^2 + \Delta_z^2}.
\]  

(2.76)

An alternative formulation proposed by Mason and Callen [37] is

\[
\frac{1}{l} = \frac{1}{C_s (\Delta_x \Delta_y \Delta_z)^{1/3}} + \frac{1}{\kappa (1 - |y|)}
\]

(2.77)

in which \( \kappa \) is the Von Kármán constant.

Piomelli, Ferziger and Moin [48] suggested an expression of the form

\[
l = C_s \left[ 1 - \exp \left( -y^{+3} / A^{+3} \right) \right]^{1/2} (\Delta_x \Delta_y \Delta_z)^{1/3}.
\]  

(2.78)
Except (2.77) all of the above expressions are in the form of exponential damping. Among them (2.78) is expected to be more successful in simulating the wall layer because of the advantage that it ensures the proper behaviour for the SGS Reynolds stress $\tau_{12}$ near the wall. But it appears that simulations of channel flows are not very sensitive to the precise form of the damping function, and all the theoretical treatments are fairly satisfactory in LES of channel flows. The reason for that may be [48] that in channel flow the linear shear stress constitutes a strong constraint on the flow, and the large structures have to adjust themselves to modelling errors to yield the correct total shear stress. In other flows (boundary layers, for example) that might not be the case. Under this circumstance, a different formulation of the length scale, like the dynamic SGS model, may be necessary to resolve more accurately the wall layer.

Note that a damping function of any type is only required in those simulations in which the wall layer is resolved. When the Reynolds number is high enough that the first grid point is located beyond the linear sublayer ($y^+ < 5$), where the boundary condition is a natural, no-slip condition, the wall layer has to be modelled by approximate boundary conditions. For example, when the nearest mesh point to the wall is very much larger than the height of the viscous (or linear) sublayer, say $y^+ > 30$, which is the case of a classical LES at high Reynolds number, the mean flow at the first grid points should obey the logarithmic "law of the wall". In spite of the absence of any theoretical explanation, the following logarithmic law has been used in many simulations of channel flows satisfactorily for $y^+ > 30$

$$u^+ = 5.5 + 2.5 \log(y^+)$$

(2.79)

where $y^+ = y u_r / \nu$, $u^+ = u / u_r$ and $u_r$ is the wall shear velocity. Besides the above cases, there is also another case corresponding to an intermediate resolution, i.e. the nearest mesh point to the wall lies neither in the linear sublayer ($y^+ < 5$, full or nearly full simulations) nor well out in a logarithmic layer ($y^+ > 30$, simulations...
2.4 Subgrid-scale model with $k_c$ not in the $k^{-5/3}$ range

at high or infinite Reynolds numbers). An adaptive boundary condition used by Voke [66] has included this buffer zone. It is an approach to smoothly switching the natural no-slip condition to the "log law" condition through the Von Kármán interpolation. It is in the following form

$$u^+ = \begin{cases} 
  y^+ & y^+ < 5 \\
  5 + 5 \log (y^+ / 5) & 5 < y^+ \leq 30 \\
  5.5 + 2.5 \log (y^+) & y^+ > 30 
\end{cases} \quad (2.80)$$

2.4 Subgrid-scale model with $k_c$ not in the $k^{-5/3}$ range

In the classical LES, the grid-scale cutoff falls in an inertial range. However, the classical arguments are not applicable for a grid-scale cutoff in the dissipative range, in which the viscous dissipation is important and no longer negligible compared to eddy viscosity. Since dimensional analysis does not result in a precise description of the energy spectrum $E(k, t)$ within the dissipative range, we must try to solve the dynamic equation

$$\frac{\partial}{\partial t} E(k, t) = T(k, t) - 2\nu k^3 E(k, t) \quad (2.81)$$

$$\frac{\partial}{\partial t} \int_0^k E(k, t) \, dk = \int_0^k T(k, t) \, dk - 2\nu \int_0^k k^2 E(k, t) \, dk \quad (2.82)$$

where $T(k, t)$ is energy transfer function and $\nu$ is the kinematic viscosity. In the equilibrium range where $(k \gg k_c, k_c$ is energy containing wavenumber), the term $\frac{\partial E(k, t)}{\partial t}$ is very small, and consequently $\frac{\partial}{\partial t} \int_0^k E(k, t) \, dk$ is negligible with respect to $\frac{\partial}{\partial t} \int_0^\infty E(k, t) \, dk$. Hence, we can obtain

$$\frac{\partial}{\partial t} \int_0^k E(k, t) \, dk \approx \frac{\partial}{\partial t} \int_0^\infty E(k, t) \, dk$$

$$= -\epsilon$$

$$= -2\nu \int_0^\infty k^2 E(k, t) \, dk \quad (2.83)$$
2.4 Subgrid-scale model with $k_c$ not in the $k^{-5/3}$ range

where $\varepsilon$ is the rate that the turbulent energy per unit mass is fed to the small scales from the large scales.

According to (2.83), we can write (2.82) as

$$-\varepsilon = \int_0^k T(k,t)dk - 2\nu \int_0^k k^2 E(k,t)dk$$

or

$$\int_0^k T(k,t)dk + 2\nu \int_k^\infty k^2 E(k,t) dk = 0.$$  

(2.85)

To obtain the energy spectrum in the equilibrium range, several proposals based on assuming some relation among the transfer function $T(k,t)$, the energy spectrum $E(k,t)$, and the wavenumber $k$, have been made to solve the above equations.

Obukhoff [27] assumed that the energy transfer across the wavenumber $k$ is analogous to the expression for the turbulent energy production in a shear flow and then he obtained

$$\int_0^k T(k,t)dk = -\alpha \left[2 \int_0^k k^2 E(k,t)dk\right]^{1/2} \int_k^\infty E(k,t)dk.$$  

(2.86)

By substituting (2.86) into (2.85), we can obtain the following equation

$$\alpha \left[2 \int_0^k k^2 E(k,t)dk\right]^{1/2} \int_k^\infty E(k,t)dk = 2\nu \int_k^\infty k^2 E(k,t)dk.$$  

(2.87)

Setting

$$\theta = 2 \int_0^k k^2 E(k,t)dk,$$

(2.88)

we can obtain

$$\frac{d\theta}{dk} = 2k^2 E(k,t)$$

$$E(k,t) = \frac{1}{2k^2} \frac{d\theta}{dk}.$$  

(2.89)

Substituting (2.88) into (2.87), we can obtain

$$\alpha \theta^{1/2} \int_k^\infty E(k,t)dk = \varepsilon - \nu \theta.$$  

(2.90)
2.4 Subgrid-scale model with \( k_c \) not in the \( k^{-5/3} \) range

Differentiation with respect to \( k \) and replacement of \( E(k,t) \) by (2.89) yield

\[
\theta^{-3/2} + \frac{\nu}{\epsilon} \theta^{-1/2} - \frac{\alpha}{k^2 \epsilon} = 0
\]

(2.91)

the solution of which is

\[
\theta = \left[ \left( \frac{\alpha}{2k^2 \epsilon} + \sqrt{\frac{\alpha^2}{4k^4 \epsilon^2} + \frac{1}{27e^2 k_d^4}} \right)^{1/3} + \left( \frac{\alpha}{2k^2 \epsilon} - \sqrt{\frac{\alpha^2}{4k^4 \epsilon^2} + \frac{1}{27e^2 k_d^4}} \right)^{1/3} \right]^{-2}
\]

(2.92)

where \( k_d = (\epsilon/\nu^2)^{1/4} \). From the above equation and (2.89), an energy spectrum function can be deduced. When \( k \) falls in the inertial subrange, i.e. \( k \ll k_d \),

\[
\theta = \left( \frac{\epsilon}{\alpha} \right)^{2/3} k_d^{4/3}
\]

(2.93)

and then we can obtain

\[
E(k,t) = \frac{2}{3} \alpha^{-2/3} \epsilon^{2/3} k^{-5/3}
\]

(2.94)

which agrees with the Kolmogorov spectrum.

On the other hand, since \( \theta = 2 \int_0^k k^2 E(k,t) dk \), we should obtain \( \theta = \epsilon/\nu \) with \( k \to \infty \). But with the above expression (2.92), when \( k \to \infty \), \( \theta \to \infty \). So the spectrum function must be assumed to fall discontinuously to zero at a certain value of \( k \), at which the total dissipation is equal to \( \epsilon \). Therefore, we can assert that there must be something wrong with the assumption (2.86).

Ellison modified Obukhoff's assumption by suggesting [27]

\[
\int_0^k T(k,t) dk = -\alpha E(k,t) k \left[ 2 \int_0^k k^2 E(k',t) dk' \right]^{1/2}
\]

(2.95)

The substitution of (2.95) in (2.85) gives

\[
k E(k,t) \left[ 2 \int_0^k k^2 E(k',t) dk' \right]^{1/2} = \frac{2\nu}{\alpha} \int_k^\infty k''^2 E(k'',t) dk''
\]

(2.96)

when \( k \) falls in the inertial subrange, the right hand of (2.96) becomes equal to \( \epsilon/\alpha \). Putting \( \theta(k,t) = \left[ 2 \int_0^k k^2 E(k',t) dk' \right]^{1/2} \), thereby \( \frac{d\theta}{dk} = k^2 E(k,t) \), gives the following
2.4 Subgrid-scale model with $k_c$ not in the $k^{-5/3}$ range

differential equation for $\theta$

$$\theta^2 d\theta = \frac{\varepsilon}{\alpha} k \, dk. \quad (2.97)$$

From the equation (2.97) with boundary condition $\theta(0, t) = 0$, $\theta$ is easily obtained

$$\theta = \left( \frac{3\varepsilon}{2\alpha} \right)^{1/3} k^{2/3}. \quad (2.98)$$

Then the final solution for $E(k, t)$ is in the form

$$E(k, t) = \left( \frac{2}{3\alpha^2} \right)^{1/3} \varepsilon^{2/3} k^{-5/3}. \quad (2.99)$$

As expected, this agrees with the Kolmogorov spectrum, for wavenumbers $k \ll k_d$.

Kovasznay [27] gave an alternative assumption that $\int_0^k T(k, t) \, dk$ is a function only of $E(k, t)$ and $k$, which reads

$$\int_0^k T(k, t) \, dk = -\alpha [E(k, t)]^{3/2} k^{5/2}. \quad (2.100)$$

By substituting (2.100) in (2.84), the equation (2.84) can be written as

$$\varepsilon = \alpha [E(k, t)]^{3/2} k^{5/2} + 2\nu \int_0^k k^3 E(k, t) \, dk. \quad (2.101)$$

Assume

$$\theta(k, t) = \int_0^k k^2 E(k, t) \, dk \quad (2.102)$$

with boundary condition $\theta(0, t) = 0$. Equation (2.101) then reads

$$\varepsilon - 2\nu \theta = \alpha [E(k, t)]^{3/2} k^{5/2} \quad (2.103)$$

the solution of which is

$$\theta = \frac{\varepsilon - (\varepsilon^{1/3} - \nu \alpha^{-2/3} k^{4/3})^3}{2\nu}. \quad (2.104)$$

Hence, $E(k, t)$ can be written as

$$E(k, t) = \left( \frac{\varepsilon}{\alpha} \right)^{2/3} k^{-5/3} \left[ 1 - \frac{\alpha^{-2/3}}{2} \left( \frac{k}{k_d} \right)^{4/3} \right]^2. \quad (2.105)$$
When $k \ll k_d$, i.e. $k$ falls in the inertial subrange, (2.105) agrees with the Kolmogorov spectrum. With the above energy spectrum, we can also obtain the dissipation integral
\[
2\nu \int_0^k k^2 E(k, t) dk = 2\nu \int_0^k k^2 \left( \frac{\epsilon}{\alpha} \right)^{2/3} k^{-5/3} \left[ 1 - \frac{\alpha^{-2/3}}{2} \left( \frac{k}{k_d} \right)^{4/3} \right]^2 \, dk
\]
\[
= \epsilon \left\{ 1 - \left[ 1 - \frac{\alpha^{-2/3}}{2} \left( \frac{k}{k_d} \right)^{4/3} \right]^3 \right\}^{2/3}
\]
when $1 - \frac{\alpha^{-2/3}}{2} \left( \frac{k}{k_d} \right)^{4/3} = 0$, that is, $k = \left( 2\alpha^{2/3} \right)^{3/4} k_d$, the above integral is equal to $\epsilon$. That means that Kovasznay's deduced spectrum, like Obukhov's, also has a cutoff value of $k$, that is, at this certain cutoff value of $k$, the total dissipation is already equal to $\epsilon$ and therefore the value $E(k, t)$ must be zero for all wavenumbers higher than $k = \left( 2\alpha^{2/3} \right)^{3/4} k_d$. This form of dissipation-range spectrum would appear to be erroneous since the spectrum is expected to fall smoothly below the $k^{-5/3}$ range, and eventually enable the integral $2\nu \int_0^k k^2 E(k) \, dk = \epsilon$ to be fulfilled.

Another assumption was suggested by Heisenberg [27], who accounted for the effect of small eddies as equivalent to an eddy viscosity and obtained
\[
\int_0^k T(k, t) \, dk = -2\epsilon(k, t) \int_0^k k^2 E(k', t) \, dk'
\]
(2.107)
where $\epsilon(k, t)$ represents the kinematic turbulence viscosity due to the eddies with wavenumbers ranging from $k$ to infinity. Dimensional analysis gives
\[
\epsilon(k, t) = \text{const.} \int_k^\infty \frac{E(k'', t)}{k''^{5/3}} \, dk''
\]
(2.108)
and hence
\[
\int_0^k T(k, t) \, dk = -2\alpha \int_k^\infty \frac{E(k'', t)}{k''} \, dk'' \int_0^k k^2 E(k', t) \, dk'.
\]
(2.109)
The substitution of (2.109) in (2.85) gives [27]
\[
E(k, t) = \left( \frac{8}{9\alpha} \right)^{2/3} \left( \epsilon \nu \right)^{1/4} \left( \frac{k}{k_d} \right)^{-5/3} \left[ 1 + \left( \frac{8}{3\alpha^2} \right) \left( \frac{k}{k_d} \right)^{4} \right]^{-4/3}
\]
(2.110)
2.4 Subgrid-scale model with $k_c$ not in the $k^{-5/3}$ range

When $k \ll k_d$, i.e. $k$ lies in the inertial subrange, (2.110) reduces to the Kolmogorov spectrum. But when $k \gg k_d$ that is, in the viscous dissipation range, (2.109) becomes

$$E(k, t) = \left(\frac{\alpha^2}{2}\right)^2 (\epsilon \nu^5)^{1/4} \left(\frac{k}{k_d}\right)^{-7}$$

(2.111)

with equation (2.110), we calculate the dissipation integral as follows

$$\epsilon = 2\nu \int_0^\infty k^2 \left(\frac{8}{9\alpha}\right)^{2/3} (\epsilon \nu^5)^{1/4} \left(\frac{k}{k_d}\right)^{-5/3} \left[1 + \left(\frac{8}{3\alpha^2}\right) \left(\frac{k}{k_d}\right)^4\right]^{-4/3} dk$$

$$= 2\nu \int_0^\infty \left(\frac{8}{9\alpha}\right)^{2/3} (\epsilon \nu^5)^{1/4} k_d^{5/3} k^{1/3} \left[1 + \left(\frac{k}{k_d}\right)^4 \left(\frac{8}{3\alpha^2}\right)\right]^{-4/3} dk . \quad (2.112)$$

From (2.112), we can find that a significant contribution to the above dissipation integral is from the transition range of value of $k$ lying in the neighbourhood of $k = (3\alpha^2/8)^{1/4} k_d \approx k_d$, which agrees with the interpretation that the wavenumber $k_d = 1/\eta$ marks the location of the range of strong viscous dissipation.

The notion of Heisenberg’s suggestion is compatible with the energy cascade since he assumed that the effect of small eddies is equivalent to an eddy viscosity. But there exists a fundamental conflict in a physical point of view between his energy transfer theory and the requirement of statistical independence between smaller and larger eddies. This requirement, as Hinze [27] has pointed out, is essential for using a turbulence viscosity to account for the transfer of energy from the larger to the smaller eddies. Yet Heisenberg’s assumption (2.109) is in such a form that the main contribution to the two integrals on the right-hand side are from the eddies with the wavenumber $k$ close to the lower limit of the first integral (which represents the effect of the smaller eddies with wavenumber $k''$ on the eddy with wavenumber $k$) and close to the upper limit of second integral (which represents the transfer of the energy from the larger eddies with wavenumber $k'$ to the eddy with wavenumber $k$). On the other hand, some researchers have argued that the Heisenberg energy spectrum in the viscous dissipation range (2.111), which leads to a $k^{-7}$ behaviour,
2.4 Subgrid-scale model with $k_c$ not in the $k^{-5/3}$ range

is less desirable due to its power-law dissipation-range spectrum.

Pao [50] assumed that energy flux is proportional to the energy density $E(k, t)$ at wavenumber $k$, that is

$$\int_0^k T(k, t) dk = f E(k, t)$$

where $f$ is the rate at which the energy is transferred through a cascade process across the wavenumber range, which is assumed to be dependent on $\epsilon$ and $k$. Furthermore, according to dimensional reasoning, Pao obtained

$$\int_0^k T(k, t) dk = \alpha^{-1} \epsilon^{1/3} k^{5/3} E(k, t).$$

The substitution of (2.114) in (2.85) gives

$$\alpha^{-1} \epsilon^{1/3} k^{5/3} E(k, t) = -2\nu \int_k^\infty k''^2 E(k'', t) dk''.$$  

(2.115)

Setting $\theta(k, t) = \int_k^\infty k''^2 E(k'', t) dk''$ with the boundary condition $\theta(0, t) = \epsilon/2\nu$, $E(k, t)$ can be written as

$$E(k, t) = \alpha \epsilon^{2/3} k^{-5/3} \exp\left(-\frac{3}{2} \alpha \nu \epsilon^{-1/3} k^{4/3}\right),$$

(2.116)

or in non-dimensional form

$$E(k, t)/(\epsilon \nu^5)^{1/4} = \alpha \left(\frac{k}{k_d}\right)^{-5/3} \exp\left[-\frac{3}{2} \alpha \left(\frac{k}{k_d}\right)^{4/3}\right].$$

(2.117)

It is easy to show that $E(k, t)$ agrees with the Kolmogorov spectrum for $k< k_d$, that is $k$ lies in the inertial subrange. When $k \gg k_d$, that is $k$ falls in the dissipative range, the exponential factor in Pao's energy spectrum (2.117) is less than 1, thereby representing the reduction in the energy transfer in the dissipation region. In addition, its smooth behaviour above $k_d$ is very attractive.

From all the above dissipation-range spectra, we can see that those having a $k^{-5/3}$ behaviour at the low $k$ end and obeying $\epsilon = 2\nu \int_0^\infty k^2 E(k) dk$, are of utility for
the construction of a subgrid-scale model suitable for low-Reynolds-number flows. Voke [67] has worked on several spectra of the above desired properties, such as the Heisenberg, Kovasznay and Pao spectra, and found the following relation

\[ u \rightarrow C_s \tau - \beta \]  

(2.118)

where \( \tau = \mu / \nu, \tau = \Delta^2 (S^2)^{1/2} / \nu \) with \( S^2 = 2 \overline{S}_{ij} \overline{S}_{ij} \) and \( \beta \) is an addition offset, with which each of the new models based on the Heisenberg, Kovasznay and Pao spectra respectively, has the Smagorinsky model as a parallel asymptote in the high \( \nu \) (and high \( \tau \)).

According to Voke [67], for the Heisenberg spectrum, \( \beta = 1 \), the same asymptote as the modified low-Reynolds-number Smagorinsky model by extending the Kolmogorov spectrum \( E(k) = \alpha \epsilon^{2/3} k^{-5/3} \) to the dissipation-range. For the Kovasznay spectrum, \( \beta = 0.5 \). Since the subgrid viscosity based on this spectrum does not vanish at \( \tau = 0 \) as required, the offset was then subtracted from \( \tau \) and \( \beta = 0.31 \) obtained. This asymptotic behaviour turns out to be very close to that based on the Pao spectrum, for which \( \beta = 0.25 \). After revealing the asymptotic behaviour of the above new models and considering that it is difficult to resolve \( \nu \) explicitly, if not impossible, Voke found the following function that fits the desired behaviour of \( \nu(\tau) \) rather accurately

\[ \nu = \nu_s - \beta [1 - \exp(-\nu_s/\beta)] \]  

(2.119)

where \( \nu_s \) is the Smagorinsky eddy viscosity calculated in the normal way. Clearly, the above formulation has the parallel asymptote \( \nu \rightarrow \nu_s - \beta \) and \( \nu = 0 \) when \( \tau = 0 \).

The value of \( \beta \) was found to be \( \frac{2}{5} \) to give the best fit over the whole range when the new model was applied in the simulations of turbulent channel flows at low Reynolds number [68]. The present study will evaluate its performance.
2.5 The economy of large-eddy simulations

Another great challenge, almost as severe as SGS modelling, is the economy of LES. It is believed that it is difficult to use LES as a real computational tool in practical engineering before its cost is reduced greatly. Most current efforts to accelerate LES are made by improving the numerical methods or grid designing. Another method developed on the basis of understanding of turbulence mechanisms, called the multiple mesh method, was first proposed by Voke [64]. By the multiple mesh method, simulations were performed alternately on a set of two or more nested meshes. Once the coarse mesh fields move forward to a new realisation, the velocity field is interpolated to a fine mesh. Then the simulation is performed on the fine level until the mesh-scale eddies on it are regenerated. By a injection procedure, the fine velocity field is switched to the coarse mesh, and the difference (called the residual velocity field) between the velocity fields on them is stored and remains frozen during the simulation on the coarse mesh. The frozen residual velocity field is later added to the velocity field interpolated back to the fine mesh from the coarse mesh.

The results of Voke’s multiple mesh simulation of turbulent channel flows [64, 65, 68] indicated that the multimesh technique is a cheap way of turning pseudorandom initial velocities into something like real turbulence and is capable of producing results comparable in quality to those that would be actually performed wholly on the fine mesh for a much longer period of time. However, there exists a deficiency in his method, that is adding frozen residual fields to the fine mesh at the moment of interpolation. This interaction is quite unphysical, due to small scale eddies being absent from regions where they should exist and being present elsewhere. This deficiency can be recognized from the computed statistics [68] involving the streamwise velocity fluctuations which show persistent oscillations through lack of correlation.
between the fine mesh and the coarse mesh fields, even after a period of simulation on the fine mesh that is estimated theoretically to be sufficient to regenerate the correlation. The advantage of the multimesh technique is therefore partially traded off. To overcome the deficiency, a new multiple mesh technique avoiding frozen residual field has been proposed by the present study. Its implementation will be presented and tested later.

