Large Eddy Simulations of complex turbulent flows

Mayank Tyagi
Louisiana State University and Agricultural and Mechanical College

Follow this and additional works at: https://digitalcommons.lsu.edu/gradschool_dissertations

Part of the Mechanical Engineering Commons

Recommended Citation
https://digitalcommons.lsu.edu/gradschool_dissertations/279

This Dissertation is brought to you for free and open access by the Graduate School at LSU Digital Commons. It has been accepted for inclusion in LSU Doctoral Dissertations by an authorized graduate school editor of LSU Digital Commons. For more information, please contact gradetd@lsu.edu.
LARGE EDDY SIMULATIONS OF COMPLEX TURBULENT FLOWS

A Dissertation

Submitted to the Graduate Faculty of the
Louisiana State University and
Agricultural and Mechanical College
In partial fulfillment of the requirements of
Doctor of Philosophy

In

The Department of Mechanical Engineering

by

Mayank Tyagi
B.S. Mechanical Engineering
Indian Institute of Technology, Kanpur, India, 1995
May 2003
To My Parents
Acknowledgements

I would like to thank my advisor Dr. S. Acharya for providing financial support and stimulating discussions over past several years. I would like to thank Dr. J. R. Dorroh for serving as the minor professor in mathematics area and teaching me mathematical details of the Navier-Stokes equations very patiently, Dr. M. Inoue for serving as the minor professor in oceanography area and Dr. W. Wiseman for discussions on films of fluid stability experiments. All my friends have been very encouraging and patient with me. They have helped me in so many ways than they would ever know.
# Table of Contents

Acknowledgements ........................................................................................................ iv

Abstract ......................................................................................................................... vi

Chapter
1 Physics, Mathematics and Simulation of Fluid Turbulence ...................... 1
2 Large Eddy Simulations of Complex Turbulent Flows ................. 11
3 Large Eddy Simulations with Immersed Boundary Method:  
  Solution Procedure and Validation Studies ................................. 35
4 Mathematical Tools for the Analysis of Turbulent Fields .......... 56
5 Jets in Crossflow and Film Cooling of Gas Turbine Blades .......... 61
6 Internal Cooling of Gas Turbine Blades ............................................... 184
7 Large Eddy Simulations of Trapped-Vortex Combustor ............. 216
8 Concluding Remarks and Future Directions .......................... 250

References ................................................................................................................... 255

Appendix
I Relationship Between Reynolds Stress Tensor and SGS Stress  
  Tensor ....................................................................................................................... 283
II The Eddy Viscosity Concept and Smagorinsky’s Model ................ 286
III The Dynamic One-Equation Subgrid Scale Model .................. 288
IV The Differential Stress Model .............................................................. 290
V Direct Solver for Poisson Equation in Complex Geometries .... 291
VI Zonal/Local Refinement of Cartesian Mesh ......................... 295

Vita .............................................................................................................................. 299
Abstract

In this dissertation a solution methodology for complex turbulent flows of industrial interests is developed using a combination of Large Eddy Simulation (LES) and Immersed Boundary Method (IBM) concepts. LES is an intermediate approach to turbulence simulation in which the onus of modeling of “universal” small scales is appropriately transferred to the resolution of “problem-dependent” large scales or eddies. IBM combines the efficiency inherent in using a fixed Cartesian grid to compute the fluid motion, along with the ease of tracking the immersed boundary at a set of moving Lagrangian points.

Numerical code developed for this dissertation solves unsteady, filtered Navier-Stokes equations using high-order accurate (fourth order in space) finite difference schemes on a staggered grid with a fractional step approach. Pressure Poisson equation is solved using a direct solver based on a matrix diagonalization technique. Second order accurate Adams-Bashforth scheme is used for temporal integration of equations. Dynamic mixed model (DMM) is used to model subgrid scale (SGS) terms. It can represent large scale anisotropy and back-scatter of energy from small-to-large scale through scale-similar term and maintain the energy drain through eddy viscosity term whose coefficient is allowed to change with in the computational domain. This code is validated for several bench-mark problems and is demonstrated to solve complex moving geometry problem such as stator-rotor interaction.

A number of parametric studies on jets-in-crossflow are performed to understand complex fluid dynamics issues pertaining to film-cooling. These studies included effects of variation of hole-aspect ratio, jet injection angle, free-stream turbulence intensity and
free-stream turbulence length scales on the coherent structure dynamics for jets-in-crossflow. Fundamental flow physics and heat transfer issues are addressed by extracting coherent structures from time-dependent three dimensional flow fields of film-cooling by inclined jet and studying their influence on the film-cooled surface heat transfer. A direct method to perform heat transfer calculations in periodic geometries is proposed and applied to internal cooling in rotating ribbed duct. Immersed boundary method is used to render complex geometry of trapped vortex combustor on Cartesian grid and fluid mixing inside trapped vortex cavity is studied in detail.
Chapter 1  Physics, Mathematics and Simulation of Fluid Turbulence

Turbulence is generally referred to as the 19th century problem that is a challenge for the 21st century. Sometimes it is called “one of the unresolved problems of classical physics”. This is partly due to the insufficient mathematical understanding and partly due to the astronomically high computational requirements needed for the full simulations of practical problems. Adding to the frustration is the fact that most of the industrial and geophysical flows are turbulent in nature. With the given physical observations, computational ability and mathematical knowledge, it is then a researcher’s aim to provide the explanation of the flow physics and to develop the predictive capability for the problem of interest. The limited understanding of physics of fluid turbulence makes it difficult to come up with a consistent and accurate mathematical theory for this phenomenon. Moreover, approximation to these mathematical theories will yield simulation models with various degree of success or failure.

Turbulence can be regarded as non-linear, stochastic, highly damped system. It is a state of flow exhibiting randomness in spatial and temporal scales, three-dimensionality of vorticity fluctuations, enhanced diffusion, wide spectrum of excited scales and dissipation (Tennekes and Lumley (1972), Hinze (1975), Monin and Yaglom (1975), Lesieur (1987), Hunt et. al. (1994), Pope (2000)). The enhanced diffusion due to mixing and energy cascade due to non-linear interactions give the turbulent flows a visco-elastic nature. However, this behavior is a flow state not a fluid property. Usually the onset of turbulence is attributed to the loss of stability of laminar flow state to some disturbances with ever increasing rapidity as a control parameter in a flow problem is increased. This control parameter is usually Reynolds number ($Re = UL/\nu$) that expresses the balance
between the nonlinear and dissipative properties of the flow. In shear flows, disturbances that have not yet adapted to the flow, are rotated to become adapted and complemented by their non-linear interaction, which remixes and recreates misfit components anew.

“This mechanism rests on the bunching of a subset of eigenfunctions of disturbances, induced by their coupling to the laminar flow advection leading to an algebraically increasing and sustained fluctuating disturbance called turbulence” (Grossmann, 2000).

“Turbulence and critical phenomenon share the feature that a continuous range of scales is excited in both; however, they are different in that the fluctuations in turbulence are strong and there exists no small parameter. Thus, turbulence is a paradigm in non-equilibrium statistical physics, in which fluctuations and macroscopic space-time structure coexist. It is an example like no other of spatially extended dissipative systems” (Sreenivasan, 1999).

The Navier-Stokes (N-S) equations govern the evolution of the flows of interest. These equations represent the momentum balance for continuum where the stress field is proportional to the rate of strain. Non-linearity appears naturally in these equations via chain rule of differentiation. The existence, uniqueness and smoothness of the Navier-Stokes solutions in three-dimensional space is an open problem and is announced as one of the seven millennium prize problems by Clay Mathematics Institute (http://www.claymath.org/prizeproblems/navierstokes.htm). These equations are used as model for fluid turbulence. It would not be an understatement to say that despite the huge amount of experimental and theoretical/computational studies done on this subject for more than a century, our understanding of these equations and fluid turbulence is at an infancy stage (Tsinober, 2002 and Frisch, 1995).
The Navier-Stokes equations are

\[ \frac{\partial u_i}{\partial t} + u_j u_{i,j} = -p_{,i} + \nu u_{i,ji} \]  

(1.1)

The flow must also satisfy mass conservation (also known as continuity equation) which in the case of incompressible fluids simplifies as

\[ u_{i,i} = 0 \]  

(1.2)

The possibility that these equations can produce finite-time singularities is not merely a mathematical formality. Such singularities imply that the subsequent evolution of the solution may be non-unique and that is in contradiction with the deterministic nature of the model. These singular solutions also represent the generation of structures on arbitrarily small scales by the equations and that is in contradiction with the separation-of-scales assumption used to derive the Navier-Stokes equations from the microscopic models. The non-linear terms that are mathematically uncontrollable represent the generation mechanism of turbulence via vortex stretching. Therefore, the question of existence, uniqueness and regularity is directly related to the efficacy of the Navier-Stokes equations as a model for fluid turbulence (Doering and Gibbon, 1995). Although there is huge amount of experimental and computational support for the Navier-Stokes equations as a model for fluid turbulence (Foias et al., 2001), yet whether or not these equations may display the above-mentioned pathologies remains an open problem. The results of existence and uniqueness are different according to space dimension (Sohr (2001)):

- “In space dimension d=2, the theory is fairly satisfactory, the problem is well posed in the sense of Hadamard; existence and uniqueness of weak solutions, of
strong solution if the data are suitably regular; more generally the solution is as regular as allowed by the data, and we have continuous dependence on the data in the corresponding function spaces.

- In space dimension $d=3$, we have only partial results: existence and uniqueness of a strong solution on some interval $(0, T^*)$, $T^*$ depending on the data; existence of weak solution on $(0, +\infty)$. Uniqueness of weak solutions is still an open problem, as well as the existence for all time of strong solutions.”

![Schematic of energy cascade and energy spectrum for three-dimensional turbulence.](image)

Figure 1.1 Schematic of energy cascade and energy spectrum for three-dimensional turbulence. (Physical space picture depicts the Richardson cascade of breakdown of large eddies into smaller eddies and eventually dissipating the energy to viscous forces. The spectral space picture shows the Kolmogorov-Obukhov inertial range)
The mathematical description of turbulence involves several approaches with different phenomenological approximations to Navier-Stokes equations:

1. Kolmogorov theory of locally homogeneous and isotropic turbulence is based on the statistical independence of the small and large scales of turbulence. It assumes that the transport of the energy from large energy containing scales to the dissipation range proceeds by a cascade process the mechanism of which is independent of the energy production events and fluid viscosity (Kolmogorov 1941, Obukhov, 1941). This theory has little bearing on Navier-Stokes equations. However, the experimental support for Kolmogorov’s ideas on inertial range is enormous (Frisch, 1995). Statistical description is given in terms of correlation tensor of fluctuating fields and their structure functions. Clearly, the mathematical simplicity is achieved at the price of throwing away the deterministic features of flow. (Taylor, 1935, Batchelor, 1953)

2. Coherent structure description has been accepted as an essential step in understanding the inhomogeneous turbulent flows, however the relationship with Navier-Stokes equations is fairly weak (Townsend, 1956). Moreover, the variations of the coherent structures topology, production mechanism and their dynamics from one flow situation to another make it extremely difficult to come up with a unified and general theory.

3. Characteristic functional approach of Hopf is based on the fact that the knowledge of the characteristic function is equivalent to that of the probability density function (p.d.f.), of which it is the Fourier transform. It is used as the generating
functional of (single-time) moments of the velocity fields to derive the hierarchy of cumulants (Stanisic, 1988).

4. Renormalized Perturbation Theories (RPT) use the idea of expanding the flow fields as the perturbation terms in non-linearity. Subsequently, the assumptions on the non-linear triadic interactions and restoration of random Galilean invariance can yield different flavors e.g. Direct Interaction Approximation (DIA) and Lagrangian History DIA (Kraichnan, 1959, Leslie, 1973)

5. Renormalization Group (RG) method relies on determining the effective turbulent viscosity by systematically reducing the number of degrees of freedom. It is assumed that in viscous range of wavenumbers, the injected energy is dissipated locally by the effects of molecular viscosity. Recursive application of this procedure would produce an increased effective viscosity and a reduced number of degrees of freedom (McComb, 1990). The notion of such effective viscosity has existed for a long time (Heisenberg, 1948).

6. Dynamical systems approach postulates a global attractor to the long time behavior of the solutions. (Ruelle and Takens (1971)). It also suggests that the turbulent state would be reached after the fluid system had undergone a finite and small number of bifurcations.

Since Navier-Stokes equations are non-linear in nature and the flows are associated with a spectrum of scales ranging from the dissipative Kolmogorov scales to the energy containing integral scales, the dynamics of these scales or their effects must be accurately represented and resolved in the simulations. With the advent of supercomputers, it is now possible to perform full direct numerical simulations (DNS) of simple turbulent flows at
moderate Reynolds numbers. In DNS, the unsteady Navier-Stokes equations are solved on spatial and temporal meshes that resolve the smallest scales. The information obtained by such simulations is enormous and has led to understanding the turbulence physics and modeling issues with greater insight. However, the computational requirements for DNS of complex turbulent flows at high Reynolds number are beyond the capabilities of the foreseeable supercomputers. As an engineer, it is important to perform calculations in a cost-effective manner, and this requires the modeling of some universal aspects of turbulent. This has led to the development of turbulence modeling and subgrid-scale modeling (Gatski et al. (1996), Durbin and Pettersson Reif (2001)).