2.6 Summary

From the foregoing survey of existing SGS models and related issues, it can be seen that there is still considerable scope for development and improvement in the LES method if LES is to become a useful engineering tool, especially for the low Reynolds number situation and complex flow configurations, where most difficulties occur. The present study will therefore concentrate on both improvement of SGS modelling and the economy of the LES method.
3 Mathematical Formulation

In this chapter, the governing equations of turbulent flows are presented, followed by a description of the space-averaging procedure giving rise to terms which have become known as SGS stresses. Then, a discussion on the SGS models involved in the simulations being performed in the present study is given. Finally, details of the numerical method used to solve the governing equations are described.

3.1 Governing equations

For incompressible flows, the Navier-Stokes equations, which are based on the universal laws of momentum and mass conservation, can be written in the following form:

\[
\frac{\partial u_i}{\partial t} = \frac{\partial p}{\partial x_i} - \frac{\partial}{\partial x_j}(u_i u_j) + \nu \frac{\partial^2 u_i}{\partial x_j^2},
\]

\[
\frac{\partial u_i}{\partial x_i} = 0.
\]

All velocities, co-ordinates and time in the above equations have been made dimensionless by means of the wall shear velocity \( u_r \), by the half-width of the channel \( \delta \), and by the time scale \( \delta/u_r \) which on dimensional grounds is expected to be the time-scale of the large eddies in a channel flow. Unfortunately, the solutions for the equations (3.1) and (3.2), the true realizations of turbulent flows, are impossible for flows at a practical Reynolds number. Therefore, some approximation must be introduced. In LES, the approximation is made by averaging the primitive Navier-Stokes equations (3.1) and (3.2) over a small region in space. The space-averaging procedure involves decomposition of instantaneous flow variables into the grid scales and the subgrid scales. In the present study, the realization of the decomposition is via discretization used as an implicit filter, rather than an explicit filter, like the Gauss-
3.2 Subgrid-scale model

sian or the sharp Fourier cutoff filters. This implicit filter yields the space filtered Navier-Stokes equations for incompressible flows in the following non-dimensional form:

$$\frac{\partial \bar{u}_i}{\partial t} = -\frac{\partial \bar{p}}{\partial x_i} - \frac{\partial}{\partial x_j} (\bar{u}_i \bar{u}_j) + \frac{\partial \tau_{ij}}{\partial x_j} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_j^2}$$  \hspace{1cm} (3.3)

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0$$  \hspace{1cm} (3.4)

in which overbars represent the filtered flow field. The terms $\tau_{ij}$ arising from the averaging process are the SGS stresses requiring to be modelled. They take the following form,

$$\tau_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j$$  \hspace{1cm} (3.5)

which are unknown and must be modelled so that the equations (3.3) and (3.4) could be closed. Next the SGS modelling is discussed.

3.2 Subgrid-scale model

The need for SGS modelling has been pointed out in chapter 1. In addition, chapter 2 has made an exhaustive review of the main approaches to SGS modelling. Although chapter 2 has given several SGS models, only the Smagorinsky and dynamic models are actually used in the present study for the reasons presented there. This section will focus on the difficulties encountered by the standard versions of the Smagorinsky model and the Germano-Lilly dynamic model presented in chapter 2 if they are used in the simulations of low Reynolds number channel flow being performed in chapters 5, 6, 7 and 8. Finally, a modified dynamic SGS model for the low Reynolds number situation will be presented in detail.
3.2 Subgrid-scale model

3.2.1 Smagorinsky model

Among the various existing SGS models, the Smagorinsky model is the one developed earliest and used most widely. Also it is in a simple form,

$$\nu_t = (C_s\Delta)^2 \overline{S}$$  \hspace{1cm} (3.6)

with

$$\overline{S}^2 = 2\overline{S}_{ij}\overline{S}_{ij}$$  \hspace{1cm} (3.7)

where $C_s$ is a constant, $\Delta$ the mesh separation distance, and $\overline{S}$ the amplitude of the large-scale strain rate tensor (or velocity deformation). If $l$ denotes the turbulence length scale, it will take the following form,

$$l = C_s\Delta.$$  \hspace{1cm} (3.8)

Several problems related to this model have received extensive attention over the years of LES practice. The major deficiencies have been pointed out in section 2.2.2. For the wall bounded channel flow being simulated in the present study, the drawbacks of the Smagorinsky model are largely due to its erroneous prediction of the turbulence length scale. Clearly, the length scale becomes smaller than the filter size near the wall where the growth of turbulence structures is limited, and eventually should approach zero at the wall in the proper manner. However, even under these circumstances, the Smagorinsky model still relates it to the filter size, which never becomes zero (see equation (3.8)). Some form of modification, like a wall damping function, has been used to correct this inadequacy of the model, but it is difficult to justify this correction on theoretical grounds. In addition, the Smagorinsky assumption that energy production and energy dissipation are in balance fails in the near-wall region since viscous dissipation becomes important there. Despite the above drawbacks of the Smagorinsky model, many simulations using this model
are still successful to a large extent. The reason for the success may be explained. First, the Smagorinsky model is able to predict the global energy transfer adequately though the local energy transfer is incorrect. Second, in LES, the SGS model only accounts for a small fraction of the Reynolds stresses: consequently, relatively large errors in a subgrid-scale model do not contaminate the resolved field excessively. Third, the Smagorinsky energy balance assumption becomes more reasonable and the modification to the reduction of the subgrid length scale near the wall is not required as long as the wall layer is dealt with by a wall model, rather than resolved explicitly. In that case, the uncertainty about the reliability of the various existing wall damping functions will be removed. Accordingly, the Smagorinsky model is still widely used. Furthermore, some modern SGS models also depend on this model to a certain extent. For instance, the Smagorinsky model is used as a dissipative factor in the mixed model, and as a base model in the dynamic SGS model. In addition, the simple form of the Smagorinsky model is its advantage compared to other improved SGS models, since any modification will definitely lead to an increase of the complexity of SGS modelling and therefore the cost of simulations. Hence, for some fundamental studies, such as the investigation of the effect of grid refinement on the turbulence statistics being carried out in the present study, the Smagorinsky model with some theoretical treatment (to be discussed later in this chapter) is satisfactory and more convenient.

### 3.2.2 Dynamic subgrid-scale model

As presented in section 2.2.2, the dynamic SGS model seems to have overcome most of deficiencies of SGS models of eddy-viscosity type. But, several issues need to be discussed before it is actually used in LES:

1) The appropriate form of averaging to remove the numerical instability.
2) The choice of the “test” filter width.

3) The ability to account for energy backscatter.

First of all, when homogeneous directions exist in turbulent flows concerned, an average over the homogeneous direction or plane, where appropriate, is fairly effective to deal with the encountered numerical instabilities, as demonstrated by other researchers [20, 47] and by the present study. In other flows without any homogeneous direction, other averaging choices, such as time averaging, may be more appropriate. If some more severe situation occurs, such as the instability attributed to negative total viscosity (sum of the molecular and the eddy viscosity) sustained over many time steps at some position, the remedy for this can be achieved by artificially setting the total viscosity to zero at those locations.

The second issue arises from the fact that if the ratio of the “test” filter width to the grid filter width is too small, the dynamic model uses little information of scales of motion between the “test” scale and the grid scale. In that case, the model is not considered to be reliable, since the elements of $\mathbf{L}_{ij} = \mathbf{u}_i \mathbf{u}_j - \mathbf{u}_i \mathbf{u}_j$ involved in the dynamic model (see below) are closely associated with these scales of motion. On the other hand, if it is too large, important local information is averaged away since some type of average has to be used to compute the flow variables of “test” grid from the grid flow field. Germano found that the ratio $\hat{\Delta}/\Delta = 2$ is optimum for channel flows. His results also suggested that the simulations are not sensitive to the value of this ratio. But the difficulty will arise in simulations of turbulent flows at low Reynolds number. Cabot and Moin [7] found that low Reynolds number LES results for channel flow indicate a greater sensitivity to the choice of $\hat{\Delta}/\Delta$. In the present study a modified dynamic model is proposed and applied to a low Reynolds number channel flow with varying filter width ratios ($\hat{\Delta}/\Delta$) to test the new model’s performance.
3.2 Subgrid-scale model

As presented in section 2.2.2, the methodology described by Germano et al. [20] and the modification proposed by Lilly [35] give the dynamic model (Germano-Lilly dynamic model) coefficient in the form

\[ C(y,t) = \frac{\langle \tilde{L}_{kl} M_{kl} \rangle}{2 \langle M_{ij} M_{ij} \rangle} \]  \hspace{1cm} (3.9)

with

\[ \tilde{L}_{ij} = \bar{u}_i \bar{u}_j - \hat{u}_i \hat{u}_j = T_{ij} - \hat{r}_{ij} \]  \hspace{1cm} (3.10)

and

\[ M_{ij} = \Delta^2 \tilde{S}_{ij} - \Delta^2 \hat{S}_{ij} \]  \hspace{1cm} (3.11)

where overbars represent the grid-filtered field, and carets over the overbars the test-filtered field; \( \langle \cdot \rangle \) indicates that the quantities are averaged over the homogeneous plane parallel to the wall. In the above Germano-Lilly dynamic model, the numerator and denominator are averaged over the homogenous plane respectively. That means they have been assumed to be functions of \( y \) and \( t \) only before the model coefficient is computed. Thus the computed model coefficient through the averaged numerator divided by the averaged denominator is no longer on a local basis. Instead, the model coefficient is given in the present study as

\[ C(y,t) = \frac{1}{2} \left( \frac{\tilde{L}_{kl} M_{kl}}{M_{ij} M_{ij}} \right). \]  \hspace{1cm} (3.12)

The difference of this formula to the Germano-Lilly formula is that the model coefficient here is truly computed on a local basis. Then the dynamically calculated model coefficient, which is time-dependent and varies in space, is averaged over the homogenous plane parallel to the wall. The advantage of this modification is that more local information can be kept than the model (3.9). The local information is very important for the quality of the dynamic model. In addition, the physical meaning of the average procedure used in (3.12) is more obvious than that in (3.9). The
performance of this modified model in the simulations of the low Reynolds number channel flow will be tested and compared with that using the Germano-Lilly dynamic model later.

Thirdly, in the author's opinion, the ability of the SGS dynamic model to account for backscatter might be partially reduced by artificially setting the total viscosity to zero wherever this value becomes negative, thereby perhaps losing one conceptual advantage of the dynamic formulation, the ability of adding randomness to the explicit scales due to the upgrid energy transfer. So a better dynamic model should be capable of avoiding large negative value of eddy viscosity as much as possible to get rid of the need of artificially setting the total viscosity to zero. The low Reynolds number LES results for channel flow (to be presented later) using the modified dynamic model (3.12) was more numerically stable.

3.3 Near-wall subgrid-scale modelling

Another issue of great importance in the simulations of channel flow being performed in the present study is to ensure the proper behaviour of eddy viscosity when the wall is approached, i.e. the eddy viscosity and its derivatives must vanish in the proper manner at the walls. Two ways of achieving this are used here. One is to use an exponential damping function to reduce $\Delta$ as the wall is approached; another is to use a dynamic method to enable the eddy viscosity to adjust itself to flow conditions. In this section, both of the methods are presented.

3.3.1 Empirical models

From the formulation of the Smagorinsky eddy viscosity (3.6), it is easy to find that it gives a non-zero eddy viscosity at the wall. When the Smagorinsky model
is used, a modified length scale is therefore required to account for the reduction of SGS length scale in the near-wall region. This is achieved in the present study by multiplying by a Van Driest [15] exponential damping function which is in the following form

$$1 - \exp\left(-y^+/A^+\right)$$

(3.13)

where $A^+ = 25$ and $y^+ = y_{ur}/\nu$, the distance to the nearest wall in the wall units. Thus the turbulence length scale reads

$$l = C_s \left[1 - \exp\left(-y^+/A^+\right)\right] (\Delta_x \Delta_y \Delta_z)^{1/3}.$$  

(3.14)

There are also some other wall damping functions available, for example,

$$l = C_s \left[1 - \exp\left(-y^+/A^+\right)\right] \sqrt{\Delta_x^2 + \Delta_y^2 + \Delta_z^2}$$

(3.15)

$$\frac{1}{l} = \frac{1}{C_s (\Delta_x \Delta_y \Delta_z)^{1/3}} + \frac{1}{\kappa(1 - |y|)}$$

(3.16)

$$l = C_s \left[1 - \exp\left(-y^{+3}/A^{+3}\right)\right]^{1/2} (\Delta_x \Delta_y \Delta_z)^{1/3}.$$  

(3.17)

Among these various existing wall damping functions, it is believed that (3.17) is the best one due to the advantage of its ensuring the proper behaviour for the SGS Reynolds stress $\tau_{12}$ near the wall, but we would rather choose one in a simpler form, like (3.14), for the investigation of the effect of varying $C_s$ and resolution on predicted turbulence statistics. There are two reasons for doing so: first, the simulations of turbulent channel flows are not very sensitive to the precise form of the damping function [68, 48]; second, for the investigation just mentioned above, the accuracy of the wall model is not important as long as the same damping function is used for all the cases.
3.3.2 Dynamic method

Although the length scales near the walls are fairly amenable to theoretical treatment like damping functions, difficulties will arise for flows in which the form of the wall function is unknown or in which the concept of a wall function is inappropriate or inapplicable. Under these circumstances, the best way of dealing with the turbulence structures near the wall so far might be to use a dynamic SGS model which is capable of correctly predicting the behaviour of the subgrid eddy viscosity in the near-wall region, without any ad hoc wall damping functions. This achievement enables the LES method to be applied to complex flows without requiring any tuning to match the experimental data for each situation. The ability of this dynamic method to adjust itself from flow to flow to ensure the proper near-wall behaviour will allow LES to play a more positive role in turbulence simulations, and to be applied to even more complex turbulent flows.

3.4 Boundary conditions

For all meshes used in the simulations being performed in the present study, the boundary condition is a natural, no-slip condition. This is appropriate since the mesh point closest to the wall $y^+_1$ is less than 5 wall units even on the coarsest mesh, where $y^+_1 = 0.635$ which is well inside the linear sublayer.

3.5 Modification of eddy viscosity at low Reynolds number

The simulated flow in the present study is a channel flow at low Reynolds number. When the resolution becomes fine enough, the effect of SGS eddy viscosity $\nu_t$ will be small and be comparable to molecular viscosity $\nu$, unlike true LES, where $\nu$ can be negligible compared to $\nu_t$. Accordingly, an additional modification has to
be made. Several hypotheses for energy transfer which are used to deduce energy spectra in the dissipation range have been discussed in section 2.4. The method used in the present study is to apply a modified $\nu_t$ to the simulations of the low Reynolds number, which was derived by using a curve fitting technique [68] (details have been given in section 2.4),

$$\nu_t = \nu_s - \beta \nu \left[ 1 - \exp \left( -\nu_s / (\beta \nu) \right) \right]$$  

(3.18)

where $\nu_s$ is the Smagorinsky eddy viscosity calculated in the normal way. This modification to the eddy viscosity, based on the assumption that the cutoff grid falls in the dissipation range, will be tested later for a low Reynolds number channel flow.

### 3.6 The finite difference equations

With a SGS model, the space filtered Navier-Stokes equations (3.3) and (3.4) are closed. In principle, all that needs to be done is to solve the equations. Unfortunately, no analytic solution for the governing equations of large scale motions (3.3) and (3.4) exists. Therefore, they are generally expressed in differential form to seek the numerical solution. The control volume approach is employed here to derive the finite difference equations. It involves dividing the computational domain into a great number of finite volumes called "cells". In the present study, a staggered grid mesh is applied to the different independent variables. For such a grid mesh, grid points at which $\bar{v}$ is stored are in the centres of each cell surface parallel to the boundaries. Grid points for $\bar{p}$ are located vertically midway between grid points for $\bar{v}$; grid points for $\bar{u}$ are located horizontally in $x$ midway between grid points for $\bar{p}$; grid points for $\bar{w}$ are located laterally midway in $z$ between grid points for $\bar{p}$. An advantage of this arrangement is that the finite difference form of the continuity equation is expressible compactly at grid points for $\bar{p}$, which is very convenient when it comes to solving the Poisson equation for $\bar{p}$. 
For incompressible flows, the governing equations of larger scale motions (3.3) and (3.4) are discretized by central differences in the following differential form

\[ \nabla_t \tilde{u}_i = -\nabla_i \tilde{p} - \nabla_j (\tilde{u}_i \tilde{u}_j) + \nabla_j \tau_{ij} + \nu \nabla^2 \tilde{u}_i \]  
(3.19)

\[ \nabla_i \tilde{u}_i = 0. \]  
(3.20)

where \( \nabla_t \) is the central time differencing operator, \( \nabla_i \) the central differencing operator in the \( x_i \) direction, and \( \nabla^2 \) the discrete Laplacian operator.

### 3.7 Finite difference implementation

The finite volume multiple mesh version of the code ECCLES, originally written by Gavrilakis [19] and others, then modified to suit the present study, was used here. The code is capable of simulating three-dimensional time-dependent turbulent channel flows having two statistically homogeneous dimensions. The third dimension can involve strong shear.

The code integrates the resulting equation (3.19) forward in time to second order in the time step using the Adams-Bashforth (A-B) algorithm,

\[ \tilde{u}_i(t + \Delta t) = \tilde{u}_i^*(t) - \frac{3}{2} \nabla_i \tilde{p}(t) \Delta t \]  
(3.21)

where

\[ \tilde{u}_i^* = \tilde{u}_i(t) + \left[ \frac{3}{2} H_i(t) - \frac{1}{2} H_i(t - \Delta t) \right] \Delta t + \frac{1}{2} \nabla_i \tilde{p}(t - \Delta t) \Delta t \]  
(3.22)

and

\[ H_i = \nabla_j [\tilde{u}_i \tilde{u}_j + \tau_{ij} + \nu \nabla_j \tilde{u}_i]. \]  
(3.23)

The * superscript indicates provisional velocity components. It can be seen that all terms in the momentum equation are treated explicitly, apart from the pressure term, which is unknown at the current step. The pressure will be found by solving...
3.7 Finite difference implementation

a Poisson equation derived through applying the divergence operator to (3.21) and enforcing continuity (3.20) on \( \bar{u}(t + \Delta t) \), the velocity at the next time step. The deduced Poisson equation takes the following form

\[
\nabla^2 \bar{p}(t) = \frac{2}{3} \nabla_i \bar{u}_i^*(t).
\] (3.24)

In the code, this equation is solved at each step by Fourier transforming it in the two directions, which are assumed periodic (for the present study, they are the \( x \) and \( z \) coordinates).

In summary, the numerical method actually employed here is a standard pressure-correction algorithm. It is started by solving (3.22) for the provisional velocity field at time \( n\Delta t \). Then the pressure at time \( n\Delta t \) is calculated by solving the Poisson equation (3.24). Finally the velocity is advanced in time to the time \((n + 1)\Delta t\) by appending the pressure gradient term to the provisional velocity. Note that continuity (3.20) is not computed explicitly, but is satisfied at the time \((n + 1)\Delta t\) since the derivation of the Poisson equation entails the enforcement of the continuity condition on \( \bar{u}_i(t + \Delta t) \). In addition, the fast direct Poisson solver by Fourier methods employed here forces periodic boundary conditions on the solutions in the two directions Fourier transformed. This requirement can be easily satisfied for a channel flow, as in the present study, provided that the computational box dimensions exceed the spatial correlation lengths of the turbulence.

In the code, there is also another numerical scheme called Crank-Nicolson (C-N) for solving the pressure. It gives the velocity at the next time step a form

\[
\bar{u}_i(t + \Delta t) = \bar{u}_i^*(t) - \frac{1}{2} \nabla_i \bar{p}(t)\Delta t
\] (3.25)

and then the Poisson equation, derived by applying the divergence operator to the above equation and enforcing continuity on \( \bar{u}_i(t + \Delta t) \), reads

\[
\nabla^2 \bar{p}(t) = 2 \nabla_i \bar{u}_i^*(t).
\] (3.26)
However, the above equation for pressure has been proved not suitable for the channel flow being simulated here by two test simulations on a very coarse mesh with grid points $8 \times 4 \times 6$. For comparison, the C-N algorithm was used for the pressure term in one simulation, and the A-B algorithm for the pressure term in the other simulation. For the rest of the terms, the A-B algorithm was used in both of the test runs.

In table 3.1, the pressure field at a monitored grid point and its volume-average values are presented. The volume-average values of the velocity fluctuations are given in table 3.2. From the performances of the A-B and the C-N algorithms demonstrated in the tables 3.1 and 3.2, it was found that the pressure resolved using the C-N algorithm shows persistent oscillations at the monitored grid point and that using the A-B algorithm enables the initial error arising from the unreal initial pressure field, which is set to be zero at the first time step, to decrease quickly during the integration of the Navier-Stokes equations forward in time. Even though these pressure oscillations do not contaminate the velocity field (see table 3.2) due to the pressure gradients being very small compared to the other terms of the equations, the pressure field is unstable. For this reason, the A-B algorithm was used for the pressure term, like other terms, to obtain the turbulent statistics in the present study.

### 3.8 Problem geometry and computational grid

The flow geometry and naming conventions are shown in figure 3.1. The box dimensions are $4\pi \delta \times 2\pi \delta \times 2\delta$ in the streamwise $x$, spanwise $z$, and cross-stream $y$ directions respectively, where $2\delta$ is the distance between the walls. Since a fully developed turbulent channel flow, like the channel flow being simulated in the present study, is homogeneous in the streamwise $x$ and spanwise $z$ directions,
Table 3.1: The Pressure

<table>
<thead>
<tr>
<th>step</th>
<th>Crank-Nicolson</th>
<th>Adams-Bashforth</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$p^2$</td>
<td>$p(4, 2, 3)$</td>
</tr>
<tr>
<td>1</td>
<td>2.074E+7</td>
<td>0.977E+3</td>
</tr>
<tr>
<td>2</td>
<td>2.076E+7</td>
<td>-0.971E+3</td>
</tr>
<tr>
<td>3</td>
<td>2.074E+7</td>
<td>0.977E+3</td>
</tr>
<tr>
<td>4</td>
<td>2.079E+7</td>
<td>-0.971E+3</td>
</tr>
<tr>
<td>5</td>
<td>2.074E+7</td>
<td>0.977E+3</td>
</tr>
<tr>
<td>6</td>
<td>2.076E+7</td>
<td>-0.971E+3</td>
</tr>
</tbody>
</table>

Table 3.2: The Velocity

<table>
<thead>
<tr>
<th>step</th>
<th>Crank-Nicolson</th>
<th>Adams-Bashforth</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$u^2$</td>
<td>$v^2$</td>
</tr>
<tr>
<td>1</td>
<td>6.8709</td>
<td>0.9621</td>
</tr>
<tr>
<td>2</td>
<td>6.8456</td>
<td>0.9613</td>
</tr>
<tr>
<td>3</td>
<td>6.8237</td>
<td>0.9602</td>
</tr>
<tr>
<td>4</td>
<td>6.8024</td>
<td>0.9592</td>
</tr>
<tr>
<td>5</td>
<td>6.7816</td>
<td>0.9581</td>
</tr>
<tr>
<td>6</td>
<td>6.7612</td>
<td>0.9571</td>
</tr>
</tbody>
</table>
periodic boundary conditions were used in both of the directions. The reason for choosing such a channel flow as the problem geometry being studied here is that the shortcomings of the standard Smagorinsky model in the near-wall region, which have been pointed out in section 2.2.2, have shown clearly even though channel flow has a very simple geometry. Of particular importance is that these near-wall problems of the standard Smagorinsky model appear to be amenable to theoretical treatment for a channel flow so that the results of the simulation using a modified Smagorinsky model can be used as a comparison to test new SGS models developed to remove the shortcomings of the standard Smagorinsky model. In addition, there is the database of the full simulation of channel flow available for comparison. Furthermore, choosing a channel flow enables the simulation results to decouple the effect of complex configurations when a new SGS model is tested. In the present study, we shall take all these advantages of channel flows to test performance of the modified dynamic SGS model (3.12) and a new technique for LES, multiple mesh method being presented in the next chapter.

The selection of the computational grid is generally guided by the requirement for resolving the important scales of motion in the turbulent flow and the availability of computer resources. Normally, some compromise between these two factors should be worked out. In the present study, the mesh was stretched in the direction perpendicular to the solid wall boundaries since the flow is anisotropic in this direction, and thus weighted averaging was used to ensure that the nonlinear term (treated by second-order Adams-Bashforth method) conserves kinetic energy as well as momentum. To maintain second-order accuracy the stretching in the \( y \) direction was modest, having the following form [68]

\[
y_k = \frac{\tanh(Q\zeta_k)}{\tanh(Q)}
\]  

(3.27)

where \( \zeta_k \) was the cross-stream location of grid points uniformly distributed between -1 and +1, and \( Q \) is an adjustable parameter; a large value of \( Q \) distributes more
points near the walls. In the present study $Q$ had the value 2.35. For the grid distribution in the y-direction $N_y = 64$ and $N_y = 128$ in the simulations being performed, this value of $Q$ respectively gives the mesh point closest to the wall a co-ordinate $y_1^+ = 0.638$ and $y_1^+ = 0.32$ which are well inside the viscous sublayer ($y^+ < 5$). This indicates that both of the above y-direction resolutions are sufficient to resolve the viscous sublayer. In the streamwise and spanwise directions in which the flow is homogenous, the downstream length and the lateral width were subdivided uniformly. The grid distribution in these two directions depends on the requirements of various simulations being performed in the present study. The computational grids for the different calculations will be given where the simulation is discussed.
Figure 3.1: The Flow Geometry


4 Multiple Mesh Simulation

In this chapter, a brief introduction to the concept of the multiple mesh simulation will be given, followed by a detailed description of its realization in a real LES.

4.1 Introduction

An exhaustive discussion on turbulence modelling has been given in the above chapters. Thus it can be seen that the better the subgrid-scale modelling is, the more efficient the LES technique becomes. As the understanding of turbulence mechanism on the basis of a large body of experimental and computational results has been increasing over the years, there seems a better chance of constructing a superior SGS model than ever before. If this occurs, the very large eddy simulation (VLES) technique, which models even more scales than LES, may be capable of tackling industrially important flows. Besides the acknowledgement of the importance of the quality of the SGS model, we are now also aware of a further issue on the potential of the LES application in most engineering flows. For the majority of flows of engineering interest, LES is likely to remain too expensive for day-to-day use in the near future. Therefore, to enhance the efficiency of LES, a great effort toward improving LES economy should be made parallel to that of developing SGS modelling. The way of achieving this goal is to reduce the cost of LES in terms of computer time to make LES be more easily handled by the present computers.

In LES, long-term integration periods sufficient to reach a statistically stationary state are required to gather reliable turbulent statistics independent of the initial conditions. Simulation times of at least several $T$ ($T$ is the correlation period of large structures) are usual. Accordingly, it appears not efficient to start a simulation from
4.2 Description of multiple mesh method

Multiple mesh simulation is a method to accelerate LES, which was initiated by Voke [64, 65]. His multimesh simulation was designed to work on a set of two or more nested meshes (referred to as multimesh levels), on each of which the simulation time spent was determined by an equal-cost strategy [64] requiring that the same amount of CPU time be expended on each of the nested meshes. Since Voke found [68] that the equal-cost strategy enables a two-level multimesh simulation with the typical resolutions of his test run to move the coarse mesh fields forward to a new realisation in a period of the computation time which is roughly the same as the computation time required to regenerate the mesh-scale eddies on the fine mesh, his multimesh simulation was actually implemented on a set of two nested meshes, placing a coarse mesh over a fine mesh such that each block of eight cells is within a coarse mesh cell. Their relation and the definitions of the velocity components on the two meshes are illustrated in figure 4.1. These definitions of the velocities are carefully chosen to ensure continuity on both meshes [68] (details will be given later). Voke started his multimesh simulation on the fine mesh, then injected the velocities to the coarse mesh after a fraction of T (the correlation period of the large structures). At the moment of the injection, the residual velocity field, the difference between the velocity field on the fine and coarse meshes, was computed and stored. Note the resulting residual velocities are defined on every grid point of
4.2 Description of multiple mesh method

the fine mesh. On the coarse mesh, the simulation was carried out for a period of order \( T \), during which, unlike the coarse mesh part of the simulation, the residual velocity field did not proceed. By the end of this period, the velocity field was then interpolated onto the fine mesh, where the frozen residual field stored was added to it. For the typical multimesh simulation performed using an equal-cost strategy [68], the ratio of the computation time spent on the fine mesh to that spent on the coarse mesh is \( \frac{1}{16} \) (to be explained later). The whole process described above is referred to as a multimesh cycle [68]. The cycles were repeated until adequate statistics had been gathered. Gathering statistical samples was done only once on the fine mesh in each cycle. Owing to the definition of the velocities on the nested meshes, and the injection and interpolation methods (see below), both injection process generating the coarse mesh velocity field and the interpolation process creating the fine mesh velocity field preserve continuity; so does the residual velocity field. In the multimesh simulation [68], the injection into the coarse mesh is given by

\[
U_1(\Delta y_{ab} + \Delta y_{cd}) = \frac{1}{2}(u_a + u_b)\Delta y_{ab} + \frac{1}{2}(u_c + u_d)\Delta y_{cd}.
\] (4.1)

All the symbols in the above formula are shown in figure 4.1, in which the coordinate convention has been adapted according to the present study, i.e. \( x \) is the streamwise direction, \( y \) the cross-stream direction and \( z \) the spanwise direction. Here, \( U_1 \) is defined as the surface average velocity through one whole face perpendicular to the \( x \) axis, and \( u_a, u_b, u_c \) and \( u_d \) are the individual parts of \( U_1 \). A similar weighted average is used for \( W_1 \), while for \( V_1 \) the flux is simply the mean of the four \( u \) fluxes on the fine mesh since both of the downstream length and the lateral width are divided uniformly. According to the definition of the residual velocities given earlier, the differences between \( U_1 \) and \( u_a, u_b, u_c \) and \( u_d \) are the residual velocities on the surfaces of the coarse mesh cells, while on the interleaving planes linearly interpolated coarse mesh velocities are subtracted from the fine mesh velocities to
obtain residual velocities there. So the residual velocities are
\[
\Delta u_i^n = u_i^n - U_i^n \quad \text{surfaces} \tag{4.2}
\]
\[
\Delta u_j^n = u_j^n - \frac{1}{2}(U_1^n + U_2^n) \quad \text{interleaving planes}
\]

where \(i = a, b, c, d\) and \(j = e, f, g, h\). The \(n\) superscript indicates the moment of the injection. On the other hand, at the moment of interpolation back to the fine mesh, precisely the same form of interpolation is used to define coarse mesh velocity fields on the interleaving planes as was used to compute the residual velocity fields during injection, and the residual velocity field is added to the interpolated quantities on the interleaving planes as well as to the coarse mesh velocities on the surfaces of the coarse mesh cells. So the velocities on the fine mesh after interpolation are given as
\[
u_i^{n+1} = U_i^{n+1} + \Delta u_i^n \quad \text{surfaces} \tag{4.3}
\]
\[
\nu_j^{n+1} = \frac{1}{2}(U_1^{n+1} + U_2^{n+1}) + \Delta u_j^n \quad \text{interleaving planes}
\]

where \(i = a, b, c, d\) and \(j = e, f, g, h\). The \(n + 1\) superscript indicates the moment of the interpolation. The above formula clearly shows the residual velocity field does not advance during the simulation on the coarse mesh. Also from the above equations (4.1), (4.2) and (4.3), it is easy to show that the velocities on the nested meshes and the residual velocities obey continuity.

As described earlier, the fine mesh in Voce's multimesh code has twice the number of grid points of the coarse mesh, in each of the three directions. This nested-mesh collocation will result in eight times more grid points, and a tighter CFL limit and thereby a halved time step on the fine mesh, which in turn results in a simulation on the fine mesh sixteen times more costly in terms of computer time. So the multimesh simulation strategy that the simulation spends most of computation time on the coarse mesh to advance the flow field to a new distinct realisation of the flow, and spends a fraction of period expended on the coarse mesh to produce the fine mesh-grid eddies is very attractive. In addition, retaining the residual velocity field on the fine mesh at the moment of interpolation of the
4.2 Description of multiple mesh method

velocities of the coarse mesh back to the fine mesh is another important property of this multimesh method. Although the residual velocity field remains frozen while the coarse mesh part of the simulation proceeds, it still carries some information of the important fine scales of eddy motions. Therefore, the small scale field will not be entirely destroyed due to adding this frozen residual velocity field to the fine mesh velocity field even though the fine mesh and coarse mesh velocity fields are not considered to be realistically correlated. A two-level simulation performed by Voke has shown that the cost ratio between the multimesh simulation actually preformed and equivalent simulations performed wholly on the fine mesh is about 7.5. However, the unrealistic interaction between fine mesh scale structures and the flows simulated on the coarse mesh resulting from adding frozen residual velocity indicates that there is still room for improvement in the multimesh method.

As Voke has pointed out, if a more sophisticated method of interpolation is designed to be capable of recreating physically realistic fine mesh scale structures to interact properly with the flow simulated on the coarse mesh, the multimesh method would be even more promising. A straightforward way of achieving this could be by deducing the fine-mesh velocity directly from the coarse-mesh velocity by using the physical relationships between them (the algebraic relation between the resolved turbulent stresses and the SGS stresses at both the levels, for example). Doing so would provide a physically realistic field at the fine level, which in turn would lead to a shorter phase at this level. Thus we would expect the economy of numerical simulations of turbulence to be improved further.

Unfortunately, this problem cannot be solved by the multimesh code [68] that replaces each block of eight fine mesh cells by a combined coarse mesh volume (see figure 4.1). In that case, each (coarse) mesh volume carries three (coarse) velocity components, which form three constraining relationships with the fine mesh velocities by the injection method described earlier, but only two of them are independent
4.2 Description of multiple mesh method

if we consider the eight continuity conditions on the eight fine mesh subcells to be independent because among three constraining relationships and the eight continuity conditions, one is dependent on the other ten (which can be easily shown by adding all the eight continuity conditions up). In addition, there are five independent Reynolds stress algebraic relations among the six between the fine and coarse meshes due to $\overline{S}_{ii} = 0$ for an incompressible flow. Now we end up with 7 equations to solve for the fine-mesh velocity. On the other hand, the eight fine mesh subcells within each coarse mesh volume carry 24 velocity components, 16 of which are independent due to the continuity conditions in each subcell of the eight. Thus there are nine independent degrees of freedom in the fine mesh field remaining undetermined. Under these circumstances, recreating the velocity field on the fine mesh by retaining the frozen residual velocity field seems a good compromise.

In the present study, a modified multimesh scheme with the same motivation as that of the multimesh method described above, is devised to overcome the deficiency of using the frozen residual velocity during the process of recreating the fine mesh scale structures. As presented above, this deficiency results from the nested-mesh collocation replacing each block of eight fine mesh cells by a combined finite volume on the coarse mesh, which gives more degrees of freedom in the fine mesh field to be determined than can be determined. To reduce the number of the undetermined degrees of freedom to zero, the only way is to reduce the number of fine mesh subcells within each coarse mesh volume, which definitely results in the reduction of the number of degrees of freedom in the fine mesh field. If the number of grid points of the coarse mesh is doubled only in the streamwise $x$ and spanwise $z$ directions, as illustrated in figure 4.2, each coarse mesh volume will contain four fine mesh subcells. For this nested-mesh collocation, there will be eight independent velocity components on these four fine mesh subcells. Since each (coarse) mesh volume carries three (coarse) velocity components, which form two independent constraining
relationships with the fine mesh velocities, and there are still 5 independent Reynolds stress algebraic relations between the fine and coarse meshes, we end up with one more degree of freedom in the fine mesh field to be determined than there are equations of velocities on the fine mesh. The same problem as that encountered by Voke's multimesh method arises even if the number of undetermined degrees of freedom in the fine mesh field has been decreased.

The way to circumvent this impasse is to reduce the number of the fine mesh subcells within each coarse mesh volume further. For doing so, the number of grid points of the coarse mesh is doubled only in one direction, as illustrated in figure 4.3, which results in a coarse mesh volume containing two fine mesh subcells. For this nested-mesh collocation, two fine mesh subcells within each coarse mesh volume carry six velocity components. On the other hand, each coarse mesh volume carries three velocity components, which form three constraining relationships with the fine mesh velocities. As has already been pointed out, among the two continuity conditions on the two fine mesh subcells and three constraining relationships, only four of them are independent. There are still five Reynolds stress algebraic relations between the fine and coarse meshes, and we can choose six from the above nine independent equations to obtain the fine mesh velocity field. The multimesh method using this nested-mesh collocation is referred to as a modified multimesh method. Now it appears that the undetermined problem has been overcome.

Besides the different nested-mesh collocation, the implementation of the modified multimesh simulation is also different to that of the multimesh simulation of Voke. In the present study, the modified multimesh scheme is devised to start LES initially from a reasonably coarse mesh and then interpolate its velocities onto a finer mesh after spending a period of simulation time just sufficient to generate the mesh-scale eddies on the former. The refinement could be continued until an adequate numerical resolution is achieved. The most outstanding feature of the
modified multimesh method is that the interpolation procedure is to be carefully
designed to create a physically realistic velocity field on the finer mesh by making
judicious use of the nested grids, otherwise the economic advantage of the multi-
mesh method would be offset by the lack of the correlation between the fine and the
coarse meshes. Fortunately the guideline has been given above. The following will
present its implementation.

4.3 Multiple mesh implementation

The multimesh scheme actually used in the test simulations being performed
in chapter 9 works on a series of nested meshes. To solve the fine-mesh velocity
from the coarse-mesh flow field, the nested meshes are designed in such a way that
the coarse mesh points are half as many as those of the fine mesh in just one of the
three directions, x (streamwise) or y (cross-stream) or z (spanwise), because, as has
been pointed out earlier, doing so will result in enough equations for determining
the new degrees of freedom on the fine mesh. Clearly, there is no constraint on the
sequence of refining the mesh. The sequence chosen in the present study will be x, 
z, and y, i.e. first the resolution in the streamwise direction x is doubled, next that
in the spanwise direction z, and then that in the cross-stream direction y. Hereafter
this process is referred to as a cycle whose meaning is different to that defined by
Voke, the three refinings in x, z and y directions being called phase 1, 2 and 3
respectively. The two meshes in a phase are referred to as the coarse and the fine
meshes respectively.

As illustrated in figure 4.3, the multimesh implementation in the present code
is to halve each finite volume cell of the coarse mesh to generate the next fine mesh
level (superscripts denote phase number, and subscripts, subcell number). The cells
shown in figure 4.3 are mass conservation cells, with the velocities indicated being
surface-averaged mass flux vectors. Define, for example in phase 1, the surface average velocity through whole faces perpendicular to the y and z axes as $V$ and $W$ respectively, and the individual parts of these as $v_1$ and $w_1$ respectively ($i = 1, 2$). However, the surface average velocity through the whole face perpendicular to the x axis $U$ is equal to $u_1$ because no individual part exists. The relationships of the velocity components between the coarse and the fine meshes in the three phases are constructed by weighted averaging.

In phase 1,

$$
U = u_1 \\
V(\Delta x_1 + \Delta x_2) = v_1 \Delta x_1 + v_2 \Delta x_2 \\
W(\Delta x_1 + \Delta x_2) = w_1 \Delta x_1 + w_2 \Delta x_2 ;
$$

(4.4)

in phase 2,

$$
u_1(\Delta z_1 + \Delta z_2) = u_1^2 \Delta z_1 + u_2^2 \Delta z_2 \\
v_1(\Delta z_1 + \Delta z_2) = v_1^2 \Delta z_1 + v_2^2 \Delta z_2 \\
w_1^1 = w_1^2 ;
$$

(4.5)

in phase 3,

$$
u_2^3(\Delta y_1 + \Delta y_2) = u_1^3 \Delta y_1 + u_2^3 \Delta y_2 \\
v_1^3 = v_1^3 \\
w_1^3(\Delta y_1 + \Delta y_2) = w_1^3 \Delta y_1 + w_2^3 \Delta y_2 .
$$

(4.6)

As has been mentioned earlier, among these three relationships of the velocities on the nested meshes and two continuity conditions exerting on the two fine mesh subcells within each coarse mesh volume, only four of them are independent. Here we choose all three relationships of velocities on the nested meshes and one continuity condition. On the other hand, the two fine subcells within each coarse mesh volume
4.3 Multiple mesh implementation

carry six velocity components. Two of them remain undetermined due to the three
constraining relationships and enforcement of one independent continuity condition
on one of the two fine mesh subcells. Thus there are two independent degrees
of freedom in the fine mesh field. Fortunately, five independent Reynolds stress
algebraic relations between the fine and coarse meshes are available, i.e.

\[ T_{ij} - \hat{\tau}_{ij} = \overline{u_i u_j} - \hat{u}_i \hat{u}_j, \tag{4.7} \]

where overbars represent the fine mesh, and carets over the overbars the coarse
mesh field. The \( T_{ij} \) are SGS stresses at the coarse mesh level, and \( \tau_{ij} \) SGS stresses
at the fine mesh level. The algebraic relations (4.7) are closely related to (2.66) used
by Germano to derive his dynamic SGS model, in which the SGS stresses of grid
and “test” scales are involved rather than those of fine and coarse scales involved
here. After being averaged over several neighbouring grid points, \( \tau_{ij} \) becomes \( \hat{\tau}_{ij} \).
Note that equation (4.7) represents five independent equations because \( \mathcal{S}_{ii} = 0 \) for
an incompressible flow. Among them only three Reynolds shear stress algebraic
relations are chosen because we cannot obtain the SGS turbulence intensities from a
LES due to the removal of the trace terms of SGS stresses \( u'_i u'_j \) with \( i = j \). Among
these three Reynolds shear stress algebraic relations, we have to choose two of them
to determine the two independent degrees of freedom in the fine mesh field which
remain undetermined at this stage. In order to avoid solving nonlinear equations
(the reason for doing so will be discussed later), further issues should be taken into
consideration. First, some approximation should be made due to the existence of
\( \hat{\tau}_{ij} \) in the equations (4.7), which contain the nonlinear terms of the unknowns, the
velocities of the fine mesh. If the Smagorinsky model is used, \( \tau_{ij} \) takes the following
form,

\[ \tau_{ij} = (C_s \Delta)^2 \mathcal{S} S_{ij} \tag{4.8} \]

where \( \mathcal{S}_{ij} = \frac{1}{2}(\partial \bar{u}_i / \partial x_j + \partial \bar{u}_j / \partial x_i) \) and \( \mathcal{S}^2 = 2 S_{ij} S_{ij} \). It can be seen that \( \tau_{ij} \) contains
complex second-order terms of the velocities on the fine mesh. The problem is solved
4.3 Multiple mesh implementation

Here by omitting $\hat{\tau}_{ij}$ from the equation (4.7), which therefore becomes

$$T_{ij} = \hat{u}_i \hat{u}_j - \hat{u}_i \hat{u}_j. \quad (4.9)$$

We should not expect that this approximation causes large errors in the solutions of the fine mesh field because SGS eddies carry only a fraction of total turbulent kinetic energy, especially for a fine mesh. Thus the omission should not contaminate the resulting velocity field of the fine mesh. Second, in phase 1 the algebraic relations of Reynolds stress components 12 ($T_{12} = \overline{u' v'} - \hat{u} \hat{v}$) and 13 ($T_{13} = \overline{u' w} - \hat{u} \hat{w}$) are chosen. They are linear equations due to the fact that velocity components $u$, involved in $T_{12} = \overline{u' v'} - \hat{u} \hat{v}$ and $T_{13} = \overline{u' w} - \hat{u} \hat{w}$, are known values because they are the velocity components only on the surfaces of the coarse mesh cells (see equation (4.4)). By contrast, the algebraic relation of Reynolds stress component 23 ($T_{23} = \overline{v' w} - \hat{v} \hat{w}$) contains the product of $v$ and $w$, which are unknown, and therefore result in a second-order equation. For the same reason, in phase 2 the algebraic relations of Reynolds stress components 12 and 23 are chosen, and in phase 3 the algebraic relations of Reynolds stress components 13 and 23 are chosen.

The formulations of the multimesh scheme are now derived for phase 1 only because the methods for all of the three phases are similar. For SGS stress components 12 and 13, (4.9) being applied to all the coarse mesh grid points yields,

$$T_{12}^i = \overline{(u' v')}^i - \hat{u}^i \hat{v}^i$$  \hspace{1cm} (4.10)

$$T_{13}^i = \overline{(u' w)}^i - \hat{u}^i \hat{w}^i$$

$$T_{12}^j = \overline{(u' v')}^j - \hat{u}^j \hat{v}^j$$  \hspace{1cm} (4.11)

$$T_{13}^j = \overline{(u' w)}^j - \hat{u}^j \hat{w}^j$$

$$T_{12}^k = \overline{(u' v')}^k - \hat{u}^k \hat{v}^k$$  \hspace{1cm} (4.12)

$$T_{13}^k = \overline{(u' w)}^k - \hat{u}^k \hat{w}^k$$

where $i = 1, 2, \ldots, N_1$, $j = 1, 2, \ldots, N_2$ and $k = 1, 2, \ldots, N_3$, with $N_1$, $N_2$ and $N_3$ being the resolutions in the $x$, $y$ and $z$ directions respectively at the coarse mesh
level. As has been addressed earlier, only the velocity components $u$ on the surfaces of the coarse mesh cells are involved in (4.10), (4.11) and (4.12), which are known. Thus the equations (4.10), (4.11) and (4.12) together with (4.4), form a linear set of equations, from which $v$ and $w$ can be found. Finally the continuity condition is used to obtain $u$ on the interleaving planes of the combined finite volume cells of the coarse mesh. Unlike in Yoke [68, 64, 65], in the present study continuity in the fine mesh velocity field is satisfied in an explicit way, i.e. enforcing continuity on the fine mesh velocity field to solve $u$ on the interleaving planes of the combined finite coarse volume. But now, a physically realistic velocity field can be created at the moment of a simulation moving from the coarse mesh to the fine mesh.