It was more than a century ago, when Osbourne Reynolds proposed the description of turbulent flows in terms of ensemble-averaged fields and the fluctuation fields. Due to the non-linear nature of the problem, a closure in terms of the resolved scale fields is impossible. Thus, it is necessary to introduce a model that mimics the essential physics, but uses only the information from the resolved fields. The simplest turbulence models are based on the approximations similar to those made in the kinetic theory of gases. The higher level of approximation needs more parameters to be resolved. However, at the end of such higher level approximations, one relies on a simplistic closure approximation with the belief that error incurred at the higher level in approximation hierarchy does not effect the flow physics severely. The effect of fluctuation fields on the ensemble-averaged fields is usually modeled using the aforementioned approximations. Such decomposition is related to long time experimental averages under restrictive conditions. In Reynolds-Averaged Navier-Stokes (RANS) simulation, one is interested in the mean flow fields only. However, the issues of
modeling errors and the non-universality of such turbulence models have rendered the use of such approximations futile in the case of complex turbulent flows (particularly for the cases for which the turbulence models are not calibrated).

Large eddy simulation (LES) of turbulent flows is an approach intermediate to DNS and RANS. In LES, one simulates the large scales of the flows that are dependent on the boundary conditions and contains most of the kinetic energy of the flow. The small scales or subgrid scales (SGS) are expected to be more universal and isotropic in nature. Since, the small scales are problem independent and contain small fraction of energy, modeling these scales would yield more universal and accurate turbulence models. To achieve decomposition in terms of resolved fields and subgrid fields, one generally applies a spatial filtering operation. Though, LES seems to have better physical ground for turbulence modeling, it has some severe mathematical constraints. Moreover, the computational requirements are still very high as compared to those of RANS (Sagaut, 2001).

The current CFD practitioners in industry continue to use RANS modeling in view of the more modest computational requirements. However, current supercomputers have provided enough computational capability to attempt DNS and/or LES of complex turbulent flows at moderate Reynolds numbers. The quest for universal turbulence models has been the subject of rigorous research for several decades. DNS and LES have given new directions for such modeling issues. In this chapter, only an overview of the underlying physics, mathematical techniques and computational challenges for fluid turbulence is presented. However, interested readers are strongly encouraged to follow the cited references for more complete account of the issues.
In chapter 2, the basic equations for LES are presented. Fundamental steps involved in deriving LES equations are SGS modeling and filtering. Review of few state-of-the-art SGS models is presented along with derivation of Dynamic Mixed Model (DMM). Some issues with filtering are also explained. In chapter 3, the details of numerical procedure are provided. Immersed boundary method (IBM) is very attractive approach to simulate complex moving geometries on fixed grids using body force terms in numerical procedure. A general methodology to incorporate complex moving geometries while retaining high-order of accuracy is presented. It combines IBM with LES solution procedure. Several validation cases are documented. Unsteady stator-rotor interaction is studied to demonstrate the vast potential of this methodology in complex moving geometries. In chapter 4, some mathematical tools are provided to analyze the details of turbulent fields. In particular, a brief introduction to proper orthogonal decomposition (POD) is presented. A very simple criterion for extracting coherent structures from time-dependent three-dimensional flow fields is derived. Some indicators for mixing of passive scalar field are also presented. In chapter 5, several parametric studies for jets-in-crossflow configuration are presented. These studies include: effect of hole-aspect ratio, effect of jet injection angle, effect of freestream turbulence intensity and effect of freestream turbulence length scale. To understand the film-cooling of gas turbine blades, LES with heat transfer calculations are performed for an inclined circular jet injection in crossflow. Fundamental flow physics and unsteady heat transfer processes are explained by extracting coherent structures from time-dependent flow fields. In chapter 6, internal cooling of gas turbine blades is studied. A direct procedure to calculate source term in unsteady non-dimensional energy equation is derived for periodic
geometries. Larger computational domain is selected that convincingly demonstrates the non-periodic nature of flow and heat transfer in single pitch. Coherent structure dynamics revealed the disintegration of organized roller vortices shed from the rib turbulators by secondary flows in the duct. POD analysis of two hundred snapshots revealed the low-dimensional character of the system. In chapter 7, mixing processes and flow physics of a trapped vortex combustor are analyzed using LES-IBM. Complex geometry of dumb-bell shaped flame holder is rendered using IBM. Flow details are presented with implications on mixing derived from turbulent stress distributions. Finally, concluding remarks and future directions for the research work are presented in chapter 8. Several appendices are also provided to elaborate and supplement the discussions in the dissertation.
Chapter 2   Large Eddy Simulations of Complex Turbulent Flows

Direct numerical simulation (DNS) of turbulent flows solves the Navier-Stokes equations along with continuity equation without using any kind of closure approximation or turbulence model. Since the degrees of freedom active in a turbulent flow at Reynolds number $Re$ are approximately $Re^{9/4}$, the full resolution requirements for high Reynolds number flows are astronomical. Moreover, most of the energy is contained in a few of the low wavenumber or frequency modes and a very large number of degrees of freedom correspond to the high wavenumbers or frequencies present in the flow. Therefore, it is beneficial from computation cost as well as engineering point of view to resolve only the energy containing low wavenumbers or frequencies. DNS can be viewed as one end of turbulence computations, needing no turbulence model at all. On the other extreme of turbulence modeling, Reynolds averaged Navier-Stokes (RANS) equations are obtained by performing ensemble-averaging operation on Navier-Stokes equations. Thus, the velocity or scalar fields are decomposed into a mean and fluctuating component. The effect of unresolved fluctuating components is rendered through the Reynolds stress tensor. The governing equations for the turbulent Reynolds stress tensor can be obtained by taking the moments of governing equations for fluctuation fields. These equations will contain terms involving unclosed higher moments of fluctuating or unresolved fields. This is the classical “closure problem” of turbulence. Therefore, the components of Reynolds stress tensor must be modeled in terms of the mean fields at some level of this hierarchy. However, the task of embodying underlying physics for various flow situations into these turbulence models is an extremely difficult task. Moreover, there is no rational basis to assume that the fluctuating components contain small fraction of energy. Large
eddy simulation (LES) is an intermediate approach to DNS and RANS (Sander, 1998). In LES, the energy containing large scales are separated from instantaneous flow field using “appropriate” filter. These large scales depend strongly on the boundary conditions and hence determine the basic features of the flow field for various flow situations. Thus, the governing equations are obtained for filtered fields containing the unclosed correlations of sub-filter or sub-grid fields (Germano, 1992 and Mason, 1994). The unresolved small scales are mostly isotropic and more universal in nature and hence, modeling of these subgrid scales (SGS) would be more rational and universal. The non-linear transport of energy generates ever smaller scales like a cascade process (also called Richardson cascade) until it reaches the viscous dissipation range or the size of Kolmogorov scales. The SGS model should account for this energy drain from resolved large scales to the unresolved small scales properly.

![Energy spectrum of high Reynolds number turbulent flow.](image)

Figure 2.1 Energy spectrum of high Reynolds number turbulent flow.

The energy spectrum for a high Reynolds number turbulent flow depicting the energy cascade process is shown in figure 2.1 (also see Figure 1.1). The energy is transferred
from low wavenumber (large energy containing) scales to high wavenumber (dissipative) scales. For a given simulation, the resolution limitation determines $k_r$, the resolvable wavenumber on a finite grid. For DNS, the resolution limit is the dissipative wavenumber (Kolmogorov scale) $k_d$. The rationale of LES is to resolve up to the cut-off scale $k_c$, which contains most of the energy of the flow and uses SGS model to simulate the effect of energy drain by high wavelengths. The inclusion of SGS model is essential for energy preserving numerical schemes. In the absence of any SGS model, the simulation (referred to as coarse DNS) will accumulate energy at the high wavenumber end and will give rise to unphysical results. The energy conserving schemes are preferred for such accurate numerical computations, rendering the use of SGS models inevitable. Therefore, an appropriate SGS model determines the success of LES. There are other issues pertaining to filtering operations in complex geometries and will be addressed later in this chapter.

The need for SGS models in meteorological simulations was the first step towards the development of LES. Smagorinsky (1963) proposed an eddy-viscosity type model to account for energy cascade in spatially under-resolved time dependent simulations. In this model, the components of SGS tensor were assumed to align with the resolved strain-rate tensor. The proportionality constant (called the Smagorinsky constant) for this model was derived for homogeneous and isotropic turbulence by Lilly (1966). Deardorff (1970) performed the numerical calculations for three-dimensional channel flows at high Reynolds number using this model. Leonard (1974) introduced the idea of separating the resolved field by convoluting the instantaneous field and a filter kernel. Schumann (1975) formulated such filtering operations as the spatial volume averages. Leslie and Quarini (1979) presented the formal theoretical arguments for subgrid modeling procedures.
Clark (1977) investigated the SGS tensor formulations using fully simulated isotropic turbulence cases. Bardina (1983) presented the idea of scale-similarity in SGS modeling, which later was proven to be the leading term of the expansion of subgrid fields in terms of filtered fields. Speziale (1985) introduced the constraint of Galilean invariance on the SGS model, since the original unfiltered equations satisfy this constraint. Germano et al (1991) presented a dynamic subgrid scale model that did not need model coefficient \( a \) \textit{a priori}. The dynamic procedure is very important because it removed all empiricism from a computation mathematically, yielding a consistent and universal approach. Germano (1992) presented an operational approach based on general algebraic properties of filtered representations of a turbulence field at different levels. Ronchi et al (1992) discussed the basic assumptions and conceptual difficulties of this approach. Mason (1994) critically reviewed the LES technique and the assumptions involved in SGS closure. Vreman et al (1994) derived the constraints on the filter kernel using “realizability conditions”. Ghosal and Moin (1995) derived the basic governing equations for LES in complex geometry to address the issue of commutation errors. Ghosal (1996) presented the analysis of numerical errors in LES. Carati and Eijnden (1997) discussed the underlying self-similarity assumption in dynamic procedure for LES. Fureby and Tabor (1997) presented the mathematical and physical constraints on LES. Oberlack (1997) applied the symmetries of Navier-Stokes equations as constraints on filtered LES equations to determine the filter kernel and SGS model. Canuto and Cheng (1997) showed that the SGS model coefficient depends on the combination of physical processes that may differ from flow to flow and hence, is a variable that should dynamically adjust itself to different flows. Adrian (1999) used the concept of mean-square optimal algorithms to
derive the governing equations in physical space of large eddies defined by an explicit and inhomogeneous filtering operation. Pope (2001) presented a LES methodology with projection on local basis functions to circumvent the issues regarding the numerical discretizations. All of the above mentioned work has contributed towards the foundations of the theory for LES.

Applications of LES have been growing at an enormous pace in almost every kind of turbulent flow. Since the origin of LES was in geophysical flows, the research done in the areas of atmospheric sciences, physical oceanography and various environmental flows has greatly increased (Galperin and Orszag (1993), Zang (1993), Khanna and Brasseur (1997), Seigel (1998), Agee and Gluhovsky (1999)). However, the focus of current work is on complex industrial flows and therefore, only the recent advances in LES applications during last decade will be cited. Moreover, only those applications that use the dynamic procedure for the SGS model will be presented. El-Hady et al (1994) simulated transitional boundary layer flow along a cylinder at Mach number 4.5. Jones and Wille (1996) presented the calculations of a plane jet in a crossflow with different SGS models and used mesh expansion ratios such that the commutation errors were an order of magnitude smaller than the associated spatial derivative terms. Ghosal and Rogers (1997) studied the self-similarity in a turbulent plane wake using LES with dynamic localization subgrid model. Horiuti (1997) demonstrated the better predictive ability of dynamic two-parameter mixed models for plane channel and mixing layer flows. Wu and Squires (1997) used LES for prediction of an equilibrium three-dimensional turbulent boundary layer. Im et al (1997) applied LES to turbulent premixed combustion. Vreman et al (1997) presented the results of LES of turbulent mixing layer

Center for Turbulence Research (CTR) at Stanford University, has contributed a lot towards foundations of LES and the industrial applications of LES (http://www.ctr.stanford.edu). Further three international conferences were organized by AFOSR dedicated to the advances in the area of DNS and LES in the past five years.