We now explain the reason for avoiding a nonlinear set of equations of the velocities on the fine mesh. For a nonlinear set of equations, especially for a multi-variable problem here, no analytic solution exists. Therefore, the equations have to be solved numerically. In general, solution finding for a nonlinear set of equations invariably proceeds by iteration. Under this circumstance, several difficulties arise if we do not avoid solving a nonlinear set of equations to obtain fine mesh velocities.

Firstly, the expressions of Reynolds shear stresses, such as $\overline{uv}$, $\overline{uw}$ and $\overline{vw}$, are expressed in terms of the fine mesh velocities at the neighbours of the coarse mesh points at which the coarse mesh Reynolds shear stresses $T_{ij}$ and the averaged fine mesh Reynolds stresses $\overline{T_{ij}}$ over several neighbouring points are defined. These neighbours would, in turn, couple to their neighbours, and so on. Ultimately, the equations of the fine mesh velocities would involve the velocities of the fine mesh at all grid points in the flow domain. Therefore, all the equations are interlinked through the velocities of the fine mesh and should be satisfied simultaneously during the iterated solution finding process. Since each finite volume cell on the coarse mesh has six equations (two Reynolds stress algebraic relations between the fine and coarse meshes, three constraining relationships of the velocities between the fine and
coarse meshes and one continuity condition) to solve six velocity components on the two subcells within it, for the typical resolution currently in use \((48 \times 64 \times 40)\), the nonlinear set of equations contains \(6 \times 48 \times 64 \times 40\) equations to be solved. During a iterated solution process, roundoff errors are inevitable. So, for such a vast number of equations, which have to be satisfied simultaneously, a long iteration process is required. If it occurs, accumulated roundoff errors in the solution process swamp the true solution since the velocity components in the spanwise \(z\) and cross-stream \(y\) directions are small. In addition, owing to the coupled velocities, the errors will spread over the whole flow domain. Furthermore, for a multi-variable problem, like the one encountered here, solving a nonlinear set of equations will entail evaluating the Jacobian matrix \(J\) [51], whose dimension is the square of the number of equations \((6 \times 48 \times 64 \times 40)\). It is clear that such a large matrix is not easy to handle. Secondly, for an iteration scheme, of extreme importance is the rate of convergence, which depends on the initial guess for the solution and the quality of algorithms. Unfortunately, we only have certainty that good algorithms will always converge, provided that the equations to be solved vary smoothly and the initial guess is good enough [51]. For the problem here this is not the case. If the rate of convergence is slow, then we lose the advantage of the multimesh method to improve the economy of LES.

### 4.4 Summary

The modified multimesh scheme in the present study is designed to reduce the computational requirements by the strategy that a simulation is carried out on a mesh whose mesh-grid is comparable to the eddy scale being realized by the simulation, and then moves to a finer mesh to advance a new realization of flow corresponding the finer mesh-grid. The performance of this method will be tested in
chapter 9. If this method enables a physically realistic velocity field to be created on the finer mesh, which will result in saving of simulation time, the modified multimesh method will hold promise of enhancing the efficiency of LES.

In brief, the advantages of such a modified multiple mesh scheme are threefold. Firstly, the multiple mesh strategy is based on the fact that in the early stage of a simulation, it is unnecessary to spend computer time on a mesh with a much smaller grid scale than the characteristic length scale of the turbulent structures being produced. For this reason, multiple mesh simulation will be carried out on a mesh until the simulation has generated eddies whose scale is comparable to this current mesh scale, then moves to a finer mesh where the simulation progresses further to realize new eddies corresponding to the new finer mesh, and so on. Clearly this strategy will enable the multiple mesh simulation to save computer time compared with a simulation performed wholly on the finest mesh on which the multiple mesh simulation also gathers its statistical samples, because doubling the resolution of each dimension will result in a simulation two times more costly. In addition, a tighter CFL limit imposed on one of the mesh dimensions, caused by the doubled resolution, will make the simulation even more expensive.

Secondly, since the modified multiple mesh method is capable of creating a physically realistic velocity field on the fine mesh at the moment of the simulation moving from the coarse mesh to the fine mesh, these realistic fine mesh structures will interact properly with the flow simulated on the coarse mesh. Therefore, changing mesh in a simulation by using this method does not damage the correlation of the fine mesh with the coarse one very much, though we have made some approximations when deriving a linear set of equations of the velocities on the fine mesh. Therefore, the computation time required to regain the correlation between the coarse and fine mesh will decrease.
Thirdly, when a velocity field from another related simulation on a coarser mesh is used as an initial state, the modified multimesh method enables the simulation to reach a statistically stationary state more quickly due to a physically realistic fine scale structure produced by this method.
Figure 4.1: A coarse mesh computational cell with eight fine mesh subcells
Figure 4.2: A coarse mesh computational cell with four fine mesh subcells.
Figure 4.3: A coarse mesh computational cell. with fine mesh subcells.

(c) phase 3
Simulations With Varying Model Constant

5 Simulations With Varying Model Constant

5.1 Problem description

There has been debate among LES researchers about the value of the Smagorinsky model constant $C_s$ since the early LES practices, owing to an apparent lack of a 'universal' value of $C_s$ that can be used for a wide range of flows. For instance, studies of the decay of homogeneous turbulence found that the value of the constant $C_s \approx 0.23$ gives the correct decay. By contrast, Lilly's [34] methodology predicting the model constant gave a value of 0.17 for homogeneous isotropic turbulence. Another value of about 0.2 also based on homogeneous isotropic turbulence was deduced by Métais and Lesieur [40] through assuming that the strain rate ratio is unity ($\langle \mathcal{S} \rangle^3 / \langle \mathcal{S}^3 \rangle$). As we should expect, Smagorinsky models with these values of about 0.2 did a good job for isotropic turbulence. However, for wall bounded inhomogeneous shear flows, this value of $C_s$ had to be reduced by half or more to obtain satisfactory results. This difficulty in reconciling the value of $C_s$ used to obtain the results of channel flows with that used in simulations of homogeneous turbulence has been noted by Deardorff [14] who suggested that $C_s$ might be reduced by the flow rotation occurring near the wall. Other researchers recommended that the reduction in the value of $C_s$ appears to be related to an increase in the backscatter in flows with mean strain or shear [39]. In the present study, the effect of the constant on calculated statistics was investigated by varying $C_s$. 


5.2 The test runs

The investigation of the effect of varying $C_s$ on the turbulence statistics was carried out in a channel flow. The simulated flow is driven by a constant pressure gradient between parallel infinite walls. The friction Reynolds number $Re^+$, based on the mean friction velocity $u_\tau$ and half channel width, is 205. The mean friction velocity is fixed in advance, since the imposed pressure gradient must be balanced by the wall friction in the mean. By contrast, the mean flow velocity varies in the early stage of the simulation. It will eventually settle to a value on which there is no explicit constraint, and thereby the Reynolds number based on mean velocity and full channel width will have a settled value when the simulation reaches a statistically stationary state.

Three simulations using the Smagorinsky model (2.11) with $C_s = 0.2, 0.1$ and 0.08 respectively were performed. Table 5.1 summarises the computational region and the grid. Table 5.2 gives the numerical parameters of the simulations, in which each simulation is denoted by “LES” followed by a number which indicates the varying value of $C_s$, i.e. run LES-08 has $C_s = 0.08$, run LES-10 has $C_s = 0.1$ and run LES-20 has $C_s = 0.2$. From table 5.1, it can be seen that the grid point nearest to the wall is at $y^+_l = 0.68$, which is well inside the wall layer ($y^+ < 5$) and indicates that the resolution in the $y$-direction is sufficient to resolve the wall layer. A natural, no-slip boundary condition is therefore appropriate. The reason for resolving the wall layer explicitly is that under these circumstances, the Smagorinsky model will reveal its drawbacks, such as erroneous subgrid length scale near the wall and lack of a ‘universal’ value of $C_s$. The first problem can be solved satisfactorily by a wall damping function. Here we use

$$1 - \exp(-y^+ / A^+).$$

which has been proved successful in the simulations of channel flow. The second issue
is what we shall concentrate on in this chapter: we vary $C_z$ to observe the subsequent effect. For the other two directions, streamwise $x$ and spanwise $z$ directions, in which the flow is homogenous, the grid points are distributed uniformly. The only requirement for them is that the resolutions in these two directions are the same for all the test runs. High resolution for the investigation being carried out here is unnecessary as long as the subgrid scales filtered out from the Navier-Stokes equations are suitable for modelling.

For comparison, these three simulations all started from the same initial velocity field containing random turbulence on a $1 - y^8$ profile, with an amplitude of disturbance equal to 10. The time step was $0.00065/u_r$, which was chosen to satisfy the stability limits set to ensure that simulations are numerically stable. One is the Courant-Friedrichs-Levy (CFL) number, which is defined as

$$C = \Delta t \left( \frac{u}{\Delta x} + \frac{v}{\Delta y} + \frac{w}{\Delta z} \right) < 0.5. \quad (5.2)$$

The other one is the viscous number, which is defined as

$$\Delta t \nu \left( \Delta x^{-2} + \Delta y^{-2} + \Delta z^{-2} \right) < 0.5. \quad (5.3)$$

For the given resolution in table 5.1, each time step required 0.453 seconds on a Cray YMP8. A total of 20,000 time steps were spent on each of the test runs, which are sufficient for a sound comparison.

<table>
<thead>
<tr>
<th>$N_x \times N_y \times N_z$</th>
<th>$L_x$</th>
<th>$L_y$</th>
<th>$\delta^+$</th>
<th>$\Delta_x^+$</th>
<th>$\Delta_y^+_{\min}$</th>
<th>$\Delta_y^+_{\max}$</th>
<th>$\Delta_z^+$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$48 \times 56 \times 64$</td>
<td>$4\pi \delta$</td>
<td>$2\pi \delta$</td>
<td>$205$</td>
<td>$54\nu/u_r$</td>
<td>$0.68\nu/u_r$</td>
<td>$17.5\nu/u_r$</td>
<td>$20\nu/u_r$</td>
</tr>
</tbody>
</table>
5.3 Results and discussion

Table 5.2: Numerical parameters of the simulations with varying model constant

<table>
<thead>
<tr>
<th>Run No</th>
<th>$R_+^e$</th>
<th>$C_s$</th>
<th>Time Step</th>
<th>$\nu$</th>
</tr>
</thead>
<tbody>
<tr>
<td>LES-20</td>
<td>205</td>
<td>0.20</td>
<td>0.0006</td>
<td>0.004878</td>
</tr>
<tr>
<td>LES-10</td>
<td>205</td>
<td>0.10</td>
<td>0.0006</td>
<td>0.004878</td>
</tr>
<tr>
<td>LES-08</td>
<td>205</td>
<td>0.08</td>
<td>0.0006</td>
<td>0.004878</td>
</tr>
</tbody>
</table>

5.3 Results and discussion

Figures 5.1 - 5.4 illustrate the history of the principal volume-averaged statistics ($u'^2$, $v'^2$, $w'^2$ and $k$) of the simulations LES-20 (dashed line), LES-10 (continuous line) and LES-08 (dotted line). The results show that the turbulence fluctuations in the run LES-20 die out quickly. By contrast, the run LES-10 is capable of turning a random disturbed initial field into something like realistic turbulence, since the value $C_s = 0.1$ used in run LES-10 neither caused the resolved motions to damp out nor allowed turbulent motions to become excessively large. The pictures for $u'^2$, $v'^2$, $w'^2$ and $k$ obtained from run LES-08 are similar to those of run LES-10 but with a different amplitude for each of the various fluctuations which is clearly due to a different value of $C_s$ used in LES-08 from that in LES-10.

The one-dimensional energy spectra in $x$, $y$ and $z$ directions were also computed. They are plotted in figures 5.5 - 5.7 (LES-20, dashed line; LES-10, continuous line; LES-08, dotted line). These pictures show that the energy densities associated with low wavenumbers in the run LES-20 are too low, which indicates that the Smagorinsky SGS model with $C_s = 0.2$ removes too much energy from the large eddies. Note that energy pile-up at high wavenumbers was found in all of the simulations LES-08, LES-10 and LES-20. This arises from the insufficient simulation time. It can be confirmed by pictures (figures 5.8 - 5.10) for the one-dimensional
energy spectra in \( x, y \) and \( z \) directions obtained from LES-10 after a long period of time integration. However, the greater energy pile-up at high wavenumbers found in run LES-20 than that in runs LES-10 and LES-08 suggests again that the value of \( C_s = 0.2 \) causes the resolved motions to damp out. By contrast, the value of \( C_s = 0.1 \) appears to be appropriate for a channel flow, since the energy spectra, obtained from the run LES-10, shown in figures 5.8 - 5.10, reveal that the energy densities associated with the high wavenumbers are several decades lower than the energy densities corresponding to low wavenumbers and there are no energy pile-ups at high wavenumbers. Another interesting finding is that little difference between the energy densities at lower wavenumbers, obtained from LES-10 and LES-08 respectively, was found. This would indicate that a slightly lower value of \( C_s \) does not cause excessively large turbulent motions. So if the value of \( C_s \) is in a certain range, the Smagorinsky model will enable simulations of a channel flow to be successful.

The above findings agree with the results of Deardorff [14]. He performed a series of simulations using the Smagorinsky model with \( C_s \) varying from 0.06 to 0.17 and found that the total statistics were fairly insensitive to the particular choice of \( C_s \) provided \( 0.08 \leq C_s \leq 0.12 \). He also concluded that the optimum value of \( C_s \) for channel flows seemed to be 0.1. This value is chosen to be applied to all the simulations using the Smagorinsky model in the present study. The reason for doing so can be explained as follows. Since \( C_s \Delta \) is related to the resolved turbulence length scale in the Smagorinsky model, for a fixed \( \Delta \), low values of \( C_s \) allow more resolvable scales but have potential finite-difference errors, whereas high values of \( C_s \) limit the range of resolvable scales but give good finite-difference solutions [37]. Therefore, a value in the middle of the range of \( C_s \), to which simulations are not very sensitive, is likely to give a good compromise.
5.3 Results and discussions

Figure 5.1: The history of total volume-averaged $u'^2$.
Dotted line, LES-08; continuous line, LES-10; dashed line, LES-20.

Figure 5.2: The history of total volume-averaged $v'^2$.
Dotted line, LES-08; continuous line, LES-10; dashed line, LES-20.
5.3 Results and discussions

Figure 5.3: The history of total volume-averaged $w^2$.

Dotted line, LES-08; continuous line, LES-10; dashed line, LES-20.

Figure 5.4: The history of total volume-averaged kinetic energy $k$.

Dotted line, LES-08; continuous line, LES-10; dashed line, LES-20.
Figure 5.5: Streamwise one-dimensional energy spectra at \( t = \delta/u_r \).

Dotted line, LES-08; continuous line, LES-10; dashed line, LES-20.

Figure 5.6: Cross-stream one-dimensional energy spectra at \( t = \delta/u_r \).

Dotted line, LES-08; continuous line, LES-10; dashed line, LES-20.
Figure 5.7: Spanwise one-dimensional energy spectra at $t = \frac{9\delta}{u_\tau}$.

Dotted line, LES-08; continuous line, LES-10; dashed line, LES-20.
5.3 Results and discussions

Figure 5.8: LES-10 streamwise one-dimensional energy spectra at $t = 40\delta/u_r$.

Figure 5.9: LES-10 cross-stream one-dimensional energy spectra at $t = 40\delta/u_r$. 
Figure 5.10: LES-10 spanwise one-dimensional energy spectra at $t = 406/u_r$. 

5.3 Results and discussions
6 Simulations With Varying Numerical Resolution

6.1 Problem description

In wall-bounded channel flows, owing to the inhibited growth of turbulent structures near the walls, the turbulence structures become smaller. There are two methods to deal with the wall layer of channel flow. One is to use approximate boundary condition designed to ensure that the logarithmic law is obeyed. The other one is using high resolution in the near-wall region to resolve the wall layer explicitly. The first approach, modelling the effect of the wall layer by the approximate boundary condition, can result in large savings in terms of computer time since it allows larger grid spacings not only in the direction normal to the wall, but in the streamwise and spanwise directions as well. Due to the availability of more powerful computers, the second approach to resolving the wall layer explicitly can be handled at present. Clearly, it has the advantage of giving further insight and understanding into the mechanics of turbulent flows near a wall, where most of the turbulence is produced. In the simulations in which the wall layer is resolved, the quality of turbulence statistics is greatly affected not only by the resolution normal to the wall, but also the resolutions of the streamwise and spanwise directions. That is because when the wall layer is computed directly, the small structures present in the near-wall region have to be captured. Therefore, it is necessary to investigate how varying resolution affects the accuracy of the predicted turbulent structures near the walls and the difference between the effects brought about by the resolution in the normal direction and the other two directions.
6.2 The test runs

The simulations of the same channel flow with the same geometry as described in chapter 5 were performed at three levels, i.e. a mesh with $48 \times 64 \times 40$ grid points, a mesh with $96 \times 64 \times 80$ grid points and a mesh with $192 \times 128 \times 160$ grid points. For this simulated flow at low Reynolds number $R_{e_r}^+ = 205$, there are experimental data of Nishino and Kasagi (1989) and the database produced by a direct simulation of Kim, Moin and Moser*(1987) available for comparison. The characteristics of the simulations, compared with the governing parameters characterizing the simulation of Kim, Moin and Moser, are summarized in table 6.1. LESc, LESm and LESf denote the simulations at the coarse, medium and fine levels respectively. As already presented in table 6.1, the mesh point closest to the wall even at the coarsest level (LESc) is well inside the viscous sublayer ($y^+ < 5$). Therefore the wall layer is resolved explicitly in the simulations at all the three levels. The relations of these three levels are that the number of grid points of LESm is twice as many as that of LESc in the streamwise direction $x$ and spanwise direction $z$ only, while the number of grid points of LESf is doubled in all the three directions compared with LESm, i.e. the streamwise $x$, the spanwise $z$ and the cross-stream $y$ directions. This arrangement enables us to investigate how the grid refinement in the streamwise and spanwise directions affect the turbulence statistics of a channel flow by comparing the results between LESc and LESm. Having gained the knowledge of the effect of the grid refinement in the streamwise and spanwise directions, we can examine the effect of the grid refinement in cross-stream direction on the turbulence statistics of a channel flow through a comparison between the result of LESm and that of LESf since the grid refinement occurs in all the three directions for LESf, compared to LESm.

As has already been pointed out, the mesh points closest to the wall at all

* at $R_{e_r}^+ = 179$, hereafter known as KMM.
6.2 The test runs

the three levels are well inside the viscous sublayer, and therefore a natural, no-slip condition is perfectly appropriate. Here the Smagorinsky model was used, but with low-Reynolds-number modification (3.18) since the simulated flow is a low Reynolds number channel flow. The model coefficient $C_s$ was set to be 0.1 in all these simulations since this value was optimum for turbulent channel flows as suggested in chapter 5.

The run LESc was started from an initial velocity field with random turbulence on a $1 - y^+$ profile, with an amplitude of disturbance equal to 10. The reason for choosing an initial condition of arbitrary form and a large amplitude is to provide a good chance of developing self-sustaining turbulence. For such a resolution of LESc, the time step $\delta t = 0.0004$, and each time step required 0.3244 seconds on a Cray XMP. The overall time spent on this run was $88 \delta/\nu$, approximately 2 CPU hours. Between $t = 80\delta/\nu$ and $t = 88\delta/\nu$, the statistical samples were gathered.

The run LESm was started from the instantaneous state of the run LESc at $t = 88$ by interpolating the instantaneous velocity field of the run LESc onto the grid points of the mesh used in the run LESm. The number of grid points of the mesh used in the run LESm is twice as many as that of the mesh used in the run LESc in both of the streamwise ($x$) and spanwise ($z$) directions, but remains the same as that of the mesh used in the run LESc in the $y$ direction normal to the wall. Figure 6.1 shows a combined finite volume cell of LESc containing a block of four subcells of LESm. Since the method of interpolation used in the multiple mesh simulation, which can create physically realistic fine mesh scale structures to interact with the flow simulated on the coarse mesh, has not been worked out, a simple linear interpolation algorithm is used here to generate the velocity field on
the mesh used in the run LESm, i.e.

\[
\begin{align*}
    v_{m1} &= v_{m2} = v_c \\
    v_{m1} &= v_{m2} = v_{m3} = v_{m4} = v_c \\
    w_{m1} &= w_{m3} = w_c
\end{align*}
\]  

where \( u_c, v_c \) and \( w_c \), having been obtained by the simulation of LESc, are the surface average velocities through the whole faces, perpendicular to the \( x \), \( y \) and \( z \) axes respectively, of the combined finite volume cell, and \( u_{mi}, v_{mi} \) and \( w_{mi} (i = 1, 2, 3, 4) \) are the individual parts of \( u_c, v_c \) and \( w_c \) respectively and are used as an initial velocity field to start the run LESm. It is easy to prove that \( u_{mi}, v_{mi} \) and \( w_{mi} (i = 1, 2, 3, 4) \) obey the discrete continuity condition on the mesh used in the run LESm.

Being started from \( t = 88\delta/u_r \), the simulation was continued till \( t = 107.8\delta/u_r \), the statistical samples being gathered between \( t = 101.5\delta/u_r \) and \( t = 107.8\delta/u_r \). For such a resolution of LESm, the time step \( \delta t = 0.0003 \) and each time step required 1.2259 seconds on a Cray XMP. The total time spent on the simulation was \( 19.8\delta/u_r \), approximately 22 hours.