With ever-increasing computational capabilities, LES seems to be the approach that should be adopted for high Reynolds number flows. However, LES lacks a sound mathematical foundation Layton and his group has been working on the function analytic theory for errors and modeling in LES (Their publications are available on the URL http://www.math.pitt.edu/~wjl/). They proved that the commutation error goes to 0 in $L^p$ if and only if the normal stress of the turbulent velocity is identically zero everywhere on the boundary. This means that any numerical method discretizing the strong form of the LES equations, such as a finite difference method, makes an $O(1)$ error! Moreover, the commutation error does go to 0 in an appropriate weak sense. Therefore, variationally based methods, such as FEM, spectral methods and spectral element methods are acceptable. Research efforts in the area of SGS modeling are also very important. It is expected that DNS and LES will aid in the development of RANS turbulence models. Hence, for the cases, where computational requirements are beyond today’s supercomputers, better RANS models will be available.

In the following sections of this chapter, the issues of SGS modeling and filtering techniques will be discussed in detail.
2.1 Subgrid Scale (SGS) Modeling

The unresolved or subgrid scales affect the dynamics of the resolved flow field through the subgrid scale stress tensor. If, somehow, an exact representation of this tensor were available and there were no numerical errors, LES would produce the exact large-scale field and we would also have the unresolved contributions. Such a simulation would yield all the data one could hope for. Thus, an accurate subgrid scale model is the key to quality large eddy simulations. As the Reynolds number increases, the fraction of the total field that is unresolved also increases, the model is required to represent a larger range of turbulence scales, and the accuracy of a simulation becomes more sensitive to the quality of the SGS model.

The SGS tensor is generally represented as a functional of the resolved fields via a model coefficient. These functional relationships are mostly intuitive and embody different physical processes. The model coefficient in an SGS model must be a spatial variable in case of inhomogeneous and anisotropic flows. The dynamic procedure can be applied to any base model to determine value of the coefficient. Based on the functional relationships, the SGS models can be broadly classified into a) eddy-viscosity models, b) One-equation SGS models, c) Reynolds stress and algebraic models and d) scale-estimation models.

The homogeneously filtered incompressible Navier-Stokes equations take the following form,

\[
\frac{\partial \overline{u}_i}{\partial t} + \frac{\partial \overline{u}_i u_j}{\partial x_j} = -\frac{\partial \overline{p}}{\partial x_i} + \nu \frac{\partial^2 \overline{u}_i}{\partial x_i^2} - \frac{\partial \overline{u}_i u_j - u_i u_j}{\partial x_j}
\]  

(2.1)
The form of the filtered equations is similar to original equations except for the last term, which is the SGS contribution of the unresolved/subfilter field on the resolved/filtered field. The SGS tensor is defined here as,

$$\tau_{ij} = u_i u_j - u_i u_j$$  \hspace{1cm} (2.2)

The aim of the SGS model is to represent various physical processes at the subgrid level in terms of the resolved fields or some estimate of the subgrid fields. In the following subsections, the details of various types of SGS models are presented.

- **Eddy-Viscosity Model**

Smagorinsky (1963) introduced the model for SGS tensor in terms of the resolved strain-rate tensor. The model coefficient referred to as Smagorinsky “constant”, must be known or calibrated prior to the simulation (Appendix II). This would result in a Heisenberg-type eddy viscosity expression in physical space (Kraichnan (1976), Stanisic (1988), McComb (1990) and Voke (1996)). Thus, the generic expression for such SGS model is

$$\tau_{ij} = -2\nu_i S_{ij}$$

$$S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$$

$$\nu_i = C_S \Delta^2 \left| \frac{S_{ij}}{S_{ij}} \right|$$  \hspace{1cm} (2.3)

The “constant” $C_S$ is the only input required before simulation. However, the spatial variation of this “constant” within the flow makes it difficult to find the “correct” value. Moreover, the model constant changes for different flow configurations and hence, renders the model non-universal. Lilly (1966) determined the value of $C_S$ for homogeneous, isotropic turbulence case ($C_S \sim 0.18$). Similar extensions of this model are available in spectral space using turbulence theories (Chollet (1983, 1985), Bertoglio...
This base model has been used extensively in the dynamic procedure for the flow prediction in various situations with reasonable amount of success. In these simulations, the model coefficient is computed dynamically using Germano’s identity and averaged spatially/temporally in some fashion to avoid numerical instability.

- **One-Equation Model**

From dimensional analysis, the eddy viscosity \([L^2T^{-1}]\) needs to be modeled from the knowledge of two scales. This is similar to the idea of two equation turbulence models expressing the turbulent eddy viscosity in RANS simulation. However, LES has a length scale for eddy viscosity determined by the filter width, \(\Delta\). Therefore, only one equation is needed to derive an expression for eddy viscosity properly (Ghosal et al (1995), Davidson, (1997), Pomraning and Rutland (2002)). This may seem to be an extension of eddy-viscosity models, but the solution of such a scale equation can resolve the issue of ad-hoc user modifications to ensure numerical stability due to direct dependence of eddy viscosity on the SGS kinetic energy (Appendix III). In another approach, the global optimization of model coefficient in dynamic procedure leads to Fredholm's integral equation of second kind (Ghosal et al (1993,1995), Piomelli and Liu (1995)).

- **Reynolds Stress or Differential Stress Model**

Most of the models assume isotropy for the SGS stress tensor. The issue of modeling flow anisotropy at SGS level in the low-resolution simulations can be addressed using a modeled version of the balance equations for the SGS stress tensor. Deardorff (1973) applied a differential SGS model to atmospheric boundary layers. Canuto and Cheng (1997) used algebraic representation of full SGS stress tensor to evaluate the assumptions
needed in Smagorinsky's model. Fureby et al (1997) simulated the homogeneous isotropic turbulence and fully developed channel flow using these models. The details of differential stress model along with the various approximations made during SGS modeling are presented in Appendix IV. The results obtained by such an elaborate model reproduce the interscale energy transfer better for a larger range of Reynolds numbers than traditional SGS models.

- **Scale Estimation Model**

The traditional approaches to SGS modeling involving Smagorinsky’s model or the gradient model make several assumptions about the relation between SGS tensor and the filtered fields. As an alternative formulation, the subgrid scales can be estimated and thus, the SGS tensor can be written explicitly in a closed form (Bardina, 1983, Germano, 1986a, 1986b, Shah and Ferziger, 1995, Scotti, 1998, Domaradzki and Loh, 1999, Geurts, 1997, Stolz and Adams, 1999, Kuerten et al, 1999). Leonard (1974) carried out the asymptotic expansion of SGS field in terms of filtered fields to estimate the resolvable part of SGS stress. Clark (1977, 1979) used similar idea to develop a gradient model for the eddy viscosity SGS model. Bardina (1983) introduced a non eddy-viscosity model based on the scale-similarity assumption. The similarity of largest unresolved scales with the smallest resolved scales led to the expression of SGS stress tensor in terms of resolved fields only. It was shown later that the leading term in Taylor series expansion of any SGS model is the scale-similar resolvable part. Hence, it is the mathematical consistency as well as the decreased load on the model that inspires the use of scale-similar term in the SGS model. However, this term all by itself can not dissipate enough energy from the large scales and therefore, a combination of Smagorinsky's eddy
viscosity model along with such scale-similar model is considered. The scale-similar part resolves the flow inhomogeneity and anisotropy at SGS level while the eddy viscosity part dissipates the energy transferred to SGS scales (Shao et al, 1999). Aldama (1990, 1993) continued the work of scale decomposition and estimation via a truncated series. Bedford and Yeo (1993) developed the theory of conjunctive filtering for a "closure-free" model. Misra and Pullin (1997) developed a vortex-based model where the subgrid structure of the turbulence was assumed to consist of stretched vortices whose orientations were determined by the resolved velocity field. Voelkl et al (1999) formulated the vortex-based SGS model in physical space. Scotti (1998) used fractal interpolation with an assumption about the existence of inertial range to estimate the subgrid scales. Domaradzki and Saiki (1997) developed a two-step estimation procedure. The first step utilizes properties of a filtering operation and the representation of quantities in terms of basis functions. In the second step, the phases associated with the newly computed smaller scales are adjusted in order to correspond to the small-scale phases generated by non-linear interactions of large-scale field. Domaradzki and Loh (1999) formulated their spectral space estimation procedure in physical space. This procedure involves a deconvolution step followed by generation of smaller scales through non-linear interactions on a finer mesh. Other deconvolution procedures such as the polynomial filter inversion of Geurts (1997) and approximate deconvolution method (ADM) of Stolz and Adams (1999) are also shown to be less computationally demanding as compared to current dynamic models. The inverse filtering approach has provided a more consistent foundation for the SGS modeling (Kuerten et al, 1999). Gallerno and Napoli (1999) has derived a tensorial eddy viscosity model that does not assume the
alignment of principal axes of resolved strain rate tensor and the modeled part of the SGS stress tensor.

All the results presented here are obtained using the dynamic smagorinsky model (DSM) or dynamic mixed model (DMM) of Zang et al (1993). The effects of various SGS models and numerical schemes are not considered here. Details of DMM are presented along with the mathematical correction suggested by Vreman et al (1994a).

Subgrid stress tensor is defined as the second moment of subfilter fields and filters are defined by the angular brackets with superscripts denoting the filter width. Equations (2.4) therefore, defines Subgrid (actually Subfilter to be absolutely correct!!) stress at grid filter and test filter levels.

\[
\begin{align*}
\tau_{ij} &= \langle u_i u_j \rangle^g - \langle u_i \rangle^g \langle u_j \rangle^g \\
T_{ij} &= \langle u_i u_j \rangle^{gh} - \langle u_i \rangle^{gh} \langle u_j \rangle^{gh}
\end{align*}
\]  

Using the fact that the difference of test filter level stress and test filtered grid level subgrid stress is defined in terms of all filtered fields only, Germano Identity is obtained.

\[
T_{ij} - \langle \tau_{ij} \rangle^h = L_{ij} \equiv \langle \langle u_i \rangle^g \langle u_j \rangle^g \rangle^h - \langle u_i \rangle^{gh} \langle u_j \rangle^{gh}
\]  

Figure 2.2 Germano’s identity for SGS stresses at two filter levels
Germando identity (eq. 2.5) is a tautology and is a well known result of classical physics (Poisson’s relation for commutators). The actual use of Germando identity is in determining the model coefficient dynamically in the solution domain. It is noted that part of Subgrid stress is resolvable at any grid level and is given by the scale similar term of Bardina model. Therefore, SGS tensor is modeled by a combination of a Smagorinsky eddy viscosity model and a scale-similar term (eq. 2.6).

\[
\tau^{a}_{ij} = \left( \langle u_i \rangle^g \langle u_j \rangle^g - \langle u_i \rangle^{gg} \langle u_j \rangle^{gg} \right)^a - 2C_s \Delta^2 g |S^g| S^g_{ij}
\]

(2.6)

\[
T_{ij} = \left( \langle u_i \rangle^{gh} \langle u_j \rangle^{gh} - \langle u_i \rangle^{ghgh} \langle u_j \rangle^{ghgh} \right)^a - 2C_s \Delta^2 gh |S^{gh}| S^{gh}_{ij}
\]

Applying Germando identity on these modeled stresses and defining the tensor with model coefficient as \( M_{ij} \) and remaining terms as \( H_{ij} \) (eq. 2.7 and 2.8), a tensor identity is obtained for the model coefficient.

\[
H_{ij} \equiv \left( \langle u_i \rangle^{gh} \langle u_j \rangle^{gh} - \langle u_i \rangle^{ghgh} \langle u_j \rangle^{ghgh} \right) - \left( \langle u_i \rangle^g \langle u_j \rangle^g \right) - \left( \langle u_i \rangle^{gg} \langle u_j \rangle^{gg} \right)^h
\]

(2.7)

\[
M_{ij} \equiv -2\Delta^2 gh |S^{gh}| S^{gh}_{ij} + 2\Delta^2 g |S^g| S^g_{ij}
\]

(2.8)

A least squares method is used to determine the coefficient for this over-determined problem. Note that the expression for numerator and denominator are smoothed in some sense to avoid numerical instabilities (eq. 2.9). Also, the model coefficient is assumed to vary smoothly over the test filter width and is taken outside the averaging operation for subgrid tensor at grid scale

\[
\Rightarrow H^{a}_{ij} + C_s M_{ij} = L^{a}_{ij}
\]

\[
\therefore C_s = \frac{\langle M_{ij} (L_{ij} - H_{ij}) \rangle}{\langle M_{ij} M_{ij} \rangle}
\]

(2.9)
• Implicit SGS Model

Monotone nonlinear convection algorithms have a built-in filter and a corresponding built-in subgrid model. These monotone "integrated" LES (MILES) algorithms are derived from the fundamental physical laws of causality and positivity in convection and do minimal damage to the longer wavelengths while still incorporating, at least qualitatively, most of the local and global effects of the unresolved turbulence expected of LES. Monotone convection algorithms such as Flux-corrected Transport (FCT) have adequate numerical diffusion and they reduce the Gibbs error optimally (ed. Lumley (1990), Grinstein and Fureby, (1998)). Thus, implicit SGS models rely upon the truncation errors for the numerical diffusion. Therefore, the comparison of solution fields becomes imprecise due to the lack of knowledge about the filter function. Moreover, an attempt to improve grid resolution to resolve mean variable may trigger instability because of decreased numerical dissipation. These models are also inferior to sophisticated dynamic models used explicitly in a simulation in terms of higher order statistics of the resolved fields. For sufficient grid resolution and simple flows, implicit SGS models can be useful. Moreover, these models do not incur any overhead associated with calculations of subgrid stress terms and can lead to substantial saving in computational effort. However, in a robust methodology for general applications, it is necessary to control the dissipation due to SGS model apart from dissipation due to numerical schemes. Hence, the explicit SGS modeling approach is followed here for the sake of accuracy of results since the intent is to calculate higher order correlations appearing in the modeled equations.
2.2 Filtering Techniques for Large Eddy Simulations

Large eddy simulation (LES) approach is aimed to make such simulations feasible with minimal modeling effort in order to retain the flow physics as much as possible. The rationale behind such an approach is the fact that, while the large-scale dynamics in turbulent flows are very sensitive to flow domain geometry and boundary conditions and, therefore, vary from flow to flow, the small-scale behavior tends to be quite universal. Accordingly, in LES calculation the large-scale motion is explicitly resolved and the effects of the unresolved scales are accounted through the use of subgrid scale (SGS) models. The explicit resolution of large scales is made possible by applying a filtering operation to the equations of motion, thus obtaining filtered equations which govern the dynamics of the large scales. These equations are numerically solved employing mesh spacings that are of proper size for resolving the large scales. Clearly, LES procedure has two crucial steps: Filtering and Subgrid scale modeling. In this section, the focus will be on the filtering techniques for LES.

Leonard (1974) introduced the concept of filtering as the convolution operation. The mathematical and physical constraints on the filter kernel have been presented (Ghosal (1998), Speziale (1985), Fureby and Tabor (1997), Vreman et al (1994b), Oberlack (1997)). Germano (1986) defined differential filters for LES so that the attenuation of filtered field can be controlled. Zhou et al. (1989) criticized the use of filters that lead to the filtered field and unfiltered field with same spectral support. This ambiguity of resolved/filtered and unresolved/subfilter scales has led to the use of compact support filter kernels (Fureby and Tabor (1997), Fureby et al (1997)). In fact, the process of separation of fields into filtered and fluctuating components yields a two-scale
separation. However, the discretization via the numerical grid introduces a third separation resulting from the introduction of the Nyquist wave number and its equivalence to the cutoff wave number (Bedford and Yeo (1993), Aldama (1990), Mason (1994)). The finite support of the numerical grid and the wave number dependent truncation errors associated with finite difference operators are assumed to define a filter operation. However, explicit filtering can be used as a means of controlling truncation error by simply removing from the simulation the smallest motions that would otherwise be affected by the error (Lund and Kaltenbach (1995), Ghosal (1996)). It can be shown that the filtering and differentiation do not commute for the variable filter width for the general case. Thus, non-commutation of these operations gives rise to the commutation errors. The basic equations for LES in complex geometry are derived in Ghosal and Moin (1995). The commutation errors were expressed in terms of the filtered field and its derivatives as an asymptotic series in the square of filter width. Clearly, such an error might not be acceptable if one is using higher order differencing scheme or pseudo-spectral method. Ghosal and Moin (1995) addressed the issue of commutation errors using correction terms. Blaisdell (1997) attempted to resolve this issue using the fact that the discrete operators (or matrices) commute if they have the same eigenvectors. Van der ven (1995) proposed a class of commuting filters with non-uniform width. Jordan (1998) proposed the methodology of filtering along the curvilinear coordinates to eliminate the commutation errors for certain boundary conditions. However, none of these methods addressed the issue of complex geometry and/or boundary conditions satisfactorily. Vasilyev et al (1998) presented a general theory for explicit filtering in which the filtering and differencing commute up to the numerical truncation error of the scheme employed.
However, the filters constructed using wavelets that satisfy the vanishing moments conditions are non-realizable. Cook (1999) presented a methodology for implementing variable filter width operator in the adaptive mesh refinement framework that is free of commutation errors. Wagner and Liu (2000) used the reproducing kernel particle method (RKPM) as the LES filter to keep the commutation errors below the truncation error. Some researchers are advocating the use of spectro-consistent discretization schemes and locally averaged direct numerical simulations (Verstappen and Veldman (1998), Veldman and Rinzema (1992), Denaro (1996)). Recently, Adrian (1999) used the concept of mean-square optimal algorithms to derive physical space equation for LES using an explicit and inhomogeneous filtering operation. This approach is also free of commutation error. Pope (2001) presented an approach using filtered field as a projection onto local basis functions. The resulting LES equations are ordinary differential equations of the evolution of the basis function coefficients.

In the following subsections, the mathematical and physical constraints for the filter kernel will be presented. Since the much-celebrated dynamic procedure for SGS modeling also uses filtering, it is important to ensure that filtering operation satisfies the basic assumptions of such procedure. Various viewpoints are explained and compared to address the aforementioned issues.

- Realizability Constraints

Proof (Vreman et al (1994b)): Realizability conditions imply that the SGS tensor should be positive semidefinite. Let the filter kernel $G(x,y)$ be positive for all $x$ and $y$. The following expression defines an inner product on the space of real functions.

$$ (f, g) = \int G(x,y) f(y)g(y) dy $$

(2.10)
Now, consider the turbulent stress as following using the definition of filter operator,

\[ \tau_{ij}(x) = u_i(x)u_j(x) - \bar{u}_i(x)\bar{u}_j(x) \]

\[ = u_i(x)u_j(x) - \bar{u}_i(x)\bar{u}_j(x) - \bar{u}_i(x)u_j(x) + \bar{u}_i(x)\bar{u}_j(x) \]

\[ = \int G(x,y)u_i(y)u_j(y)dy - \bar{u}_i(x)\int G(x,y)u_j(y)dy \]

\[ + \bar{u}_j(x)\int G(x,y)u_i(y)dy + \bar{u}_i(x)\bar{u}_j(x)\int G(x,y)dy \]

\[ = \int G(x,y)(u_i(y) - \bar{u}_i(x))(u_j(y) - \bar{u}_j(x))dy = (w_i^*, w_j^*) \]

\[ w_i^* = u_i(x) - \bar{u}_i(x) \]

Therefore, the SGS tensor forms a 3×3 Grammian matrix of inner products and hence is always positive semidefinite. On the contrary, if \( G(x,y) \) is negative at some point in its support then for a field which is non-zero at the location where the filter kernel is negative and zero elsewhere, the normal stress components of the stress tensor will be negative. Thus, the SGS tensor is positive semidefinite if and only if the filter function \( G(x,y) \) is positive.

Figure 2.3 Commonly used realizable filter kernels.

Alternatively, one can express the continuous integrals in terms of discrete sums and use the Cauchy-Schwarz inequality to show that realizability conditions are true only
for non-negative filters. However, if the Reynolds number of turbulence is sufficiently high so that the “effective filtering volume” contains a statistically significant range of eddies, it is expected that the result of filtering would be independent of the precise form of the filtering kernel (Ghosal, 1998). This argument may be in error in the near wall limit and thus, one can expect the violation of realizability by non-positive filters in that region (Fureby and Tabor, 1997).

- **Invariant Modeling Constraints**

A differential equation is said be invariant under the transformation if it leaves the equation unchanged in the transformed variables. Symmetries or invariant transformations are properties of the equations and not of the boundary conditions. In LES of turbulence not only the SGS model is constrained by symmetries, but also is the filter function (Oberlack, 1997, Ghosal, 1998). The admissible form of a filter function would be

\[
L[\mathbf{\bullet}(x)] = \frac{\alpha + 3}{4\pi l^{n+3}} \int_{R_l} |x - y|^\alpha [\mathbf{\bullet}(y)] d^3y
\]  

(2.12)

where \(R_l\) refers to a sphere with center \(x\) and radius \(l\). In deriving this form, the conditions of rotation invariance, parity invariance, Galelian invariance, scaling invariance and material frame indifference have been used (Speziale, 1985, Germano, 1986c, 1991).

- **Assumptions About Filter Kernel in Dynamic Procedure**

Carati and Eijnden (1997) evaluated the self-similarity assumption of the filter kernel for the dynamic procedure in SGS modeling. The filter can be generated using a self-similar relation for a positive kernel.

\[
G_\Delta \equiv H_\Delta * H_{\Delta/2} * H_{\Delta/4} * H_{\Delta/8} * \ldots
\]  

(2.13)
The kernel of the generator with filter width parameter equal to $\Delta$ is denoted by $H_\Delta$. Though the top-hat filter is not self-similar, it can be used as the generator to construct a self-similar filter. For positive generator kernel, relation between the filter widths of the self-similar filter and its generating kernel can be derived by analogy with probability distribution functions (PDF).

Apart from above mentioned constraints, the normalization constraint and the mesh refinement limit imply the following

$$\int_\Omega G(x, \Delta) dx = 1$$

$$\lim_\Delta \rightarrow 0 \quad G(x, \Delta) = \delta(x)$$ (2.14)

The mesh refinement limit ensures that for very small filter width, one gets back the unfiltered governing equations and hence, is consistent with the DNS limit. The normalization constraint ensures that if one filters a constant spatial field, the filtered field is the same constant value. Clearly, only a narrow class of filter kernels can admit to all of the above stated constraints. However, one should aim to retain as many desirable properties as possible for a sound foundation for a theory. For example, Gaussian filters violate the scaling symmetry, Sharp Fourier cutoff filter violates the realizability, rotation as well as scaling invariance. Classical isotropic top-hat filter admits all the symmetry requirements (Oberlack, 1997) (note: The author has a different opinion due to different interpretation of scaling invariance).

* PDE Filters

As an alternative to purely local filtering process of truncating modes is to solve a Helmholtz problem of the form

$$-\nabla^2 \bar{u} + \alpha \bar{u} = \alpha u$$ (2.15)
in $\Omega$, where $u(\mathbf{x})$ in the input function and filtered output matches the input on the boundary. Thus, if $G(\mathbf{x}, \mathbf{y})$ is taken to be the Green’s function of the problem, then solving the filtered field corresponds to the usual notion of filtering via convolution with the kernel $G$ (Mullen and Fischer, 1999). Similar ideas were presented earlier by Germano (1986a, 1986b) and Zhou et al (1989). Recently, Pantelis (1999) proposed such PDE filters to get model error for the filtered equations.

- **Commutation Errors**

  The variable width filter is essential for the inhomogeneous flows but it violates the assumption of commutation of the filtering operator and differentiation operator. This leads to extra terms in the Navier-Stokes equations corresponding to every term containing spatial derivative. Consider a filter in the form of a convolution

  \[
  \tilde{f}(x) = \int_{a}^{b} G(x - y) f(y) dy
  \]  

  (2.16)

  Using integration by parts one may show that

  \[
  \frac{d}{dx} \tilde{f}(x) = \int_{a}^{b} \frac{d}{dx} G(x - y) f(y) dy
  \]

  \[
  = G(x - a)f(a) - G(x - b)f(b) + \int_{a}^{b} G(x - y) \frac{d}{dy} f(y)dy
  \]

  \[
  = G(x - a)f(a) - G(x - b)f(b) + \frac{d}{dx} f(x)
  \]

  (2.17)

  so that the commutator is

  \[
  C[f] = \frac{d}{dx} \tilde{f}(x) - \frac{d}{dx} \tilde{f}(x) = G(x - b)f(b) - G(x - a)f(a)
  \]

  (2.18)

  This commutator goes to zero only if boundary conditions are periodic or domain is infinite and the filter kernel approaches zero sufficiently fast at infinity (Blaisdell, 1997).

  Various methodology of interpreting the filter kernel have been presented (Jordan (1999),...
Ghosal and Moin (1995)). Fureby and Tabor (1997) formulated the commutation error contributions in terms of the filter width variability and the boundary terms.

\[
[\nabla, G \star F] = \left( \frac{\partial G}{\partial \Delta} \star F \right) \text{grad}\Delta + G(x - y, \Delta)(F(y, t)n(y))
\]

(2.19)

where \( y(x) \in \partial D \) and \( n(x) \) is the outward unit normal vector to \( \partial D \). Naturally, for uniform grids, first term does not contribute at all. Moreover, the commutation error is \( O(\Delta^2) \) in terms of the filter width for the symmetric filter kernels (Note: Fureby and Tabor (1997) claimed that it is so irrespective of the filter shape, however, Vasilyev et al (1998) used the fact that the commutation errors can be represented in terms of the filter moments and the mapping function. Further, they constructed the filter kernel using Daubechies scaling function that can be chosen to have desired number of vanishing moments. Thus, the commutation error can be reduced to less than truncation errors of numerical schemes. Again, the condition of realizability and vanishing moments are in contradiction. Therefore, such filters will not be realizable (Ghosal, 1998)).