The run LESf was started from the instantaneous state of the run LESm at \( t = 107.8\delta/u_r \) and continued till \( t = 108.9\delta/u_r \), the statistical samples being gathered during the last \( 0.3\delta/u_r \). For such a resolution of LESf, the time step \( \delta t = 0.0001 \) and each time step required about 10 seconds on a Cray YMP8. The total time expended on the simulation was \( 1.1\delta/u_r \), using approximately 30 CPU hours. As already noted in table 6.1, the number of grid points of the mesh used in the run LESf is doubled in all the three directions \( x \), \( y \) and \( z \), compared with the mesh used in the run LESm. Figure 6.2 illustrates a combined finite volume cell of LESm containing a block of eight subcells of LESf. The same linear interpolation algorithm was employed for obtaining the velocity field on the mesh used in the run LESf from that of the run LESm. The velocity field on the mesh used in the run
Table 6.1: The characteristics of the simulations with varying numerical resolution

<table>
<thead>
<tr>
<th>symbol</th>
<th>KMM(1987)</th>
<th>LESc</th>
<th>LESm</th>
<th>LESf</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\delta^+$</td>
<td>179</td>
<td>205</td>
<td>205</td>
<td>205</td>
</tr>
<tr>
<td>$N_x \times N_y \times N_z$</td>
<td>192 $\times$ 129 $\times$ 160</td>
<td>48 $\times$ 64 $\times$ 40</td>
<td>96 $\times$ 64 $\times$ 80</td>
<td>192 $\times$ 128 $\times$ 160</td>
</tr>
<tr>
<td>$L_x \times L_y \times L_z$</td>
<td>$4\pi\delta \times 2\delta \times 2\pi\delta$</td>
<td>$4\pi\delta \times 2\delta \times 2\pi\delta$</td>
<td>$4\pi\delta \times 2\delta \times 2\pi\delta$</td>
<td>$4\pi\delta \times 2\delta \times 2\pi\delta$</td>
</tr>
<tr>
<td>$\Delta_t^+$</td>
<td>$12\nu/\mu_r$</td>
<td>$52\nu/\mu_r$</td>
<td>$26\nu/\mu_r$</td>
<td>$13\nu/\mu_r$</td>
</tr>
<tr>
<td>$\Delta_t^+_{\text{min}}$</td>
<td>$0.05\nu/\mu_r$</td>
<td>$0.635\nu/\mu_r$</td>
<td>$0.635\nu/\mu_r$</td>
<td>$0.32\nu/\mu_r$</td>
</tr>
<tr>
<td>$\Delta_t^+_{\text{max}}$</td>
<td>$4.4\nu/\mu_r$</td>
<td>$15.22\nu/\mu_r$</td>
<td>$15.22\nu/\mu_r$</td>
<td>$7.61\nu/\mu_r$</td>
</tr>
<tr>
<td>$\Delta_t^+$</td>
<td>$7\nu/\mu_r$</td>
<td>$31\nu/\mu_r$</td>
<td>$15.5\nu/\mu_r$</td>
<td>$7.75\nu/\mu_r$</td>
</tr>
<tr>
<td>time step</td>
<td>—</td>
<td>0.0004</td>
<td>0.0003</td>
<td>0.0001</td>
</tr>
</tbody>
</table>

LESf is therefore given in the form,

\[
\begin{align*}
    u_{f1} &= u_{f2} = u_{f3} = u_{f4} = u_m \\
    v_{f3} &= v_{f4} = v_{f7} = v_{f8} = v_m \\
    w_{f1} &= w_{f3} = w_{f5} = w_{f7} = w_m
\end{align*}
\]  

(6.2)

where $u_m$, $v_m$ and $w_m$, having already been obtained from the simulation of LESm, are the surface average velocities through the whole faces, perpendicular to the x, y and z axes respectively, of the combined finite volume cell, $u_{fi}$, $v_{fi}$ and $w_{fi}$ ($i = 1, 2, \ldots, 8$) are the individual parts of $u_m$, $v_m$ and $w_m$ respectively and are used as an initial velocity field to start the run LESf. It is not difficult to show that $u_{fi}$, $v_{fi}$ and $w_{fi}$ ($i = 1, 2, \ldots, 8$) obey discrete continuity condition on the mesh used in the run LESf.
6.3 Results and discussion

The first important finding is that the turbulence statistics gathered on the finest mesh (LESf) are comparable to the results of Kim, Moin and Moser, whose simulation was started from an initial velocity field obtained from the large eddy simulation of Moin and Kim [41] (64 x 63 x 128) which was interpolated by spectral interpolation onto a mesh with a similar number of degrees of freedom (192 x 129 x 160) as that of the mesh used in LESf, and then was integrated forward in time on this mesh until the flow reached a statistically steady state. But their simulation spent 4 times more computer time per time step and 8.3 times more overall computer time than that spent on LESf, which was started from an initial velocity field obtained from LESm (96 x 128 x 80), and even spent 4.6 times more computer time than the total time spent on LESc started from a random initial velocity, LESm and LESf together. The much higher cost in terms of computer time of the simulation performed by Kim, Moin and Moser was caused by their extremely stretched mesh, which gives the nearest collocation point to the wall a value of 0.05u/U (see table 6.1), since for such a high resolution, 6.4 times higher than that of LESf, near the wall, viscous term must be treated implicitly to allow time steps longer than the viscous unit, otherwise very small time steps will result in a much tighter viscous limit than the CFL limit. The reason for their choosing such a high resolution near the wall is that their simulation did not contain a SGS model and thereby has to ensure the resolution down to the Kolmogorov range everywhere. With 129 Chebychev polynomials chosen by their simulation, the resolution in the central region of the channel flow is just fine enough for a full simulation, but the resolution near the wall has become very fine. By contrast, despite the similar number of degrees of freedom in the present study as in Kim, Moin and Moser, the currently used mesh was moderately stretched in the direction normal to the walls to restrict the resolution near the wall to a level where the viscous and CFL limits
are comparable. As has already been compared earlier, this arrangement can result in large savings in terms of CPU time.

The history of the principal volume-averaged statistics \( (u_{\text{mean}}, w^2, v^2, w^2 \text{ and kinetic energy } k) \) is shown in figure 6.3. It can be seen that when the simulation was switched to the next finer mesh, \( u^2 \) went down. On the contrary, \( v^2 \) and \( w^2 \) jumped upwards. After about \( 5 \delta/u_r \), the mean square values of \( u^2, v^2 \) and \( w^2 \) settled down to their new mean square values, around which they fluctuated. The explanation for these upward and downward jumps will be given later. Also it was noted that the turbulence kinetic energy almost kept fluctuating around the same mean square value, which indicates the conservation of turbulence kinetic energy.

The present results of the mean properties, the velocity fluctuations, Reynolds shear stresses and higher order statistics are compared with those of the simulation of Kim, Moin and Moser (1987) (hereafter referred to as KMM), a spectral simulation performed using \( 192 \times 160 \) Fourier modes and 129 Chebyshev polynomials. The Reynolds number of their simulation based on mean velocity and full channel width, the Reynolds number based on centreline velocity and full width and the friction Reynolds number based on the mean friction velocity and half channel width were \( R_{e_{\text{mean}}} = 5600, \ R_{e_{\text{centre}}} = 6600 \) and \( R^+_{e_r} = 179 \) respectively, the channel centre being at \( \delta^+ = 179 \) in wall units. Comparison is also made with the experimental results of Nishino and Kasagi (1989) (hereafter referred to as NK), whose data were gathered using particle tracking velocimetry in water at \( R_{e_{\text{mean}}} = 6560, \ R_{e_{\text{centre}}} = 7510 \) and \( R^+_{e_r} = 205 \) with the channel centre being at \( \delta^+ = 205 \) in wall units.

The mean velocity profiles of LESc (solid triangles), LESm (solid dots) and LESf (solid boxes) are shown in comparison with those of KMM (medium dashed line) and NK (continuous line) in figure 6.4. Within the viscous sublayer \( y^+ < 5 \), all the results at the three levels (LESc, LESm and LESf), like those of KMM
and NK, exactly follow the linear law of the wall (dotted line), which indicates that the linear sublayer has been resolved perfectly even in the run LESc whose resolution is lowest, and the linear law of the wall $u_+^{\text{mean}} = y^+$ in the region $y^+ < 5$ has been clearly captured in the present simulation as in KMM. In the logarithmic region, however, there exist noticeable disagreement between the present results and those of KMM and NK. Firstly, the intercept of the logarithmic layer of the run LESc is higher than those of KMM and NK. However, the intercept becomes lower as the grid resolution increases even in the streamwise ($x$) and the spanwise ($z$) directions only, like the simulation moving from LESc to LESm which has grid points twice as many as those of LESc in the $x$ and $z$ directions. This discrepancy has been found by Piomelli, Ferziger and Moin [48] in a comparison of various mean velocity profiles with varying resolution. They explained that insufficient resolution in the streamwise and spanwise directions, like insufficient resolution in the cross-stream direction, results in turbulence structures that are too large in the normal direction as well. These unrealistically large turbulence structures lead to a wall layer thickness larger than its physical counterpart, which in turn results in underprediction of the wall stress and in overprediction of the logarithmic layer intercept. Therefore, we should expect that the intercept of the logarithmic layer of LESc decreases if resolution is increased. This argument has been confirmed by a comparison between the mean velocity profiles of LESc, LESm and LESf. Secondly, the mean velocity profile of LESc shows an slightly excessive slope of the logarithmic layer, which is also considered to be attributed to its insufficient resolution. When the resolution is higher, as in the runs of LESm and LESf, the problem is solved. Thirdly, although the mean velocity profiles of LESm and LESf perfectly follow the log law in the logarithmic region, 5 is used for the additive constant in the log law here in contrast to 5.5, the value giving a log law which the results of KMM and NK closely match. In addition, the mean profile of LESc shows a slightly higher peak value, which is due to its insufficient resolution as pointed out earlier. Nevertheless,
the peak value of the mean velocity should decrease if resolution becomes higher. However, the peak value of the mean velocity did not reach a value of KMM at LESf level, whose number of degrees of freedom is similar as that of KMM. In fact, compared with the peak values of the mean velocities of KMM (18.2) and NK (18.24), the peak values of the mean velocities of LESm (17.87) and LESf (17.99) are slightly lower. The reason for this discrepancy may be that the present simulations were performed with a fixed pressure gradient, not with a fixed mass flow rate, so there is no explicit constraint on the mean flow rate. There is also another possible reason for this quite noticeable deviation. That is the linear interpolation algorithm used for generating the velocity field on the finer mesh from the coarser mesh, such as generating LESm velocity field from LESc and LESf velocity field from LESm. The velocity interpolation process using these linear interpolation algorithms (6.1) and (6.2) will cause the resolved field to lose energy because the resolved velocity fields corresponding to the finer mesh resulting from equation (6.1) or (6.2) from that of the preceding coarser mesh, and therefore has the same kinetic energy as that of the coarser mesh whose resolved turbulence scales should carry less kinetic energy. To regain the lost energy may need a long simulation time. However, the integration time spent is 19.8 time units at LESm level and only 1.1 time units at LESf level. Such a simulation period is apparently insufficient.

The streamwise fluctuation $u_{rms}$, spanwise fluctuation $w_{rms}$, and cross-stream fluctuation $v_{rms}$ at the three levels (LESc, plotted by solid triangles; LESm, plotted by solid dots; and LESf, plotted by solid boxes), are shown in figures 6.5 - 6.7 in comparison with those of KMM (medium dashed line) and NK (continuous line). The improvement brought about by grid refinement is significant. Apart from a slightly lower peak value of $u_{rms}$, which is caused by the linear interpolation algorithm 6.2 again, and slight oscillations existing in the profiles of $v_{rms}$ and $w_{rms}$ in the central region, which arise from apparently inadequate simulation time (only 1.1
6.3 Results and discussion

\( \delta/u_r \), the statistics of the velocity fluctuations \( u_{rms}, v_{rms}, \) and \( w_{rms} \) gathered from the run LESf agree excellently with those of KMM and NK.

The pictures for the velocity fluctuations (figures 6.5 - 6.7) show that the streamwise fluctuation \( u_{rms} \) goes down; on the contrary, the spanwise fluctuation \( w_{rms} \) and cross-stream fluctuation \( v_{rms} \) go up as the resolution becomes higher. We also found great improvements in the profiles of velocity fluctuations of \( v \) and \( w \), especially in the near-wall region, brought about by the increased resolution in the streamwise and spanwise directions only, as in LESm. The fact suggests that insufficient resolution in the streamwise and spanwise directions leads to too large a wall layer thickness as insufficient resolution in the normal direction does, which in turn causes low normal and spanwise velocity fluctuations. In addition, insufficient resolution also cause errors in the mean velocity profile, which may result in increased turbulence production and high value of the streamwise fluctuation. That also explains the reason for the downward jumps of \( u'^2 \) and the upward jumps of \( v^2 \) and \( w^2 \) at the moment of the interpolation from the coarser mesh onto the finer mesh, which are found in figure 6.3 illustrating the history of the principal volume-averaged statistics.

Of considerable promise is that the cross-stream fluctuation \( v_{rms} \) obtained from the run LESf show a high level of agreement with the result of KMM over the whole channel and even \( v_{rms} \) obtained from the run LESm agrees well with that of KMM up to \( y^+ = 12 \). Such a high level of agreement suggests again that the extremely high resolution of KMM in the near-wall region, which gives the nearest grid point to the wall a value of 0.05, would be unnecessary, since LESf, whose grid point closest to the wall is at \( y^+ = 0.32 \), more than 6 times larger than that of KMM, is capable of resolving the parabolic behaviour of the cross-stream fluctuation \( v_{rms} \). Even the run LESm, whose grid point nearest to the wall is at \( y^+ = 0.635 \), more than 12 times larger than that of KMM, perfectly shows a correct near wall behaviour of \( v_{rms} \). As
Voce [68] has already pointed out, such a high resolution near the wall used in the simulation of KMM results in a viscous limit much tighter than the CFL limit. The problem can be solved either by introducing an implicit treatment of the viscous terms, which will raise questions concerning the accuracy of the simulation close to the wall, or by reducing the time step, which will greatly increase the computational time. The remedy in the present study for this dilemma is reducing the resolution near the wall by a gentle stretch in the normal direction to allow the viscous and CFL constraints to be comparable. Doing so can avoid unnecessary high resolution near the wall, which results from Chebyshev polynomial expansion in the normal direction. Figure 6.6 has confirmed its success.

The shear stress profiles (figure 6.8) show that, as we should expect, the grid scale Reynolds stress increases as the grid is refined. The agreement with the results of KMM and NK is good even at the coarsest level LESc.

The computed skewness and flatness of the streamwise, spanwise and cross-stream velocity fluctuations at all the three levels (LESc, LESm and LESf) are plotted in the figures 6.9 - 6.14. The comparison between these computed higher-order statistics and the database of KMM and the experimental data of NK reveals the improvement brought about by grid refinement.

The skewness of \( u \) (figure 6.9) of LESf shows good agreement with those of KMM and NK except in the region very close to the wall. Of particular significance is that the crossover point, at which the skewness is zero, has been brought by grid refinement to approximately \( y^+ = 13 \) which is very close to those of KMM and NK. In the central region, the skewness of \( u \) obtained from the run LESf has a higher level of agreement with that of NK than that of KMM. This may be attributed to a lower Reynolds number of KMM (\( R_{eq} = 179 \) for the simulation of KMM, by contrast, \( R_{eq} = 205 \) for the present study and the experiment of NK). When the
wall is approached, the value of LESf is about 1.2, which is higher than 0.9 from KMM, but close to 1.1 for the experiments of Kreplin and Eckelmann (1979) and Alfredsson et al. (1988) [68]. The behaviour of $u$ skewness from NK appears to be erroneous in the near-wall region and is not available very close to the wall.

The level of agreement of the flatness of $u$ (figure 6.10) of LESf with those of KMM and NK is good above $y^+ = 10$. Again, the result of LESf shows a better agreement with that of NK than that of KMM. The main disagreement occurs when the wall is approached. However, grid refinement can largely reduce this disagreement (see figure 6.10). We note that increasing resolution only in the streamwise and spanwise directions has brought about some evident improvement on the flatness of $u$ in the central region, but did not make any improvement in the near-wall region. This can be seen from the comparison of the flatness of $u$ between LESc and LESm. Nevertheless, flatness of LESf, whose resolution is doubled in all the three directions compared with LESm, shows an evident improvement over the whole channel. This may indicate that the resolution in the direction normal to the wall is more important to the behaviour of the flatness of $u$ in the near-wall region. In addition, considering that the normal resolution in the near-wall region of KMM is much higher than that of LESf, we believe that the lower normal resolution of LESf near the wall contributes to the disagreement in the pictures when the wall is approached.

The profile for the skewness of $v$ (figure 6.11) also shows an evident improvement brought about by the grid refinement, especially on the crossover point which has been brought by the simulation at the finest level (LESf) to a point very close to $y^+ = 12$, the position of the crossover points of KMM and NK. However, it is interesting to note that the sign of the skewness of $v$ in the near-wall region even at the finest level (LESf) is contrary to those of KMM and NK. However, the present results seem to agree with the quadrant analysis made by Kim, Moin and Morser[29].
They suggested that the most violent Reynolds shear-stress-producing events are from the fourth quadrant ($u' > 0$ and $v < 0$) for $y^+ < 12$. It can be seen from figures 6.9 and 6.11 that for $y^+ < 12$, the skewness of $u'$ is positive and therefore there is a positive $u'$ corresponding to this positive skewness of $u'$. On the contrary, the skewness of $v$ is negative and therefore there is a negative $v'$ corresponding to this negative skewness of $v$. This indicates that $u'$ is well correlated with $v$ in the region $y^+ < 12$. This correlation between $u'$ and $v$ in turn results in a large negative $u'v$ from the fourth quadrant ($u' > 0$ and $v < 0$) for $y^+ < 12$, which complies with the quadrant analysis of Kim, Moin and Moser. By contrast, the results of KMM and NK give both the skewness of $u'$ and skewness of $v$ a positive value for $y^+ < 10$ (see figures 6.9 and 6.11). This suggests that $u'$ is not correlated with $v$ in the region $y^+ < 10$ since in this region the shear stress $u'v$ is negative (see figure 6.8). In addition, figure 6.11 shows that the sign of the skewness of $v$ for the present results only change once rather than twice like the skewness of $v$ for the results of KMM and NK. Therefore, the present skewness of $v$ has a larger excursion of negative $v$ (responsible for the negative skewness) with a crossover point at $y^+ \sim 32$. In the central region, the present results show an apparent deviation from those of KMM and NK, which is most likely due to the inadequate simulation time spent at the LESf level, and therefore inadequate sample size. Again, the slightly higher Reynolds number of LESf than that of KMM make the curve of $v$ skewness of LESf end at $y^+ = 205$ in contrast to $y^+ = 179$ at which the curve of $v$ skewness of KMM ends. As the wall is approached, the very high value predicted by NK would not be reliable.

The picture for the flatness of $v$ (figure 6.12) shows a significant improvement for the flatness of $v$ brought about by the grid refinement again. At the finest level (LESf), good agreement with KMM and NK over most of the channel is evident, except in the wall layer $y^+ < 5$, where the flatness of $v$ of LESf is approaching about
6.3 Results and discussion

14, while that of KMM is as high as 21. This disagreement is most likely due to the lower normal resolution of LESf in the near-wall region since the value of $v_{flatness}$ of LESc, whose resolution is twice lower than that of LESf in the normal direction $y$, is even lower when the wall is approached. The apparently divergent result of NK in the near-wall region would appear to be erroneous. LESm found similar behaviour of the flatness to that of NK. The reason for this will be discussed later.

The non-zero value of the skewness of $w$ (figure 6.13) at LESm and LESf levels, which should be zero because of the reflection symmetry of the solutions of the Navier-Stokes equations, indicates again that the adequacy of the sample size used to obtain the higher-order statistics is only marginal.

The flatness of $w$ of LESf agrees well with that of KMM (figure 6.14). There exists only a slight deviation from KMM as the wall is approached, where no data of NK is available for comparison.

Furthermore, there are two noticeable issues found in the pictures for the skewness and flatness of $u'$, $v$ and $w$ fluctuations (figures 6.9 - 6.14) which need further discussion. Firstly, the experimental data of high-order statistics, such as skewness and flatness, are not always reliable or not available in the region very close to the wall. For example, the experimental results of NK used here for comparison either gives erroneous behaviour or lack data approaching the wall. The reason for this could be due to the difficulties encountered during their experimental measurements in the very near-wall region. This indicates that the cross-stream resolution near the wall is very important in predicting the correct behaviour of the higher-order statistics, i.e. in the near-wall region, higher-order statistics are more sensitive to the cross-stream resolution than the lower-order statistics like velocity fluctuations or shear stresses. Also the failure of experimentation in the near-wall region confirms that computation definitely has advantages over experimentation at least in
6.3 Results and discussion

the region close to the wall.

Secondly, it has been noticed that, in the near-wall region, the level of the agreement of skewness and flatness between LESm and KMM is even lower than that between LESc and KMM, though the results of LESm have already shown an evident improvement on the profiles of the skewness and flatness in the central region, including the crossover points. It is also interesting to note that the trend of flatness of $v$ fluctuation for LESm is similar to that of NK, which is divergent when the wall is approached. Since LESm was performed on a mesh whose number of grid points is twice as many as that of LESc in the streamwise $x$ and the spanwise $z$ directions only and remains the same in the cross-stream direction, it appears to indicate that grid refinement only in the $x$ and $z$ directions will bring about the improvement in the streamwise mean velocity profile (figure 6.4), the velocity fluctuation profiles (figures 6.5 - 6.7), and the higher-order statistics in the central region, but somehow will cause high flatness of velocity as the wall is approached, which represents high intermittency of velocity near the wall. The above two facts suggest that the resolution in the cross-stream direction will affect higher-order statistics more strongly in the near-wall region. The high resolution in the cross-stream direction improves all the turbulence statistics over the whole channel, while the high resolution in the streamwise and spanwise directions improves lower-order turbulence statistics, such as the streamwise mean velocity and the velocity fluctuations over the whole channel, but only improves the profiles of higher-order statistics in the central region of the channel.
6.3 Results and discussion

Figure 6.1: A LESc mesh computational cell with LESm mesh subcells.

Figure 6.2: A LESm mesh computational cell with LESf mesh subcells.
Figure 6.3: The history of the simulation.

Bottom to top: total volume average $v^2$, $w^2$, $u_{\text{mean}}/10$, $k$, $u'^2$. 
Figure 6.4: Mean streamwise velocity $u_{\text{mean}}$ versus $y^+$. 