Dubois et al (1999) used the notion of projective filters in the context of dynamic multilevel methods. It was shown that the commutation error is of the order of the interpolation error on the coarse level. Jordan (1999) made use of the fact that in the transformed computational domain there will not be any contribution from grid variability and showed that the commutation errors would go to zero in the computational space if the transformed fields take zero values at the boundaries. This seems to be a very restrictive application of such filtering operation. Recently, a consistent approach using uniform Cartesian grids with adaptive mesh refinement has been proposed by Cook (1999). Although, this approach is free of commutation errors, the choice of grids seems to be highly restrictive and hence, not useful for very complicated geometry cases.
Adrian (1999) proposed to derive the LES governing equations using mean square optimal algorithms. In this approach, the mean square error during the prediction of the evolution without the knowledge of sub-grid field but with complete knowledge of the filtered field is minimized. The predicted field is spatially filtered to remove the rapidly varying harmonics. With this formulation, one can use any inhomogeneous filter explicitly, without worrying about commutation errors. Wagner and Liu (2000) used the reproducing kernel particle method (RKPM) as the LES filter to derive the governing equations and multiple scale subgrid model.

- Inverse Filtering Approach to SGS Modeling

The inverse filtering approach has provided a more consistent foundation for the SGS modeling. Various expressions for deconvoluted fields are available. The details of such defiltering/deconvoluting approaches to SGS modeling are discussed in the preceding section on the SGS modeling.

In this dissertation, the top-hat filters are used. The expressions were derived as the volume weights for the linear interpolation among the cells around grid point \((i,j,k)\). The test filtered fields for the test filter width \(2\Delta\) can be evaluated as

\[
\hat{\phi}_{i,j,k} = \frac{1}{64} \left\{ \begin{aligned}
8\bar{\phi}_{i,j,k} + 4(\bar{\phi}_{i+1,j,k} + \bar{\phi}_{i-1,j,k} + \bar{\phi}_{i,j+1,k} + \bar{\phi}_{i,j-1,k} + \bar{\phi}_{i,j,k+1} + \bar{\phi}_{i,j,k-1}) \\
+ 2(\bar{\phi}_{i+1,j-1,k} + \bar{\phi}_{i+1,j+1,k} + \bar{\phi}_{i+1,j,k-1} + \bar{\phi}_{i+1,j,k+1} + \bar{\phi}_{i,j+1,k-1} + \bar{\phi}_{i,j+1,k+1}) \\
+ \bar{\phi}_{i-1,j-1,k+1} + \bar{\phi}_{i-1,j+1,k+1} + \bar{\phi}_{i-1,j+1,k-1} + \bar{\phi}_{i-1,j+1,k+1} + \bar{\phi}_{i-1,j+1,k+1} \\
+ (\bar{\phi}_{i-1,j-1,k+1} + \bar{\phi}_{i-1,j-1,k-1} + \bar{\phi}_{i-1,j+1,k-1} + \bar{\phi}_{i-1,j+1,k+1} + \bar{\phi}_{i-1,j-1,k+1} + \bar{\phi}_{i-1,j+1,k+1})
\end{aligned} \right\}
\]

The expression of grid filtered field can be obtained similarly but it has different weights. Alternatively, Trapezoidal or Simpson’s rule can be used in each direction successively to derive these filters.
2.3 Current Status

The filtering of Navier-Stokes equations gives rise to SGS tensor. This stress tensor reflects the effect of subgrid scales on the motion of the filtered large scales. However, such a filtering procedure is subjected to several mathematical and physical constraints. Several currently used filters violate these constraints. To this end, the confusion between resolved and filtered scales has led to the use of compact filters. Therefore, one can get only approximate inverse for such filters and hence subgrid scales can only be at best, estimated. Recently, the issue of application of inhomogeneous filters for complex geometry has received a lot of attention. The lack of commutation of such filters with differentiation operator leads to terms not only from the non-linear term in Navier-Stokes equations but from all the terms containing spatial derivatives. Several strategies have been proposed to take care of these extra terms. Most of the approaches either can be applied to a limited class of flows or violate some of the constraints. However, a consistent approach using adaptive mesh refinement in Cartesian coordinates has been proposed by Cook (1999). Though for highly complex geometry, such an approach may not be practical. Adrian (1999) has reformulated the LES equations using mean square optimal algorithms such that there are no extra terms and the SGS modeling is done using all the filtered fields in the estimation of the conditional averages. Pope (2001) presented a LES methodology using projection onto local basis functions to avoid numerical errors.

In SGS modeling, near wall modeling is still the challenging part. Separation of numerical errors from modeling errors has received a lot of attention. Estimation of turbulent stresses (as obtained by ensemble averaging) from LES fields and derivation of physical boundary conditions for LES simulations are still open issues.
Chapter 3  Large Eddy Simulations with Immersed Boundary Method: Solution Procedure and Validation Studies

The solution of time-dependent three-dimensional Navier-Stokes equations is a formidable task. Closed form analytical solutions are available only for a handful of simple situations. Thus, the use of numerical methods for solving these equations is necessary. Complex turbulent flows of industrial interests are very challenging tasks even with ever-increasing computational resources. Direct numerical simulation of turbulence is prohibitively computationally intensive and generates details about the flow of unmanageable sizes. Reynolds-averaged simulations have used very ingenious modeling ideas over past several decades to simulate turbulence in some statistical sense but the success is limited to simple situations only. Large eddy simulation (LES) is an intermediate approach to turbulence simulation in which the onus of modeling of “universal” small scales is appropriately transferred to the resolution of “problem-dependent” large scales or eddies. Success of a numerical simulation strategy for turbulent flows in complex geometries depends on

i) the capability of numerical schemes to maintain their high order of accuracy/resolution,

ii) the robustness of the sub-grid scale (SGS) models or turbulence models employed

iii) the flexibility of methodology for complex moving geometries.

The first requirement leads to solving unsteady, filtered Navier-Stokes equations are solved using high-order accurate finite difference schemes on a staggered grid using a fractional step approach. The pressure Poisson equation is solved using a direct solver based on a matrix diagonalization technique. The second requirement is satisfied by employing dynamic mixed model (DMM) for SGS terms. This model can be seen as a
least common denominator to all the mathematical constraints and the physical requirements on SGS tensor. It can represent large scale anisotropy and back-scatter of energy from small-to-large scale through a scale-similar term and maintain the energy drain through an eddy viscosity term whose coefficient is allowed to change with in the computational domain. For complex moving geometries, Immersed Boundary Method (IBM) combines the efficiency inherent in using a fixed Cartesian grid to compute the fluid motion, along with the ease of tracking the immersed boundary at a set of moving Lagrangian points. Thus, the third requirement is achieved by unifying the ideas of LES with IBM.

The details and applications of various computational methods are abundantly available in literature (Fletcher (1988), Strikwerda (1989) Tannehill et al (1995), Karniadakis and Sherwin (1999)). The first step to solve any problem numerically, is the grid generation or the discretization of the solution domain such that the solution can be represented at specified locations of domain (vortex methods and meshless techniques are notable exception, Cottet and Koumoutsakos, 1999). Again, a plethora of methods are available for grid-generation and form an interesting area of research by itself (Thompson et al, 1985). The preference of structured grids over the unstructured grids is primarily due to the computational efficiency. Moreover, the structured grids can be used along with immersed boundary technique (Yosuf, 1996) to simulate complex domains. Alternatively, body-fitted grids using block grid generation is an attractive approach but it inherently leads to loss of order of accuracy of numerical schemes. Use of spectral methods or spectral elements with unstructured grids for complex domains has been shown as another promising alternative. However, the focus of the research is the
understanding the flow physics and heat transfer for the film-cooling situation and therefore, the issues of grid-generation are not dealt here. The discretization techniques such as finite differences, finite volume, finite element and spectral methods have been applied to numerous problems. Spectral methods are highly accurate but computationally more demanding. Finite element methods offer an advantage in complex domains, however, they suffer from computational overheads with unstructured grids. Finite volume methods are used extensively by industries for their ability to model complex geometry and efficient computer resource usage. High order accurate finite difference schemes are able to simulate flows successfully. Moreover, these methods are computationally very efficient. With the capability of modeling immersed boundary, these methods can simulate complex domains too. Therefore, the approach of structured grids with finite difference schemes is followed here.

3.1 Numerical Discretization Schemes

The unsteady three-dimensional Navier Stokes equations are solved using the projection method (Chorin, 1967). This is a fractional step approach in which an intermediate velocity field is calculated by neglecting the pressure gradients, and the pressure field is obtained as a solution to a Poisson equation derived using the continuity equation. This pressure field is used to update the velocity in the projection step. High order accurate finite difference schemes are able to generate accurate numerical data through DNS and LES (Rai and Moin (1991, 1993), Ghosal (1996) and Strikwerda (1999)). A representative staggered cell is shown in figure 3.1. The staggered cell arrangement is used to avoid the grid level pressure oscillations (Patankar, 1980). The temporal discretization is the second order accurate explicit Adams-Bashforth scheme. For the first
step and every hundredth time step, first order accurate forward Euler scheme is used. This eliminates the spurious computational mode from the second order scheme (GARP report). The temporal integration is performed in a time-split scheme and is explained later. The calculation of the convective terms is done by a conservative formulation. A fourth order accurate finite difference scheme is used for these terms with higher order upwinding of fluxes. The viscous dissipation terms are discretized using a fourth order accurate central difference scheme.

\[
\frac{\partial^2 u}{\partial x^2} = \frac{-u_{i-2,j,k} + 16u_{i-1,j,k} - 30u_{i,j,k} + 16u_{i+1,j,k} - u_{i+2,j,k}}{12\Delta x^2}
\]  

(3.1)

Similar expressions can be derived for derivatives in other directions for various velocity components. The order of accuracy is reduced to second order near the boundaries. The discretization of convective terms is done using upwinding. A computational cell around \(u\) velocity is shown in figure 3.2. The components \(U^*\), \(V^*\) and \(W^*\) (not shown) are the upwinded components and are evaluated using fifth order
upwinding scheme. The component $U$ is upwinded only in $Y$ and $Z$ directions only. Similar treatment is done for the remaining velocity components.

![Cell around $U_{ij}$](image)

Figure 3.2 Computational cell and arrangement of variables in X-momentum equation.

The convective terms are solved in the conservative formulation. The derivatives involving different components are evaluated using fourth order accurate centered approximation of the first order derivative.

\[
\left( \frac{\partial u v}{\partial y} \right)_{i,j,k} = \frac{u_{i,j+1,k} v_{i,j+1,k} - 27 u_{i,j,k} v_{i,j,k} + 27 u_{i,j-1,k} v_{i,j-1,k} - u_{i,j-2,k} v_{i,j-2,k}}{24\Delta x} \tag{3.2}
\]

The first derivative stencil for the same component term is upwinded such that there is one extra node on the side from where the velocity information is coming. Thus, inside the domain, there are three nodes upstream and two nodes downstream of the grid point at which the derivative is evaluated. The stencil size is decreased near the boundary points.

\[
\left( \frac{\partial uu}{\partial x} \right)_{i,j,k} = 2u_{i,j,k} \left( \frac{\partial u}{\partial x} \right)_{i,j,k} \tag{3.3}
\]

The details of time-step restriction due to CFL stability criterion can be found in Pointel (1995).
3.2 The Pressure Poisson Solver and Matrix Diagonalization Approach

Chorin (1967) proposed a fractional step scheme for solving unsteady incompressible Navier-Stokes equations. This method is also known as the projection method. It is a time-splitting scheme in which an intermediate velocity field is solved during the first step without pressure gradient terms of governing equations. Then, a pressure Poisson equation is solved to obtain the pressure field subjected to the constraint of continuity. As the last step (projection step) of algorithm, the velocity field is updated using this pressure field and the intermediate velocity field.

\[
\frac{\tilde{u} - u^n}{\Delta t} = \frac{3}{2} (C^n + D^n) - \frac{1}{2} (C^{n-1} + D^{n-1})
\]

(3.4)

where the convective terms are represented by \(C\) and the diffusion terms are represented by \(D\). In case of highly refined meshes, it may be necessary to treat some directions implicitly for diffusion terms (generally using Crank-Nicholson scheme).

\[
C = -(u \cdot \nabla)u, \quad D = \frac{1}{Re} \nabla^2 u
\]

(3.5)

To obtain the pressure Poisson equation, take the divergence of the second step and enforce the continuity condition for the velocity field at the next time step

\[
\nabla^2 p = \frac{\nabla \cdot \tilde{u}}{\Delta t} - \frac{\nabla \cdot u^{n+1}}{\Delta t}
\]

\[
\therefore \nabla \cdot u^{n+1} = 0
\]

\[
\Rightarrow \nabla^2 p = \frac{\nabla \cdot \tilde{u}}{\Delta t}
\]

(3.6)

Therefore, the Poisson equation for pressure can be solved prior to the second step in the time-split scheme. However, the solution of Poisson equation needs boundary conditions
for pressure, which are not known. Moreover, the solution of Poisson equation subjected to Neumann boundary conditions lacks the existence and uniqueness. It has a solution only if the compatibility condition is satisfied (Tafti, 1995). The discrete operators for the Laplacian are subjected such constraint.

The analysis of fraction step methods has been presented by several researchers (Armfield (1991, 1994), Perot (1993), Shen (1993), Strikwerda and Lee (1999) and Armfield and Street (1999)). The application of influence matrix approach to satisfy the correct boundary conditions has been proposed by Kleiser and Schumann (1980) (Tuckerman (1989) and Werne (1995)). The numerical solution of the Poisson equation is the most computationally demanding step of the algorithm. It would be highly desirable to have a fast, efficient and robust solver for such system of equations. With the increase in computational capability, it is possible to have fast direct methods for a large system of equations (Buzbee et al (1970,1971), McKenney et al (1995) and Greengard and Lee (1996), see Appendix V for details).

For the flows having one homogeneous or periodic direction, the spectral decomposition can be utilized effectively using FFTs. The resulting equations can be directly solved using matrix diagonalization method. Without loss of generality, treating $z$-direction as the homogeneous direction and taking the Fourier transform in that direction the discrete Poisson equation can be written as (Pointel, 1995)

\[
\left( \frac{\partial^2 p}{\partial x^2} \right)_{i,j,k} + \left( \frac{\partial^2 p}{\partial y^2} \right)_{i,j,k} + \left( \frac{\partial^2 p}{\partial z^2} \right)_{i,j,k} = g_{i,j,k}
\]

\[
\left( \frac{\partial^2 \hat{p}}{\partial x^2} \right)_{i,j,k} + \left( \frac{\partial^2 \hat{p}}{\partial y^2} \right)_{i,j,k} + f(k)\hat{p}_{i,j,k} = \hat{g}_{i,j,k}
\]
The discrete operators \( \frac{\partial^2}{\partial x^2} \) and \( \frac{\partial^2}{\partial y^2} \) are represented by the matrices \( X \) and \( Y \). The function \( f(k) \) depends on the wavenumber \( k \). The equations can thus, be written in matrix form

\[
\begin{align*}
X\hat{P}(k) + \hat{P}(k)Y + f(k)I\hat{P}(k) &= \hat{G}(k) \\
X\hat{P}(k) + \hat{P}(k)\hat{Y}'(k) &= \hat{G}(k)
\end{align*}
\] (3.8)

where, \( \hat{Y}'(k) = Y + f(k)I \). The matrix diagonalization of \( X \) and \( \hat{Y}'(k) \) can be used as follows

\[
X = P_xD_xP_x^{-1}, \quad \hat{Y}'(k) = P_x(k)D_y(k)P_y^{-1}(k)
\] (3.9)

Multiplying the Poisson equation with \( P_x^{-1} \) and \( P_y(k) \) leads to

\[
\begin{align*}
D_x\hat{P}'(k) + \hat{P}'(k)D_y(k) &= \hat{G}'(k) \\
\hat{P}'(k) &= P_x^{-1}\hat{P}(k)P_y(k) \\
\hat{G}'(k) &= P_x^{-1}\hat{G}(k)P_y(k)
\end{align*}
\] (3.10)

Now, the eigenvalues of matrices \( X \) and \( \hat{Y}'(k) \) can be used to determine the pressure field as follows

\[
\hat{P}'_{i,j}(k) = \frac{\hat{G}'_{i,j}(k)}{\lambda_{x,i} + \lambda_{y,j}(k)}
\] (3.11)

\[
\hat{P}(k) = P_x\hat{P}'(k)P_y^{-1}(k)
\]

As the last step, the inverse Fourier transform is applied to get the pressure field. In the current implementation of discrete operators, the 4-2 formulation is used, i.e. the gradient operator is the fourth order accurate centered approximation and the divergence operator is the second order accurate centered approximation in the Laplacian operator. For calculation of FFTs Compaq extended math library subroutines are used. For the calculation of eigenvalues of operators and the inverses of various matrices, subroutines available at an internet repository are used (http://www.netlib.org).
3.3 Immersed Boundary Method (IBM)

Simulation of turbulent flows in complex geometries is a daunting task. LES can formally alleviate the issue of ever-increasing resolution demand for high Reynolds number flow. However, complex geometries pose the problem of commutation errors on curvilinear grids. Moreover, the representation of moving geometries using either sliding meshes or regenerating the mesh becomes overwhelmingly complicated in complex situations. IBM relies upon the body force terms added in the momentum equations to represent the geometry on a fixed Cartesian mesh (Peskin, 1977, Yusof, 1996, Glowinski et al., 1994, Stockie 1997, Fadlun et al. 2000, Kellog, 2000). This formulation is simple and ideally suited for the moving geometries involving no-slip walls with prescribed trajectories and locations.

Figure 3.3 Identification of the circular boundary on uniform 2-D cartesian mesh and evaluation of the nearest exterior point corresponding to each identified interior point.

In the immersed boundary method, the complex geometrical features are incorporated by adding a forcing function in the governing equations. The forcing function is zero everywhere except at the surface where the influence of the solid
boundaries is assigned (Subscript \( \Gamma \)). In order to explain the concept of immersed boundary, the approximation of a circle on a two-dimensional uniform grid is illustrated. The grid points interior and exterior to the circle are identified and then paired (Figure 3.3). For internal (external) flows, the boundary condition is applied on exterior (interior) points. In fractional step approach with immersed boundary method, a body force term appears in the momentum balance. The influence of the complex geometric features is distributed on the computational mesh through these body force terms. The computed velocity field needs to be consistent with the no-slip requirement at these geometric features. As a first step, the exact location of the geometric features to be rendered is solved. Note that in general, these locations will not be coinciding with computational grid nodes. The weights can now be evaluated by interpolation to satisfy the no-slip condition on these solid walls.

\[
\frac{\tilde{u} - u^n}{\Delta t} = \frac{3}{2} \left( C^n + D^n \right) - \frac{1}{2} \left( C^{n-1} + D^{n-1} \right) + f
\]

\[
f = \left[ \frac{u^{n+1} - u^n}{\Delta t} - \frac{3}{2} \left( C^n + D^n \right) + \frac{1}{2} \left( C^{n-1} + D^{n-1} \right) \right] \delta(x - x_r)
\]

\[
\frac{u^{n+1} - \tilde{u}}{\Delta t} = -\nabla p^{n+1}
\]

where the convective terms are represented by \( C \) and the diffusion terms are represented by \( D \). In case of highly refined meshes, it may be necessary to treat some directions implicitly for diffusion terms (generally using Crank-Nicholson scheme).

\[
C = -(u \cdot \nabla)u, D = \frac{1}{\text{Re}} \nabla^2 u
\]

To obtain the pressure Poisson equation, take the divergence of the second step and enforce the continuity condition for the velocity field at the next time step.
\[
\n\frac{\nabla \cdot \vec{\tilde{u}}}{\Delta t} - \nabla \cdot f - \frac{\nabla \cdot u'''}{\Delta t} = \nabla \cdot \vec{u} \\Rightarrow \nabla \cdot \vec{u}''' = 0
\]

Therefore, the Poisson equation for pressure can be solved prior to the second step in the time-split scheme.

### 3.4 Limiting Behavior of Forcing Terms in Immersed Boundary Method

**CASE A:** Forcing at only one side of the immersed boundary (inside the virtual solid)

Let \( \Delta \) be the mesh spacing and \( \delta \) be the distance of the forcing point from the immersed surface. Therefore, we apply the linear interpolation/extrapolation among the forced point, point on the immersed surface and the point just outside the virtual solid. Let \( V_d \) be the desired velocity at the point on the immersed surface and \( V_c \) be the computed velocity in the region of interest. Therefore, the velocity at the forcing point \( V_{im} \) is given by

\[
\frac{V_c - V_{im}}{\Delta - \delta} = \frac{V_d - V_{im}}{\delta} \quad (3.15)
\]

\[
V_{im} = V_d \frac{\delta}{\Delta} - V_c \frac{\Delta - \delta}{\delta} \quad (3.16)
\]

Clearly, In the limit \( \delta \) going to zero, i.e. the forcing point approaching the point on the immersed surface, we retrieve the limit \( V_{im} \) approaching \( V_d \). However, In the limit \( \delta \) approaching mesh spacing \( \Delta \), we have \( V_c \) approaching \( V_d \). \( V_{im} \) is ill-defined because it is the difference between \( V_d \) and \( V_c \) with very large coefficients.

**CASE B:** Forcing at both sides of the immersed boundary (inside the virtual solid and at the very first point outside the virtual solid)

Let \( \Delta \) be the mesh spacing and \( \delta \) be the distance of the forcing point from the immersed surface. Therefore, we apply the linear interpolation/extrapolation among the forced
points, point on the immersed surface and the computed point just outside the virtual solid. Let $V_d$ be the desired velocity at the point on the immersed surface and $V_c$ be the computed velocity in the region of interest. Therefore, the velocity at the internal forcing point $V_{int}$ is given by

$$\frac{V_c - V_d}{2\Delta - \delta} = \frac{V_d - V_{int}}{\delta} \quad (3.17)$$

$$V_{int} = \frac{V_d}{2\Delta}/\left[2\Delta - \delta\right] - \frac{V_c}{\delta}/\left[2\Delta - \delta\right] \quad (3.18)$$

Similarly, the velocity at the external forcing point $V_{ext}$ is given by

$$\frac{V_c - V_d}{2\Delta - \delta} = \frac{V_{ext} - V_d}{\Delta - \delta} \quad (3.19)$$

$$V_{ext} = \frac{V_d}{\Delta}/\left[2\Delta - \delta\right] + \frac{V_c}{\delta}/\left[2\Delta - \delta\right] \quad (3.20)$$

Clearly, in the limit $\delta$ going to zero, i.e. the internal forcing point approaching the point on the immersed surface, we retrieve the limit $V_{int}$ approaching $V_d$ and $V_{ext}$ approaching $[V_d + V_c]/2$. Moreover, in the limit $\delta$ approaching mesh spacing $\Delta$, we have $V_{ext}$ approaching $V_d$. $V_{int}$ is defined as $[2V_d - V_c]$ as it should be by reflection condition. Thus, the forcing remains physical for all positions of the immersed surface between the grid interfaces. For moving boundary implementation, it will be important for another reason. It avoids reflected velocities to show up in the region of interest.

**CASE C:** Forcing at both sides of the immersed boundary to enforce quadratic dependence on wall normal distance to satisfy continuity equation around immersed cells

Consider an immersed point between grid points $J$ and $J+1$. The distance of point $J+1$ from the immersed point is $\delta$ and the grid resolution is $\Delta$. To fit a quadratic profile between these points to evaluate the immersed point forcing, consider the following expressions.

$$V = ay^2 + by + c$$
@ y = 0, V = V_J

@ y = Δ-δ, V = V_S and ∂V/∂y = 0 (simplified continuity equation on the wall)

@ y = Δ, V = V_{J+1}

Therefore, we have following equations

2a(Δ-δ) + b = 0

V_S = a(Δ-δ)^2 + b(Δ-δ) + V_J

V_{J+1} = aΔ^2 + bΔ + V_J

Solving for V_J, we’ll get

V_J = V_{J+1}[(Δ-δ)^2/δ^2] + V_S[Δ(2δ-Δ)/δ^2] \quad (3.21)

Checking the limiting behavior for different locations of immersed point between the grid points

I: The forced point is approaching the solid wall (δ→Δ)

V_J→V_S

II: Immersed point is in the center of the grid cell (δ→Δ/2)

V_J→V_{J+1} (symmetry condition is satisfied naturally)

III: Immersed point is overlapping with the solved grid point (δ→0, thus V_{J+1}→V_S)

V_J→V_{J+1} (or V_S)

Thus, the expression written above satisfies all the limiting behaviors of immersed point location between the grid points for the quadratic profile satisfying the simplified form of continuity equation using only two-point stencil.