Solid triangles, LESc; solid dot, LESm; solid box, LESf; continuous line, NK; medium dashed line, KMM; dotted line, $u^+ = y^+$; long dashed line, $u^+ = 2.5 \ln y^+ + 5.0$; dot dashed line, $u^+ = 2.5 \ln y^+ + 5.5$. 
Figure 6.5: Streamwise velocity fluctuation $u_{rms}$ versus $y^+$. 
Solid triangles, LESc; solid dots, LESm; solid boxes, LESf; continuous line, NK; 
dashed line, KMM.
6.3 Results and discussion

Figure 6.6: Spanwise velocity fluctuation $w_{rms}$ versus $y^+$
Solid triangles, LESc; solid dots, LESm; solid boxes, LESf; continuous line, NK; dashed line, KMM.
Figure 6.7: Cross-stream velocity fluctuation $u_{rms}$ versus $y^+$
Solid triangles, LESc; solid dots, LESm; solid boxes, LESf; continuous line, NK;
dashed line, KMM.
Figure 6.8: Principal Reynolds stress component $u'v$

Solid triangles, LESc; solid dots, LESm; solid boxes, LESf; continuous line, NK; dashed line, KMM.
6.3 Results and discussion

Figure 6.9: Skewness of the streamwise velocity fluctuations
Solid triangles, LESc; solid dots, LESm; solid boxes, LESf; continuous line, NK; dashed line, KMM.
Figure 6.10: Flatness of the streamwise velocity fluctuations
Solid triangles, LESc; solid dots, LESm; solid boxes, LESf; continuous line, NK;
dashed line, KMM.
Figure 6.11: Skewness of the cross-stream velocity fluctuations
Solid triangles, LESc; solid dots, LESm; solid boxes, LESf; continuous line, NK;
dashed line, KMM.
6.3 Results and discussion

Figure 6.12: Flatness of the cross-stream velocity fluctuations
Solid triangles, $\text{LES}_c$; solid dots, $\text{LES}_m$; solid boxes, $\text{LES}_f$; continuous line, NK;
dashed line, KMM.
Figure 6.13: Skewness of the spanwise velocity fluctuations
Solid triangles, LESc; solid dots, LESm; solid boxes, LESf; continuous line, NK; dashed line, KMM.
6.3 Results and discussion

Figure 6.14: Flatness of the spanwise velocity fluctuations
Solid triangles, LESc; solid dots, LESm; solid boxes, LESf; continuous line, NK; dashed line, KMM.
7 Simulations With Low-Reynolds-Number Modification

7.1 Problem description

As described before, the simulated flow in the present study is a low Reynolds number channel flow. For such a flow, problems will arise when the resolution becomes higher and therefore the grid-scale cutoff falls in the dissipation range, where significant grid-scale molecular dissipation occurs and thereby is no longer negligible compared to the dissipation caused by eddy viscosity. Under these circumstances, the fundamental assumption of LES, namely that the energy extracted from grid-scale eddies is almost all dissipated by SGS scale eddies, is inappropriate. Therefore some modification to the standard Smagorinsky model should be made to account for the increase of molecular dissipation as the grid-scale cutoff lies in the dissipation range. The way of achieving this in the present study is to use a modified eddy viscosity (3.18) proposed by Voke [68], which is rewritten here for convenience:

\[ \nu_t = \nu_s - \beta \nu \left[ 1 - \exp \left( -\frac{\nu_s}{(\beta \nu)} \right) \right] \]  \hspace{1cm} (7.1)

where \( \beta = \frac{2}{5} \). The exponential term in the above formulation represents the reduction in the energy transfer and therefore the decrease of the eddy viscosity in the dissipation region. The ability of this modification to the eddy viscosity for a channel flow at low Reynolds number will be tested here by making a comparison between a test run using the standard Smagorinsky model and the other one using the modified low-Reynolds-number Smagorinsky model.
7.2 The test runs

Two simulations of the same channel flow with the same geometry as that in chapter 5 were performed. One of them is using a standard Smagorinsky model, the other is also using the Smagorinsky model but with the low-Reynolds-number modification. The characteristics of the simulations are summarized in table 7.1. LES1 and LES0 denote the simulations with and without low-Reynolds-number modification respectively.

As shown in table 7.1, the mesh point closest to the wall is at $y_1^+ = 0.635$, which is well inside the viscous sublayer, and therefore a natural, no-slip condition is perfectly appropriate. The Smagorinsky model constant $C_s$ has a value of 0.1 in both of the simulations.

Both of the runs LES1 and LES0 were started from the instantaneous state of LESm at $t = 101.5\delta/u_r$. The run LES1 was continued till $t = 109\delta/u_r$, the statistical samples being gathered between $t = 105.1\delta/u_r$ and $t = 109\delta/u_r$, while the run LES0 was continued till $t = 125.2\delta/u_r$, the statistical samples being gathered between $t = 110\delta/u_r$ and $t = 125.2\delta/u_r$. Each time step of the both runs required 1.2259 seconds on a Cray XMP. The overall time spent on the run LES1 was $7.5\delta/u_r$, approximately 8.5 CPU hours, and that on the run LES0 was $23.7\delta/u_r$, approximately 26.9 CPU hours. The reason for running LES0 for a longer period is that the modified low-Reynolds-number Smagorinsky model was also applied to the run LESm. Since both of LES1 and LES0 were started from an instantaneous state of LESm, LES0 needs a longer integration time to settle down.
7.3 Results and discussion

Table 7.1: The characteristics of the simulations with low Reynolds number modification

<table>
<thead>
<tr>
<th>symbol</th>
<th>LES1</th>
<th>LES0</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\delta^+$</td>
<td>205</td>
<td>205</td>
</tr>
<tr>
<td>$N_x \times N_y \times N_z$</td>
<td>$96 \times 64 \times 80$</td>
<td>$96 \times 64 \times 80$</td>
</tr>
<tr>
<td>$L_x \times L_y \times L_z$</td>
<td>$4\pi \delta \times 2\delta \times 2\pi \delta$</td>
<td>$4\pi \delta \times 2\delta \times 2\pi \delta$</td>
</tr>
<tr>
<td>$\Delta^+_x$</td>
<td>$26\nu/u_r$</td>
<td>$26\nu/u_r$</td>
</tr>
<tr>
<td>$\Delta^+<em>y</em>{\text{min}}$</td>
<td>$0.635\nu/u_r$</td>
<td>$0.635\nu/u_r$</td>
</tr>
<tr>
<td>$\Delta^+<em>y</em>{\text{max}}$</td>
<td>$15.22\nu/u_r$</td>
<td>$15.22\nu/u_r$</td>
</tr>
<tr>
<td>$\Delta^+_y$</td>
<td>$15.5\nu/u_r$</td>
<td>$15.5\nu/u_r$</td>
</tr>
<tr>
<td>model</td>
<td>$\nu_t = \nu_s - \frac{2}{3} \nu \left[ 1 - \exp \left( -\frac{2}{3} \nu_s/\nu \right) \right]$</td>
<td>$\nu_t = \nu_s$</td>
</tr>
<tr>
<td></td>
<td>$\nu_s = (C_s \Delta)^{2.5}$</td>
<td></td>
</tr>
</tbody>
</table>

7.3 Results and discussion

To illustrate the effectiveness of the modification to the eddy viscosity for a channel flow at low Reynolds number, the results of LES0 (solid triangles) and LES1 (solid dots) are shown in comparison with the database of KMM (dashed line) and the experimental result of NK (continuous line).

Figure 7.1 illustrates the mean velocity profile. The first finding from this picture is that the peak value of the mean streamwise velocity of LES1 (17.95) is higher than that of LESm (17.87). Since the only difference between LES1 and LESm is that LES1 ran $1.2 \delta/u_r$ longer than LESm, figure 7.1 confirms the argument in chapter 6 that the resolved velocity field of a finer mesh will lose energy at the moment of interpolation from a coarser mesh onto the finer one and needs a long period to regain the lost energy if the simple linear interpolation algorithm described
7.3 Results and discussion

there is used. In addition, this picture shows an evident improvement brought about by the low-Reynolds-number modification over the whole channel, particularly in the near wall region where LES1 closely follows the linear law of the wall and attains a higher level of agreement with KMM and NK than LES0.

The streamwise turbulence intensity $u_{rms}$ is shown in figure 7.2. This picture reveals that the profile of $u_{rms}$ of LES0 agrees slightly better with the database of KMM and the experiment of NK than that of LES1 in the near-wall region. In the central region, the agreement between LES1 and NK is better than the agreement between LES1 and KMM, on the contrary, the agreement between LES0 and KMM is better than the agreement between LES0 and NK.

The picture for the cross-stream fluctuation $v_{rms}$ (figure 7.3) is very impressive, and shows a considerable improvement on the $v_{rms}$ profile, brought about by the low-Reynolds-number modification. The modification raises $v_{rms}$ which was too low. Of particular significance is the collapse of the $v_{rms}$ profiles corresponding to LES1 and KMM in the near-wall region.

The spanwise fluctuation $w_{rms}$ (figure 7.4) reveals a similar picture to that for the cross-stream fluctuation. The low-Reynolds-number modification has greatly improved $w_{rms}$ profile over the whole channel. Particularly in the near-wall region, the $w_{rms}$ profile of LES1 overlap that of KMM.

The shear stress $u'v$ is shown in figure 7.5 which reveals a higher value of shear stress obtained by the run LES1. This is due to the reduced eddy viscosity resulting from the low-Reynolds-number modification.

From all the pictures for the turbulence statistics, it can be seen that the effect of the low-Reynolds-number modification on the predicted turbulence statistics is to raise the values of the turbulence statistics. Sometimes it results in a value which is too high, such as the streamwise fluctuation $u_{rms}$ or the shear stress $u'v$. This
suggests that the low-Reynolds-number modification does not always improve the results of LES with a cutoff in the dissipation region. In addition, the ability of this modification to improve the result of channel flow simulation at low Reynolds number differs over the channel. As presented above, it is more effective in the near-wall region.
Figure 7.1: Mean streamwise velocity $u_{\text{mean}}$ versus $y^+$. 

Solid triangles, LES0; solid dot, LES1; continuous line, NK; dashed line, KMM; dotted line, $u^+ = y^+$ and $u^+ = 2.5 \ln y^+ + 5.0$. 
7.3 Results and discussions

Figure 7.2: Streamwise velocity fluctuation $u_{rms}$ versus $y^+$. Solid triangles, LES0; solid dots, LES1; continuous line, NK; dashed line, KMM.
Figure 7.3: Spanwise velocity fluctuation $w_{rms}$ versus $y^+$
Solid triangles, LES0; solid dots, LES1; continuous line, NK; dashed line, KMM.
7.3 Results and discussions

Figure 7.4: Cross-stream velocity fluctuation $v_{rms}$ versus $y^+$
Solid triangles, LES0; solid dots, LES1; continuous line, NK; dashed line, KMM.
7.3 Results and discussions

(a) Linear y axis.

(b) Logarithmic y axis.

Figure 7.5: Principal Reynolds stress component $u'v$
Solid triangles, LES0; solid dots, LES1; continuous line, NK; dashed line, KMM.
8 Simulations Using Dynamic Subgrid-Scale Model

8.1 Problem description

The need, arising from the Smagorinsky model, for an ad hoc wall damping function to obtain the predicted SGS stresses with correct asymptotic behaviour in the near-wall region causes difficulties in simulating flows in which the form of the wall damping functions is unknown or in which the concept of a wall damping function is inappropriate or inapplicable. In addition, the model constant $C_s$ involved in the Smagorinsky model must be adjusted from flow to flow to which it is applied. Furthermore, the absolute positive value of the Smagorinsky eddy viscosity rules out energy backscatter completely. Clearly these major shortcomings hinder the Smagorinsky model from being used in more complex flows or in a wide range of flows. A new SGS model, called the dynamic SGS model proposed by Germano et al. [20], has already shown its immense promise in tackling the foregoing difficulties encountered by the Smagorinsky model. The success of this new SGS model is attributed to its unique ability to dynamically compute the model coefficient as a function of space and time, according to the behaviour of the simulated flow. The mathematical framework of this model has been described in section 2.2.2 and the formula actually used for the low Reynolds channel flow being simulated in the present study was given in section 3.2.2. Although a number of technical problems in the implementation of the dynamic model, such as overspecification of the model coefficient and numerical instabilities, have been, in principle, overcome over the past four years, other developments in this technique are still being undertaken, such as determining the extent to which the technique can be applied to more complex or low Reynolds number flows. As some researchers [7] have found,
low Reynolds number LES results for channel flow indicate a greater sensitivity to the choice of the ratio of filter widths. This chapter will apply a modified dynamic SGS model to a channel flow at low Reynolds number and evaluate its performance.

### 8.2 The test runs

Four simulations of the same channel flow with the same geometry as in chapter 5 were performed. The characteristics of each simulation are summarized in table 8.1. $\Delta$ and $\hat{\Delta}$ in table 8.1 are the grid and "test" filter widths respectively. According to Deardorff [12], in the present study, they are defined as

$$
\Delta = (\Delta_x \Delta_y \Delta_z)^{1/3} \\
\hat{\Delta} = (\hat{\Delta}_x \hat{\Delta}_y \hat{\Delta}_z)^{1/3}.
$$

For comparison, A2 and A3 use the Germano-Lilly dynamic SGS model which is in the following form

$$
C(y, t) = \frac{<\hat{L}_{kl}M_{kl}>}{2 <M_{ij}M_{ij}>}.
$$

To investigate the sensitivity of the simulation results to the choice of the ratio of the "test" filter width to the grid filter width, in A2, we have $\hat{\Delta}_x = 2\Delta_x$, $\hat{\Delta}_y = \Delta_y$ and $\hat{\Delta}_z = 2\Delta_z$; therefore, $\hat{\Delta}/\Delta = 2^{2/3}$, whereas in A3, we have $\hat{\Delta}_x = 2\Delta_x$, $\hat{\Delta}_y = 2\Delta_y$ and $\hat{\Delta}_z = 2\Delta_z$; therefore, $\hat{\Delta}/\Delta = 2$. By contrast, B2 and B3 use the modified dynamic SGS model

$$
C(y, t) = \frac{1}{2} \left< \frac{\hat{L}_{kl}M_{kl}}{M_{ij}M_{ij}} \right>.
$$

The value of the ratio $\hat{\Delta}/\Delta$ in B2 is $\hat{\Delta}/\Delta = 2^{2/3}$, the same as that in A2, whereas $\hat{\Delta}/\Delta$ in B3 is $\hat{\Delta}/\Delta = 2$, the same as that in A3. The base model used by both the Germano-Lilly and the modified dynamic model is the Smagorinsky model. These four runs were all started from the instantaneous state of LESm at $t = 101.5$. The time step was $0.0003\delta/u_r$ for each simulation. Run A2 required 11.313 seconds per
time step on a Cray YMP8, run B2 1.5079 seconds, run A3 2.3217 seconds and run B3 2.3271 seconds. It can be seen that A2 is most costly in term of the computer time among all the four runs. The reason for this will be given later.

8.3 Results and discussion

The computed model coefficient $C$ for the runs A2, A3, B2 and B3 are presented in table 8.2. Note that the model coefficient obtained from the run A2 at some locations became negative or very large. Figure 8.1 illustrates the turbulence length scales computed from the results of the runs A2 (empty triangles), A3 (solid triangles), B2 (empty circles) and B3 (solid circles) in comparison with the length scale $l_{MK} = C_s [1 - \exp(-y^+/(\bar{A}^+))](\Delta_x \Delta_y \Delta_z)^{1/3}$ (dashed line) proposed by Moin and Kim (hereafter referred to as MK), the length scale $l_{PFM} = C_s [1 - \exp(-y^{1.3}/A^{1.3})]^{1/2}(\Delta_x \Delta_y \Delta_z)^{1/3}$ (continuous line) suggested by Piomelli, Ferziger and Moin (hereafter referred to as PFM) and the $y^{+3/2}$ curve (dotted line), which is the correct asymptotic behaviour of the turbulence length scale. (The negative value in the run A2 has been removed to plot the logarithmic profile).

There are several important findings. Firstly, the resulting length scales from the runs A2 and A3 using the Germano-Lilly dynamic SGS model (8.2) are close to the curve of $l_{MK}$ whilst the resulting length scales from the runs B2 and B3 using the modified dynamic SGS model (8.3) are close to the curve of $l_{PFM}$ which is best tuned to ensure a correct behaviour for the length scale of turbulence near the wall among the various existing empirical wall damping functions, and therefore believed to be most capable among the existing wall damping functions of giving a more accurate length scale profile in the near-wall region. Of great promise is that the profile of length scale obtained from the run B3 has shown the correct behaviour of the subgrid eddy viscosity in the region of a plane wall to an accuracy that exceeds
### 8.3 Results and discussion

Table 8.1: The characteristics of the simulations using dynamic SGS model

<table>
<thead>
<tr>
<th>symbol</th>
<th>A2</th>
<th>A3</th>
<th>B2</th>
<th>B3</th>
</tr>
</thead>
<tbody>
<tr>
<td>(L_x \times L_y \times L_z)</td>
<td>(4\pi \delta \times 2\delta \times 2\pi \delta)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(N_x \times N_y \times N_z)</td>
<td>96 (\times ) 64 (\times) 80</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(\Delta^+)</td>
<td>26(v/u_r)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(\Delta^+<em>{y</em>{\text{min}}})</td>
<td>0.635(v/u_r)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(\Delta^+<em>{y</em>{\text{max}}})</td>
<td>15.22(v/u_r)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>(\Delta^+_{t})</td>
<td>15.5(v/u_r)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>model type</td>
<td>(C(y, t) = \frac{\langle f_{\delta, M_{\delta}} \rangle}{2\langle M_{\delta}, M_{\delta} \rangle})</td>
<td>(C(y, t) = \frac{1}{2} \langle f_{\delta, M_{\delta}} \rangle)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>(\hat{\Delta}/\Delta)</td>
<td>(2^{2/3})</td>
<td>2</td>
<td>(2^{2/3})</td>
<td>2</td>
</tr>
</tbody>
</table>

This best-tuned wall damping function (PFM), and almost collapses with the curve of \(y^{+3/2}\), which is the expected behaviour of the length scale near the wall. Secondly, the run A2 using the Germano-Lilly dynamic SGS model (8.2) with \(\hat{\Delta}/\Delta = 2^{2/3}\) encountered numerical instability whereas the run A3 using the same SGS model but with \(\hat{\Delta}/\Delta = 2\) is stable. This fact therefore indicates that the Germano-Lilly dynamic SGS model (8.2) shows a greater sensitivity to the value of \(\hat{\Delta}/\Delta\) for low Reynolds number channel. This greater sensitivity to the ratio \(\hat{\Delta}/\Delta\) has also been encountered by Cabot and Moin [7] in their simulations of low Reynolds number channel flows using a dynamic SGS model in the exact same form of (8.2). On the contrary, as can be seen clearly from the length scale profiles obtained from B2 and B3 using the modified dynamic SGS model (8.3), the changes in the value of \(\hat{\Delta}/\Delta\) did not affect the results significantly. Thirdly, in the run A2, the encountered numerical instability was remedied by artificially setting the total viscosity \((\nu + \nu_t)\) to zero at those locations where negative total viscosity occurs, like Akselvoll and Moin [1]. As already presented earlier in this chapter, the run A2 is most costly, approximately
7.5 times more expensive than B2 and even 4.9 times more expensive than A3 and B3 which would be expected to be a little more costly than A2 and B2, since A3 and B3, with \( \Delta / \Delta = 2 \), involve more neighbouring grid points, and therefore more computation than A2 and B2 when the velocity field is filtered by the "test" filter whose width is larger in A3 and B3 than in A2 and B2. The reason for the high cost of A2 is its numerical instabilities, since most of the computation time in the run A2 has to be spent on checking the locations where the total viscosity is negative, and setting them to zero. So table 8.2 and figure 8.1 appear to confirm that the modified dynamic SGS model (8.3) removes a source of singularity more effectively.

All the above findings indicate that the advantage of the modified dynamic SGS model (8.3) of keeping more local information, which is believed to be important for the reliability and accuracy of the dynamic SGS model, seems to have brought about an improvement in SGS modelling.

Since figure 8.1 has clearly shown that the run B3 using the modified dynamic SGS model gives the more precise length scale profile than the run A3 using the Germano-Lilly dynamic SGS model, we could be confident that the low Reynolds number simulation results for channel flow from the run B3 should be better than those from the run A3. To investigate the ability of the modified dynamic SGS model (8.3) to improve the Smagorinsky model in the simulation of low Reynolds number channel flow, the results for the run B3 (solid dots) using the modified dynamic SGS model are shown in comparison with the results for the run LES0 (empty triangles) using the Smagorinsky model without low-Reynolds-number modification, and with the results for the run LES1 (solid triangles) using the same SGS model as LES0, but with low-Reynolds-number modification. For evaluating the performance of the dynamic SGS model, comparison with the simulation of KMM (dashed line) and the experimental data of NK (continuous line) is also made. Note that a wall damping function proposed by Moin and Kim (dashed line in figure 8.1) was used in
conjunction with the standard Smagorinsky model in the run LES0. From figure 8.1, it can be seen that, in the near-wall region, the length scale of B3 follows the curve of $y^{+3/2}$ more closely than does the curve of $l_{MK}$ and is smaller than $l_{MK}$; in the central region, the length scale of B3 is larger than $l_{MK}$. From the first fact, it appears that the results for the run B3 should show a higher level agreement with the results of KMM and NK than should the results for the run LES0 in the near-wall region. On the other hand, the second fact indicates that the subgrid eddy viscosity of B3 should be larger than that of LES0 in the central region since the eddy viscosity is proportional to the square of the length scale. Since large subgrid-scale eddy viscosity occurs in the central region, the eddy viscosity of the run B3 is larger than that of the run LES0 on the whole, although the length scale of the run B3 is smaller than that of the run LES0 in the near-wall region.

The mean streamwise velocity profile in figure 8.2 obtained from the run B3 shows much better agreement with those of KMM and NK than does the result for the run LES0. It is very impressive that, except for slightly lower values at the shoulder than those predicted by KMM and NK, B3 has not only resolved the linear sublayer excellently, but has also revealed the precise slope and the additive constant 5.5, the value used in KMM, in the log law region.

The streamwise fluctuation $u_{rms}$ (figure 8.3) is over-predicted in the run B3 in the region $10 < y^+ < 60$ where the length scale corresponding to B3 is larger than that corresponding to LES0. Since a larger length scale gives a larger SGS eddy viscosity proportional to the length scale, this over-predicted values of $u_{rms}$ in the run B3 is apparently caused by the increased SGS eddy viscosity. From figure 8.1, it can be seen that the length scale of B3 is much closer to the length scale $l_{PMF}$ in this region. If the data of the simulation using the standard Smagorinsky model in conjunction with a wall damping function of PFM were available, its agreement with the results of B3 would presumably be good. However, in the near-wall region,
the profile of $w_{rms}$ obtained from the run B3 is very promising: it shows a good agreement with those of KMM and NK, like LES0.

The pictures for the spanwise fluctuation $w_{rms}$ (figure 8.4) and cross-stream fluctuation $v_{rms}$ (figure 8.5) are similar. By contrast to the streamwise fluctuation $u_{rms}$, the increased SGS eddy viscosity results in under-predicted values for $v_{rms}$ and $w_{rms}$. Again, in the wall region, the agreement between B3 and KMM is as good as that between LES0 and KMM.

The principal component of the resolved Reynolds stress (figure 8.6) in the run B3 reveals a similar picture, with some under-predicted values, arising from the increased SGS eddy viscosity, in the region $10 < y^+ < 60$ but the level of agreement with KMM in the near-wall region is as high as that of LES0.