Application of LES-IBM in trapped vortex combustor (TVC) demonstrates the usefulness of single-sided forcing. However, for moving geometry of stator-rotor interaction, two-sided implementation is used.
In summary for solution procedure, the governing equations are obtained by applying a filtering operation on the Navier-Stokes equations and the continuity equation. The filter function is represented as the convolution operator (Ghosal and Moin, 1995). The unfiltered fields give rise to subgrid scale stresses that require modeling. Following the dynamic mixed method of Zang et al (1993), these stresses are decomposed into a resolved part and an unresolved part (Appendix II). The resolved part is the Galilean invariant form of Bardina’s (1983) scale similarity model (Speziale, 1985). The dynamic Smagorinsky model is used for the unresolved part of the stress and the dynamic coefficient is test filtered to avoid numerical instabilities. The issue of filtering is very important for such numerically accurate simulations. On a non-uniform grid the filtering operator becomes a function of spatial location and hence gives rise to commutation error leading to low order of accuracy even with very high order accurate schemes (Ghosal and Moin, 1995). The issue of dependence of dynamic coefficient on aspect ratio of the grid cells when using grid based filters needs to be addressed on such non-uniform grids (Scotti et al, 1997). These issues are evaded here by using uniform grids with isotropic aspect ratio. In particular, the flow contains structures in almost all parts of the computational domain and hence, all parts of the domain need to be resolved with equal importance. The top hat filter is used for following reasons (Zhou et al, 1989 and Ghosal, 1999). It is easy to implement in a finite difference code. It has compact support, unlike Gaussian or exponential filters which violate this requirement for grid-based filtering. It is a positive or realizable filter i.e. if it filters a non-negative field, the filtered field is always non-negative.
3.5 Validation Studies

CASE 1 Laminar plane channel (Validation for sink term formulation):

To check the formulation of unsteady heat transfer equations in periodic geometries, a solution for the laminar case is obtained for the Poiseuille flow in the channel. The computed Nusselt number was 7.50 as compared to the theoretical value of 7.54.

CASE 2 Laminar flow past circular heated cylinder in a plane channel (Validation of Immersed Boundary Method):

Several parameters were calculated and compared to validate the implementation of immersed boundary method. The separation points at (80º-82º) are also observed.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Computed</th>
<th>Theoretical / Experimental</th>
</tr>
</thead>
<tbody>
<tr>
<td>C_Dp Pressure Drag</td>
<td>0.620</td>
<td>1.2 (Total drag)</td>
</tr>
<tr>
<td>C_Df Friction Drag</td>
<td>0.593</td>
<td>5.21 (±20%)</td>
</tr>
<tr>
<td>Nusselt Number</td>
<td>5.45</td>
<td>5.21 (-20%)</td>
</tr>
<tr>
<td>Strouhal Number</td>
<td>0.283</td>
<td>0.281-0.287</td>
</tr>
</tbody>
</table>

![Figure 3.4 Separation points in the flow over circular cylinder (80º-82º)](image1)

![Figure 3.5 Von-Karman street pattern (Animation)](image2)
CASE 3 Lid driven cavity flow (Validation of LES procedure and SGS model):

In lid-driven cavity with spanwise aspect ratio of 0.5 at Reynolds number of 10000, the flow is partly laminar in the cavity and strongly turbulent on the downstream and bottom walls of the cavity. Therefore, it forms a good benchmark case to validate LES procedure as well as SGS model. Deshpande and Milton (1998) and Leriche and Gavrilakis (2000) performed DNS of lid-driven cavity flows. Zang (1993) performed LES of the same configuration. Computations show good agreement with the experimental data of Prasad and Koseff (1989).

Figure 3.6 Comparison of LES results with experimental data of Prasad and Koseff (1989) on the centerplane of lid-driven cavity with spanwise aspect ratio of 0.5 at Reynolds number 10000 a) Mean velocity components, b) $rms$ components of the fluctuating velocity field and c) turbulent shear stress $u'v'$ along X and Y centerlines.
CASE 4 Immersed boundary method for moving complex geometries (Unsteady stator-rotor interactions):

Inherent unsteadiness of a turbomachinery flow field created by relative motion between stationary blades (stator) and the rotating blades (rotor), requires the designer to account for three-dimensional as well as unsteady effects. The unsteadiness is caused by (a) the interaction of the rotor airfoils with the wakes and passage vortices generated by upstream airfoils, (b) the relative motion of the rotors with respect to the stators (potential effect), and (c) the shedding of vortices by the airfoils because of the blunt trailing edges (Rai and Madavan, 1990, Saxer and Giles, 1994). Computation of such flows is complicated by relative motion between rotor and stator airfoils and the periodic transition of the flow from laminar to turbulent. Unsteady simulations have been performed using multitudes of approximations such as “mixing-plane” approach, “average passage” approach and unsteady RANS (Denton, 1990, Adamczyk, 1985, Sharma, Ni and Tanrikut, 1994). In “mixing-plane” approach, flow through each airfoil row in the machine is calculated for a specified circumferentially uniform inlet and an average exit boundary conditions. The effect of periodic unsteadiness is not accounted for in this approach. In “average passage” approach, the effects of adjacent airfoil rows are accounted for through the use of body forces and “apparent stresses”. Reliable models are not yet available to account for circumferential variation of “apparent stresses” (Lakshminarayana, 1995). Unsteady RANS has some potential to resolve periodic unsteadiness and can yield significantly better results. However, the modeling of energy spectrum is inaccurate if there is significant turbulent energy in other secondary flows through the airfoil blade row. Phase-lagging method (Erdos et al, 1977) is generally used
to model blade rows with unequal airfoil counts. With this procedure, the solution domain for a given row need only span one pitch rather than multiple pitches as is required for spatial periodicity. The solution at a point outside the periodic boundary is derived from the solution one pitch away at an earlier time step. The temporal period needed to retrieve the data from the time-storage arrays is a simple function of airfoil count and rotor speed. This requires storing the time history of dependent variables at points adjacent to periodic boundaries (Kelecy et al, 1995). These calculations have been performed invariably using “sliding mesh” techniques requiring further constraints on matching the interface conditions on different fluxes (all of these are not usually satisfied). In the present study, we utilize LES with moving IBM to simulate unsteady stator-rotor interactions. Though, the calculation is performed for incompressible fluid at a low Reynolds number, it demonstrates the strength of the method by avoiding all ad-hoc assumptions pertaining to RANS modeling and sliding meshes.

The geometry of the airfoils is taken from the numerical study of Kelecy et al (1995). The airfoil profile is approximated by the cubic spline surfaces. The airfoil is divided into leading edge, trailing edge, pressure surface and suction surface to ensure that immersed boundary conditions are enforced on enough grid points to realize the geometry (Interpolation around immersed points is done according to Case B of limiting behavior, as discussed earlier in this chapter). A uniform Cartesian grid of $302 \times 202 \times 11$ points is used for a domain of the size $3D \times 1D \times 0.1D$, where $D$ is the chord length of the rotor airfoil. Choice of a small spanwise dimension may not allow larger physical scales and hence may not be desirable. The details of computational stencils and interpolation weights are shown for the suction and pressure immersed surfaces. Uniform flow field is
specified at the inflow. Periodic boundary conditions are applied in the direction of rotor motion (y) and the spanwise (z) direction. Reynolds number based on the inflow velocity and rotor chord length is 5000. It must be kept in mind that this numerical simulation is performed to demonstrate the capability of LES-IBM for a very complex problem and parameters chosen for this study may not be representative of true physical problem. At the outflow, a non-reflective convective scheme is applied to convect away the flow structures out of the computational domain without any spurious reflections. The wave speed is calculated to maintain the mass flux balance in the whole domain.

Details of immersed boundary point interpolation stencils are shown around the suction side and pressure side of the rotor blade in figures 3.7 and 3.8 respectively. An indicator function is used to identify the stencil around these surfaces.

**Suction Side:**

\[
f = (y_j - y_s) \times (y_s - y_{j-1})
\]

\[
\begin{align*}
(V_j - V_s) &= (V_{j+1} - V_s) \\
(y_j - y_s) &= (y_{j+1} - y_s) \\
(V_s - V_{j-1}) &= (V_{j+1} - V_s) \\
(y_s - y_{j-1}) &= (y_{j+1} - y_s)
\end{align*}
\]

\[
\delta_1 = \frac{(y_j - y_s)}{(y_{j+1} - y_s)}, \quad \delta_2 = \frac{(y_s - y_{j-1})}{(y_{j+1} - y_s)}
\]

\[
V_j = (1 - \delta_1)V_s + \delta_1V_{j+1}
\]

\[
V_{j-1} = (1 + \delta_2)V_s - \delta_2V_{j+1}
\]

Figure 3.7 Computational stencils near the immersed boundary (suction side surface) of moving blade.
Pressure Side:

\[ f = (y_{j+1} - y_p) \times (y_p - y_j) \]

\[ \delta_1 = \frac{y_p - y_j}{y_p - y_{j-1}}, \delta_2 = \frac{y_{j+1} - y_p}{y_p - y_{j-1}} \]

\[ V_j = (1 - \delta_1)V_p + \delta_1 V_{j-1} \]

\[ V_{j+1} = (1 + \delta_2)V_p - \delta_2 V_{j-1} \]

Figure 3.8 Computational stencils near the immersed boundary (pressure side surface) of moving blade.

The animations of instantaneous vorticity field and pressure field are shown in figure 3.9(a-b) respectively. The low pressure levels correspond to the vortex-cores. The development of boundary layer vorticity on the solid surfaces and its subsequent shedding into the main crossflow near the trailing edge of the stator blade produces a mixing layer type wake. There is a separation region on the suction side of the stator. Note that such a recirculation produces boundary layer vorticity in opposite sense and is captured accurately here. The trailing edge vortices of the stator blade impact on the suction side of the rotor blade near its leading edge. The trailing edge vortices of the rotor and the vortices formed due to the interaction of stator wake and suction-side boundary layer are shed into the passage flow and convected out of the domain.
3.6 Concluding Remarks

The merit of immersed boundary method (IBM) for simulating moving complex geometries on Cartesian mesh has been demonstrated. High order of accuracy of discretization schemes is retained which is very important for LES. In stator-rotor case study, the superiority of this method is demonstrated over existing methodologies such as sliding meshes (no sliding interface implies all the fluxes are satisfying the governing equations). Moreover, the ad-hoc modeling for the “apparent stresses” is not needed in the realms of LES. In future, a zonal refinement treatment of the immersed boundaries will be implemented to capture the essential near wall physics to render this method with predictive capabilities.

Figure 3.9 Animations of vorticity component and pressure.
Computation of turbulent flows is very demanding and challenging. However, the tools for the analysis of such fields are even more demanding. The extraction of coherent structures from the turbulent flow fields, calculation of the single-point/multi-point correlations in the closure equations, development of low-dimensional models, local analysis in physical as well as spectral space using wavelets and investigation of the p.d.f.s of various turbulent variables are few of the analysis steps one might take to explain the flow physics and model the turbulent flows (Lumley, 1970, Farge and Guyon 1995, Farge et. al., 1999). A multitude of tools is needed depending on the objective of the analysis. In this research work, proper orthogonal decomposition (POD) is used to demonstrate the low-dimensionality of the turbulent fields. A simple criterion (Q>0) is used for extracting coherent structures from the time-dependent three-dimensional flow fields. A simple diagnostic indicator function (scalar dissipation rate) is used to explain the passive scalar macro-mixing.

4.1 **Proper Orthogonal Decomposition (POD)**

The rationale behind low-order modeling is to capture essential dynamics of the fully resolved system by a very small number of degrees of freedom. Lorenz (1972) constructed low order models representing realization of turbulence. Lumley (1970) introduced the concept of proper orthogonal decomposition (POD) as the optimal basis to capture the energy of system in least number of modes (Holmes et. al., 1996). Similar idea in the area of signal processing is called Karhunen-Loeve (KL) decomposition and in the area of Matrix analysis is called Singular Value Decomposition (SVD) (Heiland, 1992). If the spatio-temporal data be given as the snapshots (assuming that the time-
spacing between successive snapshots is large enough for the snapshots to be uncorrelated), then the auto-correlation tensor can be written as a convergent series (Sirovich, 1987).

\[ C(x, x') = \langle u(x) u(x') \rangle \quad \text{Autocorrelation tensor} \]

\[ u^{(n)}(x) = u(x, n\Delta t) \quad \text{Snapshot} \quad (4.1) \]

\[ C(x, x') = \lim_{M \to \infty} \frac{1}{M} \sum_{n=1}^{M} u^{(n)}(x)u^{(n)}(x') \]

Clearly, using the snapshot assumption the size of the problem can be reduced dramatically. In fact for \( N \) degrees of freedom system, the auto-correlation tensor is \( N \times N \), while with \( M \) snapshots, the size will be \( M \times M \) (assumption: \( M << N \)). It can be shown that the eigenvectors of this approximate auto-correlation tensor are the POD eigen basis functions and satisfy the optimality of approximation in some “energy” sense. In other words, these functions form a basis that spans all the snapshots in such a fashion that, for a given number of modes, this basis contains most “energy” of the field (Ravindran, 2000).

\[ E = \left\{ U^{(i)} : 1 \leq i \leq N \right\} \quad \text{Ensemble of snapshots} \]

\[ P = \sum_{i=1}^{N} w_i U^{(i)} \quad \text{Linear Superposition of snapshots} \]

\[ \left( w_i : \text{Maximization of "energy" in projected field} \right) \quad (4.2) \]

\[ C_{i,j} = \frac{1}{N} \int_{\Omega} U^{(i)}(x)U^{(j)}(x)dx \]

\[ \text{Solve:} \quad CW = \lambda W \]