The results shown in figures 8.2 - 8.6 have confirmed that the modified dynamic SGS model has performed successfully in the LES of low Reynolds number channel flow without requiring any *ad hoc* model constants and damping functions at the walls. It is capable of predicting turbulence statistics with a quality at least as good as that predicted by the Smagorinsky model with a best prescribed model constant for the channel flow and an *ad hoc* damping function near the walls.

The results for the run LES1, which contained the same SGS model as the run LES0 but with low-Reynolds-number modification, were also plotted in figures 8.2 - 8.6 for comparison. These pictures show clearly that the results for the run B3 are similar to those for the run LES0, without revealing any sign of increases in the values of the velocity fluctuations as in the run LES1. Actually, in a nearly full simulation (for instance if the resolution is high enough or the Reynolds number is low enough), we should expect higher values of velocity fluctuations which, as demonstrated in figure 7.3 and 7.4, has made the spanwise and cross-stream fluctuations agree much better with KMM and NK, especially in the near-wall region where LES1
performed almost perfectly. This similarity between B3 and LES0 seems to indicate that the modified dynamic SGS model does not show a greater promise than the Germano-Lilly model to resolve the problem associated with a cutoff grid falling in the dissipation range, even though, as has already been presented earlier in this chapter, the modified model appears capable of removing the source of numerical instability and predicting the correct near-wall behaviour of SGS eddy viscosity in the low Reynolds number LES for channel flow besides having got rid of an *ad hoc* wall damping function and adjustment of the model coefficient as the Germano-Lilly model. The failure of the SGS models of dynamic type to reduce the SGS eddy viscosity accordingly when the resolved scales are small enough to contain those in the dissipation range may result from using the standard Smagorinsky model as a base model to derive the formulation of the dynamic SGS model. Therefore we could conclude that the choice of the base model is one of the factors affecting the quality of dynamic SGS modelling. Suggestions for improving the dynamic SGS modelling further will be given in chapter 10.
8.3 Results and discussion

Table 8.2: The dynamically computed model coefficient $C$

<table>
<thead>
<tr>
<th>$y^+$</th>
<th>A2</th>
<th>A3</th>
<th>B2</th>
<th>B3</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.2946</td>
<td>0.3894E-05</td>
<td>0.3698E-06</td>
<td>0.2052E-06</td>
<td>0.8525E-07</td>
</tr>
<tr>
<td>0.9293</td>
<td>0.3207E-04</td>
<td>0.5903E-05</td>
<td>0.3585E-05</td>
<td>0.1473E-05</td>
</tr>
<tr>
<td>1.6620</td>
<td>0.1151E-03</td>
<td>0.2763E-04</td>
<td>0.1746E-04</td>
<td>0.7203E-05</td>
</tr>
<tr>
<td>2.5070</td>
<td>0.2787E-03</td>
<td>0.7946E-04</td>
<td>0.5409E-04</td>
<td>0.2224E-04</td>
</tr>
<tr>
<td>3.4820</td>
<td>0.4938E-03</td>
<td>0.1759E-03</td>
<td>0.1381E-03</td>
<td>0.5500E-04</td>
</tr>
<tr>
<td>4.6050</td>
<td>0.6633E-03</td>
<td>0.3334E-03</td>
<td>0.3596E-03</td>
<td>0.1208E-03</td>
</tr>
<tr>
<td>5.8990</td>
<td>0.9161E-03</td>
<td>0.5740E-03</td>
<td>0.4192E-03</td>
<td>0.2493E-03</td>
</tr>
<tr>
<td>7.3860</td>
<td>0.1440E-02</td>
<td>0.9291E-03</td>
<td>0.9276E-03</td>
<td>0.4906E-03</td>
</tr>
<tr>
<td>9.9060</td>
<td>0.1817E-02</td>
<td>0.1438E-02</td>
<td>0.2304E-02</td>
<td>0.9151E-03</td>
</tr>
<tr>
<td>11.060</td>
<td>0.1721E-02</td>
<td>0.2169E-02</td>
<td>0.2491E-02</td>
<td>0.1582E-02</td>
</tr>
<tr>
<td>13.310</td>
<td>0.3230E-02</td>
<td>0.3152E-02</td>
<td>0.2459E-02</td>
<td>0.2551E-02</td>
</tr>
<tr>
<td>15.880</td>
<td>0.7171E-02</td>
<td>0.4552E-02</td>
<td>0.5463E-02</td>
<td>0.3710E-02</td>
</tr>
<tr>
<td>18.820</td>
<td>0.6817E-02</td>
<td>0.6675E-02</td>
<td>0.8316E-02</td>
<td>0.5139E-02</td>
</tr>
<tr>
<td>22.170</td>
<td>0.2114E-02</td>
<td>0.9625E-02</td>
<td>0.1001E-01</td>
<td>0.7018E-02</td>
</tr>
<tr>
<td>25.980</td>
<td>0.2647E-02</td>
<td>0.1272E-02</td>
<td>0.2088E-01</td>
<td>0.8891E-02</td>
</tr>
<tr>
<td>30.300</td>
<td>0.4376E-02</td>
<td>0.1523E-02</td>
<td>0.2324E-01</td>
<td>0.1066E-01</td>
</tr>
<tr>
<td>35.180</td>
<td>0.3426E-02</td>
<td>0.1697E-02</td>
<td>0.2496E-01</td>
<td>0.1197E-01</td>
</tr>
<tr>
<td>40.680</td>
<td>0.6948E-02</td>
<td>0.1742E-01</td>
<td>0.2310E-01</td>
<td>0.1287E-01</td>
</tr>
<tr>
<td>46.860</td>
<td>0.3370E-02</td>
<td>0.1792E-01</td>
<td>0.2671E-01</td>
<td>0.1379E-01</td>
</tr>
<tr>
<td>53.760</td>
<td>0.1545E-03</td>
<td>0.1977E-01</td>
<td>0.2552E-01</td>
<td>0.1481E-01</td>
</tr>
<tr>
<td>61.430</td>
<td>-0.3090E-02</td>
<td>0.1932E-01</td>
<td>0.2380E-01</td>
<td>0.1573E-01</td>
</tr>
<tr>
<td>69.920</td>
<td>0.1434E-01</td>
<td>0.1685E-01</td>
<td>0.2679E-01</td>
<td>0.1633E-01</td>
</tr>
<tr>
<td>79.250</td>
<td>0.3699E-01</td>
<td>0.1634E-01</td>
<td>0.2913E-01</td>
<td>0.1621E-01</td>
</tr>
</tbody>
</table>

continued on the next page
### 8.3 Results and discussion

<table>
<thead>
<tr>
<th>$y^+$</th>
<th>A2</th>
<th>A3</th>
<th>B2</th>
<th>B3</th>
</tr>
</thead>
<tbody>
<tr>
<td>89.440</td>
<td>0.6638E-01</td>
<td>0.1455E-01</td>
<td>0.2940E-01</td>
<td>0.1686E-01</td>
</tr>
<tr>
<td>100.50</td>
<td>0.4586E-01</td>
<td>0.1478E-01</td>
<td>0.2986E-01</td>
<td>0.1672E-01</td>
</tr>
<tr>
<td>112.40</td>
<td>0.8154E-02</td>
<td>0.1514E-01</td>
<td>0.3260E-01</td>
<td>0.1479E-01</td>
</tr>
<tr>
<td>125.10</td>
<td>0.1153E+00</td>
<td>0.1272E-01</td>
<td>0.3279E-01</td>
<td>0.1452E-01</td>
</tr>
<tr>
<td>138.50</td>
<td>0.1847E+00</td>
<td>0.1021E-01</td>
<td>0.2179E-01</td>
<td>0.1377E-01</td>
</tr>
<tr>
<td>152.60</td>
<td>0.9771E-01</td>
<td>0.8141E-02</td>
<td>0.2310E-01</td>
<td>0.1269E-01</td>
</tr>
<tr>
<td>167.10</td>
<td>0.1568E+00</td>
<td>0.7339E-02</td>
<td>0.2709E-01</td>
<td>0.1104E-01</td>
</tr>
<tr>
<td>182.10</td>
<td>-0.4436E-01</td>
<td>0.5931E-02</td>
<td>0.2254E-01</td>
<td>0.9982E-02</td>
</tr>
<tr>
<td>197.40</td>
<td>0.1592E+00</td>
<td>0.4402E-02</td>
<td>0.2925E-01</td>
<td>0.1105E-01</td>
</tr>
</tbody>
</table>

continued from the previous page
Figure 8.1: The subgrid length scale $C$ versus $y^+$. Solid dots, B3; empty dots, B2; solid triangles, A3; empty triangles, A2; continuous line, $0.1[1 - \exp(-y^+/A^+)]^{1/2}(\Delta_x\Delta_y\Delta_z)^{1/3}$; dashed line, $0.1[1 - \exp(-y^+/A^+)](\Delta_x\Delta_y\Delta_z)^{1/3}$; dotted line, a $y^{+3/2}$ slope.
8.3 Results and discussion

Figure 8.2: Mean streamwise velocity $u_{\text{mean}}$ versus $y^+$. Solid triangles, LES1; empty triangles, LES0; solid dot, B3; continuous line, NK; dashed line, KMM; dotted line, $u^+ = y^+$ and $u^+ = 2.5 \ln y^+ + 5.5$. 
Figure 8.3: Streamwise velocity fluctuation $u_{rms}$ versus $y^+$. Solid triangles, LES1; empty triangles, LES0; solid dots, B3; continuous line, NK; dashed line, KMM.
8.3 Results and discussion

Figure 8.4: Spanwise velocity fluctuation $w_{rms}$ versus $y^+$
Solid triangles, LES1; empty triangles, LES0; solid dots, B3; continuous line, NK; dashed line, KMM.
Figure 8.5: Cross-stream velocity fluctuation $v_{rms}$ versus $y^+$
Solid triangles, LES1; empty triangles, LES0; solid dots, B3; continuous line, NK; dashed line, KMM.
Figure 8.6: Principal Reynolds stress component $u'v$

Solid triangles, LES1; empty triangles, LES0; solid dots, B3; continuous line, NK;
dashed line, KMM.
9 Simulations Using Multiple Mesh Method

9.1 Problem description

LES is one of the most powerful computational tools available today for the calculation of turbulent flows, but this method is unfortunately more expensive computationally than other approaches in common use. A way out of this dilemma is to improve the economy of LES. As has already been presented in chapter 4, the modified multiple mesh method appears to hold a high promise of success in enhancing the effectiveness of LES through reducing the cost of computation. In this chapter, this multiple mesh method will be tested and the results be evaluated.

9.2 The test runs

9.2.1 Simulations testing the theoretical foundation of multiple mesh method

The basic idea of the modified multiple mesh method is to make use of the nested grids to generate a physically realistic velocity field on a fine mesh at the moment of the simulations moving from a coarse level to the fine level. This idea has been reflected in the equations (4.7) which reveal Reynolds stress algebraic relations between the fine and coarse meshes. These equations are to be used in the modified multiple mesh scheme to determine the independent degrees of freedom in the fine mesh field. Although the equations (4.7) can be derived, from a mathematical point of view, by filtering the Navier-Stokes equations twice, as in Germano[20], the physical meaning of the equations (4.7), on which the equations used in the modified multimesh method are based, lies in the fact that the statistics of the total turbulence stresses should remain invariant. The reason for this invariance of
the total turbulence stresses for the varying filter widths is that the increase of the resolved stresses will be approximately compensated by the decrease of the SGS stresses when the filter width \( \Delta \) decreases, for example when the resolution becomes higher, or the decrease of the resolved stresses will be approximately compensated by the increase of the SGS stresses when the filter width \( \Delta \) increases, for example when the resolution becomes lower. A test of the above argument will be carried out here before the modified multiple mesh method is actually applied to LES.

The simulated flow and geometry here are the same as those described in chapter 5. A test run starting from the initial velocity field taken from a coarse mesh \((24 \times 56 \times 32)\) simulation at \( t = 2.08 \delta / u_r \) was performed on a set of two nested meshes which is composed of a coarse mesh with \( 24 \times 56 \times 32 \) grid points and a fine mesh with \( 48 \times 56 \times 64 \) grid points. The characteristics of this simulation are summarized in table 9.1. This two-level multimesh simulation is carried out alternately on the coarse and fine meshes and therefore the grid scale changes correspondingly. The time step of the simulation on the coarse mesh is 0.0006 and that on the fine mesh is 0.0004. The periods of time spent at the coarse mesh and fine mesh levels are determined by a physical strategy [64], which requires the time spent on the fine mesh be at least sufficient to regenerate the eddies there. This amount of time has been approximately given by Leslie and Quarini [32],

\[
T_{fine} < \Delta^{2/3} L^{1/3} / u = (L/\Delta)^{-2/3} T
\]

(9.1)

where \( T = L/u \) is lifetime of the large eddies. Thus for the two-level multimesh simulation being performed here, in which \( L/\Delta = 56 \) at both the coarse mesh and fine mesh levels, \( T/T_{fine} \approx 15 \). Since the time step at the coarse mesh level is 1.5 times longer than that at the fine mesh level, if the simulation is performed for a period of order \( T \) on the coarse mesh, the number of time steps performed on the fine mesh should be at least \( 1/10 \) of that on the coarse mesh to satisfy the physical strategy. In the test run being performed, 2188 time steps \((1.3\delta/u_r)\) was actually
Table 9.1: The characteristics of the simulation testing multimesh method

<table>
<thead>
<tr>
<th>symbol</th>
<th>coarse level</th>
<th>fine level</th>
</tr>
</thead>
<tbody>
<tr>
<td>$L_x \times L_y \times L_z$</td>
<td>$4\pi \delta \times 2\delta \times 2\pi \delta$</td>
<td></td>
</tr>
<tr>
<td>$N_x \times N_y \times N_z$</td>
<td>$24 \times 56 \times 32$</td>
<td>$48 \times 56 \times 64$</td>
</tr>
<tr>
<td>$\Delta^+_x$</td>
<td>$108\nu/\nu_r$</td>
<td>$54\nu/\nu_r$</td>
</tr>
<tr>
<td>$\Delta^+<em>{v</em>{\min}}$</td>
<td>$0.68\nu/\nu_r$</td>
<td>$0.68\nu/\nu_r$</td>
</tr>
<tr>
<td>$\Delta^+<em>{v</em>{\max}}$</td>
<td>$17.5\nu/\nu_r$</td>
<td>$17.5\nu/\nu_r$</td>
</tr>
<tr>
<td>$\Delta^+_z$</td>
<td>$40\nu/\nu_r$</td>
<td>$20\nu/\nu_r$</td>
</tr>
<tr>
<td>time step</td>
<td>$0.0006$</td>
<td>$0.0004$</td>
</tr>
</tbody>
</table>

spent on the coarse mesh and 312 time steps ($0.125\delta/\nu_r$) on the fine mesh. Thus, the total integration time spent on the simulation progressing on the coarse mesh and then on fine mesh once (referred to as a cycle) was $1.425\delta/\nu_r$.

In figure 9.1, the time histories of total volume average $u^2$, $v^2$, $w^2$ and $k$ are presented. They show jumps corresponding to the instants at which the simulation moves from one mesh level to the other mesh level. Since the grid-scale motions increase as the grid is refined, the jumps, shown in the picture for the histories of the volume-averaged statistics of the grid-scale motions (figure 9.1), are upward at the moment of the simulation moving from the coarse mesh to the fine mesh, whereas the jumps are downwards at the moment of the simulation moving oppositely. As we should expect, the resolved shear stress $uv$ (figure 9.2a) reveals a similar picture. Like figure 9.1, the upward jumps in this picture occur at the starting instant of the multimesh simulation at the fine level and the downward jumps at the starting instant of the multimesh simulation at the coarse level. By contrast, the picture (figure 9.2b) for the SGS shear stress shows contrary jumps, i.e. the upward jumps in this picture correspond to the instant of the multimesh simulation moving from
9.2 The test runs

the fine level to the coarse level whereas the downward jumps correspond to the
instant of the opposite move. The causes of these upward and downward jumps
in figure 9.2b are the increase of the SGS shear stress due to the increase of the
grid scale and the decrease of the SGS shear stress due to the decrease of the grid
scale respectively. However, as has already been pointed out earlier in this section,
the curve of the total turbulence shear stress should be smooth. This argument was
confirmed by figure 9.2c, in which the time history of total mean shear stress $uv + u'v'$
averaged over the computational region was plotted. This picture (figure 9.2c) shows
a curve becoming smooth after several cycles, which indicates that the upward jumps
of the resolved shear stress $uv$ shown in figure 9.2a were roughly compensated by
the downward jumps of the SGS shear stress $u'v'$ shown in figure 9.2b and the
downward jumps of $uv$ were roughly compensated by the upward jumps of $u'v'$.
Thus the equations (4.7), used to derive equations determining the independent
degrees of freedom in the fine mesh field, can be considered as reliable.

9.2.2 Multiple mesh simulations

For comparison, two multimesh simulations were performed: one (denoted by
MMS1) is to use the modified multimesh method to generate a physically realistic
velocity field on a fine mesh by following the procedures described in chapter 4,
whereas the other one (denoted by MMS0) is to use the following interpolation
scheme to generate a velocity field on the fine mesh when the simulation moves
from a coarse mesh to the fine mesh,
9.3 Results and discussion

\[ u_1^1 = U_a, \quad u_1^2 = \frac{1}{2}(U_a + U_b) \]
\[ v_1^1 = v_2^1 = V_a \]
\[ w_1^1 = w_2^1 = W_a. \]  \hspace{1cm} (9.2)

where \( U_\kappa, V_\kappa, \) and \( W_\kappa (\kappa = a, b) \) are the velocity components on the coarse mesh and \( u_i^j, v_j^j \) and \( w_i^j (i, j = 1, 2) \) are the velocity components on the fine mesh (see figure 4.3a). It is not difficult to show that the velocity field on the fine mesh generated by the above interpolation scheme in MMS0 obeys continuity there. In MMS1, continuity is satisfied by enforcing continuity on the fine mesh when the velocity component \( u \) on the interleaving plane of each finite volume cell is solved. For the multimesh simulations being performed here, the simulated flow and geometry are the same as those described in chapter 5. The characteristics of the simulations are summarized in table 9.2. Both of the simulations were started from an instantaneous state of LESc on a mesh, with \( 48 \times 40 \times 64 \) grid points, at \( t = 80 \), immediately followed by the interpolation of the velocity field onto the next fine mesh with grid points \( 96 \times 40 \times 64 \). The time step was \( 0.00026/u_r \).

9.3 Results and discussion

The instantaneous results obtained from MMS1 (empty circles), MMS0 (crosses) and LESc (dotted line) at the same instant are shown in comparison with the database of KMM (dashed line) and the experimental data of NK (continuous line).

The picture for the mean velocity (figure 9.3) shows the collapse of the mean velocity profiles corresponding to MMS0, MMS1 and LESc. The reason for this lies in the algorithms of \( u \) used in the runs of MMS0 and MMS1. For MMS0, as presented in the equation (9.2), the fine mesh velocity component \( u \) on the surface of each finite volume cell shown in figure 4.3a is equal to the coarse mesh velocity.
Table 9.2: The characteristics of the multiple mesh simulation

<table>
<thead>
<tr>
<th>level</th>
<th>symbol</th>
<th>MMS1</th>
<th>MMS0</th>
</tr>
</thead>
<tbody>
<tr>
<td>coarse</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>$N_x \times N_y \times N_z$</td>
<td>$48 \times 64 \times 40$</td>
<td>$48 \times 64 \times 40$</td>
</tr>
<tr>
<td></td>
<td>$\Delta_t^+$</td>
<td>$52 \nu/\nu_r$</td>
<td>$52 \nu/\nu_r$</td>
</tr>
<tr>
<td></td>
<td>$\Delta_{y_{\min}}^+$</td>
<td>$0.635 \nu/\nu_r$</td>
<td>$0.635 \nu/\nu_r$</td>
</tr>
<tr>
<td></td>
<td>$\Delta_{y_{\max}}^+$</td>
<td>$15.22 \nu/\nu_r$</td>
<td>$15.22 \nu/\nu_r$</td>
</tr>
<tr>
<td></td>
<td>$\Delta_t^+$</td>
<td>$31 \nu/\nu_r$</td>
<td>$31 \nu/\nu_r$</td>
</tr>
<tr>
<td>fine</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>$N_x \times N_y \times N_z$</td>
<td>$96 \times 64 \times 40$</td>
<td>$96 \times 64 \times 40$</td>
</tr>
<tr>
<td></td>
<td>$\Delta_t^+$</td>
<td>$26 \nu/\nu_r$</td>
<td>$26 \nu/\nu_r$</td>
</tr>
<tr>
<td></td>
<td>$\Delta_{y_{\min}}^+$</td>
<td>$0.635 \nu/\nu_r$</td>
<td>$0.635 \nu/\nu_r$</td>
</tr>
<tr>
<td></td>
<td>$\Delta_{y_{\max}}^+$</td>
<td>$15.22 \nu/\nu_r$</td>
<td>$15.22 \nu/\nu_r$</td>
</tr>
<tr>
<td></td>
<td>$\Delta_t^+$</td>
<td>$31 \nu/\nu_r$</td>
<td>$31 \nu/\nu_r$</td>
</tr>
<tr>
<td></td>
<td>time step</td>
<td>$0.0002$</td>
<td>$0.0002$</td>
</tr>
<tr>
<td></td>
<td>interpolation</td>
<td>(4.4) and (4.9)</td>
<td>(9.2)</td>
</tr>
</tbody>
</table>

component $U$ there. On the other hand, the velocity component $u$ on the interleaving plane of each finite volume cell is a weighted average of the velocity components $U$ of its two neighbours on the surfaces of each finite volume (see equation (9.2)). These two neighbours have the same horizontal coordinates $(x, z)$. It is therefore not difficult to show that there is no difference between the streamwise mean velocities, averaged over a plane parallel to the wall, of MMS0 and LESc. For MMS1, on the surface of each finite volume cell, the velocity component $u$ on the fine mesh is still equal to the coarse mesh velocity component $U$ there. Although the $u$ on the interleaving plane of each finite volume cell is computed by the continuity condition which involves the velocity components $v$ and $w$, resolved through the Reynolds stress algebraic relations between the coarse and fine meshes (4.9), it is
not unexpected that its mean velocity does not change since it is an average of both the \( u \) on the grid points of the surface of each finite volume cell, which is equal to the coarse mesh velocity component \( U \) there, and \( u \) on the grid points of the interleaving plane of each finite volume cell, which is solved by a linear equation (continuity of the fine mesh) of \( u \) and \( w \), over a plane parallel to the wall.

The streamwise fluctuation \( u_{rms} \) (figure 9.4) reveals a similar picture, with slightly lower values of \( u_{rms} \) corresponding to MMS1 and MMS0 compared with those corresponding to LESc after the peak value is reached.

The picture for the cross-stream fluctuation \( v_{rms} \) (figure 9.5) shows an improvement brought about by the modified multimesh method on the \( v_{rms} \) profile. By contrast, the collapse of two curves corresponding to MMS0 and LESc is found.

The spanwise fluctuation \( w_{rms} \) (figure 9.6) reveals a similar promising picture, with an evident improvement on the \( w_{rms} \) profile. The \( w_{rms} \) profile of MMS0 still collapse with that of LESc. In the central region, higher values of \( w_{rms} \) corresponding to MMS1 are found. However, there is no indication that this deficiency in the result of MMS1 arise from the modified multiple mesh method, since the profiles of \( w_{rms} \) corresponding to MMS0 and LESc have revealed the same trends in the central region. The reason for this deficiency appears to be that the results shown in figure 9.6 are produced from MMS1, MMS0 and LESc instantaneously, not by time averaging.

There is no doubt that the increase in numerical resolution will improve the quality of turbulence statistics and therefore we should expect that as soon as a simulation moves from a coarse mesh to a fine mesh, some improvement in the turbulence statistics would be found if there exists correlation between the velocity fields of the coarse and fine meshes. From the above results, it can be seen that there is no difference between the results of LESc, which was performed wholly on
the coarse mesh, and MMS0, which was interpolated onto the fine mesh from the coarse mesh through a method of interpolation described in equation (9.2). The reason for this is that the run MMS0 did not carry any information of grid scales between the coarse and fine meshes but only the information of the coarse mesh when the simulation is switched to the fine mesh. So the collapse of the profiles corresponding to MMS0 and LESc is not unexpected. On the contrary, for MMS1, the velocities on the fine mesh are resolved by using the Reynolds stress algebraic relations between the coarse and fine meshes. Doing so will create a more physically realistic fine mesh velocity field. This argument has been confirmed by a higher level agreement of MMS1 than that of MMS0 and LESc with KMM and NK. In brief, for MMS0, the simulation on the fine mesh, started from a velocity field of the coarse mesh, which is not correlated with the fine mesh in the slightest degree, has to spend a longer period of time to generate the eddies resolved on the fine mesh. By contrast, for MMS1, the simulation on the fine mesh, started from the same velocity field of the coarse mesh, but from which a physically realistic fine mesh velocity field is generated by making use of the information of the grid scales between the coarse and fine meshes, only need to correct the errors, brought by the approximations used to derive the equations of the velocities on the fine mesh, in the initial fine mesh scale structures. Therefore we should expect that MMS1 will spend less time than MMS0 to achieve a realisation of the small scale field.
Figure 9.1: The history of the simulation.

Bottom to top: total volume average $u^2$, $w^2$, $k/5$, $u'^2/5$. 
9.3 Results and discussion

Figure 9.2: (a) The history of mean resolved shear stress $uv$.

Figure 9.2: (b) The history of mean SGS shear stress $u'v'$. 

\[ \text{resolved stress } uv \]
\[ \begin{align*}
0.5 & \\
0.4 & \\
0.3 & \\
0.2 & \\
0.1 & \\
0.0 & \\
0.05 & \\
0.04 & \\
0.03 & \\
0.02 & \\
0.01 & \\
0.00 & \\
\end{align*} \]

\[ \text{t} \]

\[ \begin{align*}
0 & \\
1 & \\
2 & \\
3 & \\
4 & \\
5 & \\
6 & \\
7 & \\
8 & \\
9 & \\
10 & \\
11 & \\
12 & \\
\end{align*} \]
9.3 Results and discussion

Figure 9.2: The history of mean shear stresses, $uv$, $u'v'$ and $uv + u'v'$. 

(c) The history of mean total shear stress $uv + u'v'$. 

Figure 9.2: The history of mean shear stresses, $uv$, $u'v'$ and $uv + u'v'$. 
9.3 Results and discussion

Figure 9.3: Mean streamwise velocity $u_{\text{mean}}$ versus $y^+$.

Empty circles, MMSI; crosses, MMS0; dotted line, LESc; continuous line, NK; dashed line, KMM.
9.3 Results and discussion

![Figure 9.4: Streamwise velocity fluctuation $u_{rms}$ versus $y^+$.](image)

- Empty circles, MMS1; crosses, MMS0; dotted line, LESc; continuous line, NK;
- dashed line, KMM.

(a) Linear $y$ axis.

(b) Logarithmic $y$ axis.
9.3 Results and discussion

Figure 9.5: Cross-stream velocity fluctuation $v_{rms}$ versus $y^+$. Empty circles, MMS1; crosses, MMS0; dotted line, LESc; continuous line, NK; dashed line, KMM.
9.3 Results and discussion

Figure 9.6: Spanwise velocity fluctuation $w_{rms}$ versus $y^+$.

Empty circles, MMS1; crosses, MMS0; dotted line, LESc; continuous line, NK; dashed line, KMM.
10 Concluding Remarks

10.1 A review of the thesis

Application of large eddy simulations to flows of engineering interest has been limited by the amount of computational effort it requires. To make this approach efficient enough for practical engineering related problems, we, in the present study, mainly concentrated on developing a superior SGS model and improving the economy of LES, which, we believe, should receive equal attention. The flow of interest in the present study is low Reynolds channel flow. As is well known, the LES method was firstly developed for tackling high Reynolds number turbulent flows and has matured to a certain extent. However, the LES method cannot be extended to the low-Reynolds-number regime without any modification because at low Reynolds number some new problems arise. In the present study, we also explored ways to surmount the difficulties.

10.1.1 Preliminary study

We first carried out a comparative study of the various existing SGS models. We found that the Smagorinsky model and the dynamic SGS model are suitable for the aims to be achieved in the present study. For the channel flow simulated here, the main drawbacks of the Smagorinsky model were exposed. However, the drawbacks associated with the Smagorinsky model proved to be quite amenable to theoretical treatments in channel flow. The results of simulations containing a modified Smagorinsky model can be used as a comparison to assess the ability of a new SGS model to get rid of any ad hoc adjustment according to the flows to which it is applied. Such a new SGS model is known as a dynamic SGS model, initially proposed by Germano et al. [20], whose most promising feature is that the
model coefficient is dynamically computed as the calculation progresses. Improving it further is one of the main tasks of the present study. In addition, a detailed derivation of the second-order velocity structure function SGS model has been made to correct an error found in that model.

After the above fundamental study, several comparative simulations have been carried out in this study. Firstly, simulations with varying model constants were performed, and an optimum value of the model constant was confirmed to be 0.1 for the channel flow simulated here.

Secondly, simulations with varying numerical resolution were performed. We found that extremely fine normal resolution near the walls, as in the simulation of Kim, Moin and Moser [29], is not necessary to obtain parabolic behaviour of the cross-stream fluctuation $v_{rms}$. Instead the present study reduced the normal resolution near the wall by 6.4 times and has produced near-wall behaviour of the mean streamwise velocity, velocity fluctuations and principal component of the resolved Reynolds stresses (lower-order statistics) as correct as those predicted by Kim, Moin and Moser. However, we also found that the normal resolution is very important to the quality of the skewness and flatness (higher-order statistics) in the near-wall region. In addition, simulations with varying numerical resolution reveal that the refinement of resolution in the direction normal to the wall improves all the turbulence statistics, both higher- and lower-order statistics, over the whole channel, while the refinement of resolution in the streamwise and spanwise directions improves lower-order turbulence statistics over the whole channel, but only improves the profiles of higher-order statistics in the central region of the channel. Among the simulations with varying numerical resolution, the simulation at the highest level is a nearly full simulation. It gave a vivid demonstration of how LES is expensive even at low Reynolds number.
10.1 A review of the thesis

Thirdly, simulations with and without low-Reynolds-number modification were performed. The results have shown that the modification enable the Smagorinsky model to reduce the eddy viscosity accordingly when the cutoff scale falls in the dissipation region, where the energy transfer through the cut should be less than that through a cut in the inertial range. The results of this comparative study reveal that this modification improves the turbulence statistics of low Reynolds number LES for channel flow on the whole. Particularly impressive are the profiles of the spanwise and cross-stream velocity fluctuations in the near-wall region. However, in the author's opinion, the ability of this low-Reynolds-number modification needs further investigation in flow configurations much different from that simulated here.

10.1.2 Improvement in subgrid-scale modelling

On the basis of the above preliminary investigation, an effort was made to improve SGS modelling. The dynamic SGS model is one of the recently developed models of high promise. Moreover, the Germano-Lilly dynamic SGS model has proved successful in large-eddy simulations without the need to adjust the model coefficient or to invoke ad hoc wall damping functions. However, possibly higher sensitivity of the low Reynolds number LES results for channel flow to the choice of $\Delta_0/\Delta$ indicates that there is still room for improvement to extend this model to low Reynolds number flows. In the present study, firstly we take the advantage of the existing matured theoretical concepts for channel flow, such as wall damping function and optimum model coefficient, to assess the performance of a modified SGS model proposed in chapter 3 through a comparison of length scales computed respectively by this modified dynamic SGS model, the Germano-Lilly dynamic SGS model and two empirical wall damping functions (one is that of Moin and Kim, the other is that of Piomelli, Ferziger and Moin) in conjunction with an optimum model coefficient (0.1 for channel flow), which have been successfully used in the simulations of channel
flows. Two values of the ratio of filter widths, $\hat{\Delta}/\Delta = 2^{3/2}$ and 2, were set for each of the dynamic models. The results have confirmed that the modified dynamic SGS model is more successful than the Germano-Lilly model in low Reynolds number channel flow. The success of the modified model in low Reynolds number channel flow is reflected from two findings. One is that the resulting length scale from the LES using the modified model has shown an almost perfect behaviour in the region of a plane wall without any theoretical treatment. In contrast, the performance of the Germano-Lilly model is just marginal. The other is that, for the modified model, the choice of $\hat{\Delta}/\Delta$ did not affect the predicted length scale significantly, whereas, for the Germano-Lilly model, the choice of $\hat{\Delta}/\Delta$ has a considerable impact on the predicted length scale. We then made another comparative study of a LES using the modified dynamic SGS model with $\hat{\Delta}/\Delta = 2$ and a LES using the Smagorinsky model with an optimum model constant (0.1 for channel flow) and a wall damping function of Kim and Moin [48]. The agreement of the results of these two simulations with the database of a direct simulation of Kim, Moin and Moser is of the same level, which indicated that the dynamic model can get rid of any tuning and theoretical wall treatments.

10.1.3 Improvement of the economy of the LES method

Since the LES method is a very valuable tool of computation of turbulent flow but unfortunately still remains one of the most costly methods commonly used, the present study was also devoted to improving the economy of the LES method. One way of achieving this is a modified multiple mesh method proposed in chapter 4. A comparative study of a modified multimesh simulation and an unmodified multimesh simulation has confirmed that a physically realistic fine-mesh flow field was created from the coarse-mesh field in the modified multimesh simulation. In addition, we found that the turbulence statistics obtained from the modified multimesh
10.2 Suggestions for future work

10.2.1 Subgrid-scale modelling

Although the result of the simulation using the modified dynamic model has shown great promise, there still exists a problem related to low Reynolds number. The present low Reynolds number LES results for channel flow (table 8.2) show that the magnitude of eddy viscosity computed dynamically is larger at some locations in the central region than that obtained from the high Reynolds number simulation using the dynamic SGS model [47], which is in good agreement with the commonly used values for channel flow, that range between 0.08 and 0.1. The largest magnitude of eddy viscosity from the run A3 using the Germano-Lilly model is 41% larger than
0.1. In the run B3 using the modified dynamic model, the largest magnitude of eddy viscosity remains larger than 0.1 by 27%, though the deviation from the commonly used value has been alleviated largely, and the region where the larger magnitudes of eddy viscosity occur is smaller. The disagreement may arise from the assumption (referred to hereafter as the Germano assumption) used by Germano to derive the dynamic model, namely that the same formulation (the standard Smagorinsky model) can be used to parameterize SGS stresses at both grid and “test” scales, and therefore the grid and “test” scale cuts are limited to an inertial range. Unfortunately, this assumption is inappropriate for low Reynolds number LES. In the low Reynolds number LES or LES with high enough resolution, the grid and “test” scale cutoffs fall in the dissipation region, in which the energy spectrum is much steeper than that of inertial subrange. In that case, the Smagorinsky formulation overestimates the energy transfer across the cut, which in turn results in overprediction of the magnitude of eddy viscosity. Having considered the fact that the Smagorinsky model with low-Reynolds-number modification performed rather successfully in low Reynolds number LES, we could use this model as a base model to derive a dynamic model formulation extending to the low Reynolds number situation. As presented in chapter 2, the final form of the low Reynolds number eddy viscosity can be written as

\[ \nu_t = \nu_s + \beta \nu [1 - \exp(-\nu_s/(\beta \nu))] \] \hspace{1cm} (10.1)

Note that the meaning of \( \nu_s \) is slightly different to that defined in chapter 2. In the above formulation, \( \nu_s \) is the Smagorinsky eddy viscosity but with a model coefficient dynamically calculated by using the Germano-Lilly model, \( \nu \) the molecular eddy viscosity and the exponential term represents the reduction in the energy transfer in the dissipation region.

Substituting the Smagorinsky formulation into (10.1) yields

\[ \nu_t(C) = C \Delta^2 \tilde{S} - \beta \nu [1 - \exp(-C \Delta^2 \tilde{S} / (\beta \nu))] \] \hspace{1cm} (10.2)
where \( C \) is equivalent to \( C^2_s \) but computed dynamically as the simulation progresses. By defining \( C_m \) to be the eddy viscosity coefficient dynamically calculated by a dissipation-range dynamic SGS model to be constructed, we have

\[
C_m = C + \delta_c \quad (10.3)
\]

where \( \delta_c \) is a correction to the effect of Reynolds number. Since the small-scale turbulent structures are more universal than the large, \( \delta_c \) is not expected to change greatly from flow to flow, and hence we may assume that \( \delta_c \) is a small value.

Upon expanding \( \nu_t(C_m) \) as Taylor series in the neighbourhood of \( C \), the eddy viscosity reads

\[
\nu_t(C_m) = \nu_t(C) + \nu_t'(C) \delta_c + \frac{1}{2} \nu_t''(C) \delta_c^2 + \cdots \quad (10.4)
\]

If \( \delta_c \) is small enough, the terms of higher-order \( \delta_c \) are unimportant; hence \( \nu_t(C_m) \) can be approximated as

\[
\nu_t(C_m) \approx \nu_t(C) + \nu_t'(C) \delta_c
\]

Now the SGS stresses at the grid and "test" scale levels can be obtained by being related to the \( \mathcal{S}_{ij} \) and \( \hat{\mathcal{S}}_{ij} \) respectively through the eddy viscosity (10.5)

\[
\tau_{ij} - \frac{1}{3} \delta_{ij} \tau_{kk} = 2 \left\{ C \Delta^2 \mathcal{S} - \beta \nu \left[ 1 - \exp(-C \Delta^2 \mathcal{S}/(\beta \nu)) \right] \right. \\
\left. + \Delta^2 \mathcal{S} \left[ 1 - \exp(-C \Delta^2 \mathcal{S}/(\beta \nu)) \right] \delta_c \right\} \mathcal{S}_{ij} \quad (10.6)
\]

\[
T_{ij} - \frac{1}{3} \delta_{ij} T_{kk} = 2 \left\{ C \Delta^2 \hat{\mathcal{S}} - \beta \nu \left[ 1 - \exp(-C \Delta^2 \hat{\mathcal{S}}/(\beta \nu)) \right] \right. \\
\left. + \Delta^2 \hat{\mathcal{S}} \left[ 1 - \exp(-C \Delta^2 \hat{\mathcal{S}}/(\beta \nu)) \right] \delta_c \right\} \hat{\mathcal{S}}_{ij} \quad (10.7)
\]

Then the equation \( \tilde{L}_{ij} = T_{ij} - \tau_{ij} \) can be rewritten as

\[
\tilde{L}_{ij} - \frac{1}{3} \delta_{ij} L_{kk} = 2M_{ij} + 2N_{ij} \delta_c \quad (10.8)
\]
where
\[ \tilde{L}_{ij} = \tilde{u}_i \tilde{u}_j - \tilde{u}_i \tilde{u}_j \] (10.9)
\[ \mathcal{M}_{ij} = C \Delta^2 \tilde{S} \tilde{S}_{ij} - \beta \nu \left[ 1 - \exp\left(-C \Delta^2 \tilde{S}/(\beta \nu) \right) \right] \tilde{S}_{ij} \]
\[ - C \Delta^2 \tilde{S} \tilde{S}_{ij} + \beta \nu \left[ 1 - \exp\left(-C \Delta^2 \tilde{S}/(\beta \nu) \right) \right] \tilde{S}_{ij} \] (10.10)
\[ \mathcal{N}_{ij} = \Delta^2 \tilde{S} \left[ 1 - \exp\left(-C \Delta^2 \tilde{S}/(\beta \nu) \right) \right] \tilde{S}_{ij} - \Delta^2 \tilde{S} \left[ 1 - \exp\left(-C \Delta^2 \tilde{S}/(\beta \nu) \right) \right] \tilde{S}_{ij} \] (10.11)

Define \( Q \) to be the square of the error in (10.8), i.e.
\[ Q = (\tilde{L}_{ij} - \frac{1}{3} \delta_{ij} \tilde{L}_{kk} - 2 \mathcal{M}_{ij} - 2 \mathcal{N}_{ij} \delta_c)^2 \] (10.12)

By setting \( \partial Q / \partial \delta_c = 0 \), \( \delta_c \) can be evaluated as
\[ \delta_c = \frac{\tilde{L}_{ij} \mathcal{N}_{ij} - 2 \mathcal{M}_{ij} \mathcal{N}_{ij}}{2 \mathcal{N}_{ij} \mathcal{N}_{ij}} \] (10.13)

This \( \delta_c \) represents the minimum of \( Q \), since \( \frac{\partial^2 Q}{\partial \delta_c^2} = 4 \mathcal{N}_{ij} \mathcal{N}_{ij} > 0 \). Note that \( \frac{1}{3} \delta_{ij} \tilde{L}_{kk} \) does not appear in (10.13) because \( \mathcal{S}_{ij} = 0 \) in an incompressible flow. Now by substituting (10.13) into (10.3), we obtain
\[ C_m = C + \frac{\tilde{L}_{ij} \mathcal{N}_{ij} - 2 \mathcal{M}_{ij} \mathcal{N}_{ij}}{2 \mathcal{N}_{ij} \mathcal{N}_{ij}} \] (10.14)

When the Reynolds number is high enough, i.e. the case of the classical LES, it is expected that \( \nu_s \) is much larger than \( \nu \). Analytically, taking the limit \( \nu_s / \nu \to \infty \), we have
\[ \mathcal{M}_{ij} = C \Delta^2 \tilde{S} \tilde{S}_{ij} - \beta \nu \tilde{S}_{ij} - C \Delta^2 \tilde{S} \tilde{S}_{ij} + \beta \nu \tilde{S}_{ij} = C \left( \Delta^2 \tilde{S} \tilde{S}_{ij} - \Delta^2 \tilde{S} \tilde{S}_{ij} \right) \] (10.15)
\[ \mathcal{N}_{ij} = \Delta^2 \tilde{S} \tilde{S}_{ij} - \Delta^2 \tilde{S} \tilde{S}_{ij} \] (10.16)

Hence
\[ C_m = \frac{\tilde{L}_{ij} \left( \Delta^2 \tilde{S} \tilde{S}_{ij} - \Delta^2 \tilde{S} \tilde{S}_{ij} \right)}{2 \left( \Delta^2 \tilde{S} \tilde{S}_{ij} - \Delta^2 \tilde{S} \tilde{S}_{ij} \right) \left( \Delta^2 \tilde{S} \tilde{S}_{ij} - \Delta^2 \tilde{S} \tilde{S}_{ij} \right)} \] (10.17)
10.2 Suggestions for future work

This is the Germano-Lilly model. Thus it can be seen that the Germano-Lilly formulation is the high Reynolds number case of (10.14). Since the Smagorinsky model with low-Reynolds-number modification can improve the turbulence statistics of low Reynolds number LES for channel flow, the dissipation-range dynamic model on the basis of this model may be suitable for low Reynolds number flows.

There is also some further work that could be done on the modified dynamic model created in the present study. Firstly, the local space average instead of the plane average used to obtain the formulation of the modified model should be attempted. Secondly, the modified model should be applied to more complex flows to assess its performance. Also as we have noticed, the conclusion that the ratio $\Delta/\Delta = 2$ is optimum was made from the results of channel flow simulations. Researchers normally apply this value to very different flow configurations. In the author's opinion, further study is required to verify this.

10.2.2 Multiple mesh method

As has already been mentioned earlier in this chapter, if a physically realistic fine-mesh flow field can be created, without need to invoke the frozen residual field, from the simulated flow on a coarse mesh which is twice as coarse as the fine mesh in each direction, the multimesh method would be more appealing. In order to develop the multimesh method in that way, its implementation could be designed to refine the coarse mesh in all the three directions successively. Before the next refinement in another direction is made, a period of simulation time may be required to correct the errors caused by the approximations made in the process of the derivation of the modified multimesh formulation. Of course, the length of this period should be a compromise between the quality of the resulting velocity field on the mesh after refinement and the economy of the multimesh simulation. This needs further
10.2 Suggestions for future work

According to this strategy, firstly, the number of grid points of a coarse mesh, from which the multimesh simulation is started, is doubled in one direction to obtain a finer mesh. The velocity field on this finer mesh is produced by the modified multimesh formulation described in chapter 4. After a period of time is spent on this mesh, the velocity field there is then interpolated by the same procedure onto a next finer mesh whose number of grid points in another direction is twice as many as that of the previous mesh. Then a period of time is spent on it. Finally, the number of grid points of the previous mesh is doubled in the direction left unrefined and the velocity field is created in the same way.

On the basis of the realisation of the above idea, we could enhance the promise of the multimesh method by using Voke's multimesh concept [68]. His multimesh simulation was performed alternatively on the coarse and fine meshes. On the former, the simulation was integrated forward for a new flow realisation at a very low cost in terms of computer time, while on the latter, the errors introduced by the frozen residual velocity field were corrected and the turbulence statistics from a flow realisation on that mesh were gathered. We could inject the velocity flow field generated by the successive interpolation process described above onto the initial coarse mesh. After a period of time on the coarse mesh, during which a new realisation of the motions of large eddies is reached, then the flow field is interpolated successively back to the finest mesh. The time spent on the coarse and fine meshes in Voke’s method was determined by an equal cost strategy, which requires that the same amount of CPU time should be expended on each of the levels. This strategy may still be valid here. Although some period of time may have to be spent between the two successive refinements, the physically realistic velocity field regenerated on the finest mesh will shorten the simulation time there because there is no need to recover the destroyed correlation between the coarse and fine meshes, as in the multimesh simulation using the frozen residual field.
References