A Galerkin projection of Navier-Stokes equations on this basis will yield a Low-order system displaying the similar dynamics as a fully resolved system (Cazemier et. al., 1994).
4.2 Coherent Structures Identification

“A coherent structure is defined as a flow module with instantaneous space-correlated vorticity” (Schoppa and Hussain, 1996). However, some intricate details of the notion about coherent structures still vary. Therefore, there is a multitude of analysis tools for coherent structure eduction from turbulent flow fields. Techniques like Pattern Recognition Approach (PRA) are biased with the a-priori definition used for coherent structure. Linear Stochastic Estimation (LSE) (Adrian, 1996) and Proper Orthogonal Decomposition (POD) (Lumley, 1970) have been developed to address the unbiased extraction of coherent structures (Bonnet, 1996). Several algorithms have been presented based on the invariants of the flow fields (Wray and Hunt, 1989). Chong et. al. (1989) used critical points in terms of invariants ($P, Q$ and $R$) of the velocity gradient tensor to analyze the geometry and topology of complex flow patterns.

\[ u_{i,j} = S_{ij} + \Omega_{ij} \quad S_{ij} = \frac{1}{2} \left( u_{i,j} + u_{j,i} \right) \quad \Omega_{ij} = \frac{1}{2} \left( u_{i,j} - u_{j,i} \right) \]

\[ P = \text{Trace}(u_{i,j}) = S_{ii} \quad [-0, \text{ Incompressible fluid}] \]

\[ Q = -\frac{1}{2} \left( S_{ij} S_{ji} + \Omega_{ij} \Omega_{ji} \right) = -\frac{1}{2} \left( S_{ij} S_{ji} - \frac{1}{2} \Omega_{k} \Omega_{k} \right) = \frac{1}{2} \nabla^2 p \]

\[ R = \text{Det}(u_{i,j}) = \frac{1}{3} \left( S_{ij} S_{jk} S_{ki} + 3 \Omega_{ij} \Omega_{jk} S_{ki} \right) \]

Levy et. al. (1990) used helicity density and normalized helicity to identify and accentuate the concentrated vortices, differentiate between primary and secondary vortices and mark their separation and reattachment lines. Tanaka and Kida (1993) showed that regions of high pressure Laplacian and of high strain rate represent the tubelike and sheetlike structures respectively. Singer and Banks (1994) presented a predictor-corrector scheme for vortex identification. Ma et. al. (1995) used tracing of
cloud of particles by transforming the Eulerian vector fields to Lagrangian frame and computed the statistical dispersion of particles about the mean position with an auxiliary equation. Jeong and Hussain (1995) defined a vortex core as a connected spatial region in which the tensor $\Omega_{ik}\Omega_{kj} + S_{ik}S_{kj}$ has a negative second eigenvalue ($\lambda_2 < 0$). Kida and Miura (1998) proposed a sectional-swirl and pressure minimum (SSPM) method to decrease the datasize dramatically by using skeleton representation of vortices and avoiding isosurface storage/rendering. Cucitore et. al. (1999) developed a non-local criterion to measure the tendency of two particles in the flow to remain near each other. Dubief and Delcayre (2000) demonstrated the usefulness of positive isosurfaces of pressure Laplacian ($p_{,kk}$) to identify coherent structures in a variety of lows. In this research work, pressure Laplacian (also referred to as $Q > 0$ criterion) is used for flow feature extraction and the temporal evolution of these iso-surfaces are analyzed to understand the coherent structures dynamics in various flows.

4.3 Mixing

Mixing is ubiquitous and intuitively obvious phenomenon. However, quantitative and universal measures of this phenomenon are lacking. Mixing is intimately related to stretching and folding of material surfaces (Ottino, 1989,1990). Laplacian of pressure field ($p_{,kk} = -\nabla^2$) which was used as the indicator of the coherent vortical structures, can yield information on the mixing. The ratio of the long-time average of the real and imaginary parts of $r$ along a trajectory gives a simple measure of the strain-rotation ratio experienced by particle orbits (Tabor and Klapper, 1995). “Mixing can be thought of as a consequence of barrier destruction, and transport across partially destroyed barriers can be studied in a lobe-dynamic context, providing a basic measure of mixing” (Beigie et.
al., 1995). Usually, the geometry of isoscalar surfaces in turbulent flows is more complex than (constant-D) fractal. Their description requires an extension of the original, scale-invariant, fractal framework that can be cast in terms of a variable (scale-dependent) coverage dimension. (Dimotakis and Catrakis, 1995). Geometry of chaotic mixing can also be studied using Lyapunov exponents and curvature (Thiffeault, 2000). The mixing of passive scalar $c(x,t)$ can be quantified in terms of scalar energy dissipation rate field $(ReSc)^{-1}(c(x,t,k) \cdot c(x,t,k))$ (Southerland et. al., 1995). In our research work, we will use the scalar dissipation rate as the measure of mixing. Other simple measures like ratio of the r.m.s. fluctuations of passive scalar concentration with the mean level $(\overline{c'^2}/\overline{c^2})$ can be used to quantify the degree of mixing (Hinch, 1995). This simple measure can be derived as increase in properly defined entropy for dilute mixtures.
In order to achieve higher thermal efficiency and specific thrust, the gas turbines are operated at higher turbine inlet temperature. The direct consequence of this increased heat load on the turbine blades is reduced life and higher operational cost. To obtain the real benefits of increased thermal efficiency and power output, the life of turbine blades must be maintained. There has been significant research in developing materials to withstand extremely high thermal loads. Examples of advanced materials include nickel superalloys and various ceramics. Large local variations of the heat transfer coefficient can produce significant temperature gradients in the wall of a turbine blade. These gradients may cause high thermal stresses and ultimately could lead to blade failure. Moreover, ceramics do not withstand the mechanical loads of rotating machinery. Thus, the efficient cooling of the turbine blades (made from high temperature superalloys) below their melting point is a necessary step in the design process of gas turbines. One such cooling strategy is discrete hole film cooling.

The schematic of film cooling concept is presented in figure 5.1. The heat load on the blade surface without film cooling is represented by the heat flux $q''_0 = h_0(T_m - T_w)$, where $h_0$ is the heat transfer coefficient on the surface with a wall temperature $T_w$ and oncoming hot mainstream gas temperature $T_m$. The heat load changes upon the injection of coolant over the blade surface and is represented as $q'' = h(T_f - T_w)$ where, $h$ is the heat transfer coefficient on the surface with film cooling and $T_f$ is the film temperature (that is a mixture of hot mainstream and the coolant jet fluid). The film effectiveness is defined as $\eta = (T_m - T_f)/(T_m - T_c)$. Thus, the ratio of heat loads can be obtained as $(q'' / q''_0) = (h/h_0)[(T_f - T_w)/(T_m - T_w)] = (h/h_0)[1-\eta (T_m - T_c)/(T_m - T_w)]$. To obtain any benefit
from film cooling, the heat load ratio should be less than 1. The heat transfer coefficient ratio \( \frac{h}{h_0} \) is enhanced due to turbulent mixing of the jets with the mainstream and is normally greater than 1. Thus, the temperature ratio \( \frac{(T_f - T_w)}{(T_m - T_w)} \) should be much less than 1.0 to decrease the heat load with film cooling. The aim is to use the minimum amount of coolant through an optimal number of discrete holes with different geometry at specific locations of blade surface while maintaining the heat load ratio less than 1.0.

![Figure 5.1. The schematic of film cooling of gas turbine blade](image)

The essential features of such film-cooling flows are present in a more generic flow situation of jets-in-crossflow. Jets-in-crossflow are encountered for V/STOL applications, pollutant dispersion in the rivers and atmosphere and film-cooling of gas turbines. The characteristics of the jet and the crossflow are different for different situations; however, the large scale flow features determined by the geometry and boundary conditions are similar. This complex flow field is inherently characterized by unsteady vortices. The kidney shaped counter-rotating vortex pair (CVP) is formed due to the reorientation of jet-hole boundary layer vorticity by the crossflow. A horseshoe or necklace vortex system is formed in front of the jet due to the stagnation of crossflow there. The shear layer vortices are formed at the freestream and jet interface due to Kelvin-Helmholtz type instability (a different reason is provided by Blanchard et al
The unsteady vertical wake vortices are shed behind the jet and these vortices are formed by the entrainment of the crossflow boundary layer and/or the horseshoe vortex and their reorientation in the wake. These vortices are present in most the above mentioned applications. A robust turbulence model designed to predict film-cooling flow under the influence of a large number of parameters, should be, therefore, derived and calibrated for the general case of jets-in-crossflow.

There have been a number of experimental studies for the film cooling problem. A detailed review of research in film cooling prior to 1971 is presented in Goldstein (1971). Recent development in turbine blade film cooling is presented in Han and Ekkad (1998). There are several parameters that influence the film effectiveness and heat load on turbine blades. The freestream turbulence intensity and the energy containing turbulent scales influence the freestream and coolant jet mixing and hence the heat transfer to the blade surface. The film cooling effectiveness is strongly dependent on the discharge rate of coolant fluid, which in turn depends on many geometrical and aerodynamic parameters, such as hole geometry, pressure ratio, velocity and density ratio of coolant jet to mainstream. Camci and Arts (1990) studied the effect of coolant to freestream mass ratio and temperature ratio on heat transfer around the leading edge of the gas turbine rotor blade. Karni and Goldstein (1990) used the heat/mass transfer analogy to measure the adiabatic wall heat transfer coefficient by naphthalene sublimation technique. Ou et al (1990) and Jumper et al (1991) examined film cooling effectiveness in high turbulence flows. Teekaram et al (1991) studied the film cooling in the presence of mainstream pressure gradients. The influence of curvature on film cooling performance was studied by Schwarz et al (1990). The effect of mainstream
acceleration on the heat transfer coefficient was studied by Ammari et al (1990). Sinha et al (1991) measured the film cooling effectiveness with variable density ratio. The effect of mainflow unsteadiness caused by upstream stator vanes on film cooling effectiveness was studied by Bons et al (1995). Bons et al (1996) studied the effect of high freestream turbulence on film cooling effectiveness. The results showed reduction in film cooling effectiveness in the region directly downstream of injection hole. However, there was an increase in film cooling effectiveness in the region between the injection holes due to enhanced spanwise diffusion of coolant fluid. Kohli and Bogard (1997) conducted experiments using jet-grids for very high freestream turbulence levels and measured the temperature pdf distribution in the flow field. Several studies have been done using PIV measurements of flat plate film cooling flows with high freestream turbulence (Gogineni et al (1996) and Rivir et al (1997)). Gristch et al (1998) derived empirical approach to correlate discharge coefficients over a broad range of engine-like operating conditions. Burd et al (1998) presented the measurements in film cooling flows for several hole length to diameter ratios (L/D) under different freestream turbulence conditions. Jones (1999) studied the use of foreign gas in simulation of film cooling density difference effects. Mee et al (1999) used thermochromic liquid crystals (TLCs) to measure the temperature field downstream of leading-edge film-cooling holes. Kaszeta and Simon (1999) measured the eddy diffusivity of momentum experimentally and presented the details of anisotropy of turbulent transport in the film cooling flow. The influence of mainstream flow history on film cooling and heat transfer from two rows of simple and compound angle holes in combination was studied experimentally by Maiteh and Jubran (1999).

Numerical capabilities have increased by several folds during the last decade. A numerical study has the advantage of eliminating many of the uncontrolled factors in experiments and allowing precise control of initial and boundary conditions. Earlier RANS studies done by Patankar et al (1977), Demuren (1983) and Sykes et al (1986) used turbulence models of varying complexity. Karagozian (1986) developed a two-dimensional model describing the contra-rotating vortex pair trajectory. Coelho and Hunt (1989) derived an inviscid three-dimensional vortex-sheet model for the near field of a strong jet in crossflow. Kim and Benson (1992) used a multiple time scale turbulence model for numerical calculations of a three-dimensional jet in crossflow. O'Connor and Haji-Sheikh (1992) studied film-cooling in supersonic flow using 2-D calculation
transfer using a turbulence model. Chen et al (1999) performed a numerical study of
discrete-hole film cooling using near-wall second moment turbulence closure. Takata et
al (1999) presented the hybrid LES-RANS computations of film cooling flow on gas
turbine blade. Acharya et al (2000) summarized the numerical studies performed on jets-
in-crossflow using DNS, LES and RANS for the film cooling applications. In general, the
RANS calculations under-predict the lateral spread and mixing of the jet while they over-
predict the vertical penetration of the coolant jet.

Figure 5.2 Transverse shear stress \( u' w' \) along Z/D = -0.5. RANS vs LES predictions.
The experimental data is from Ajersch et al. (1995).

In figure 5.2, predictions of the transverse shear stress component \( u' w' \) that is
responsible for the lateral spread are shown for Reynolds Averaged Navier Stokes
(RANS) calculations using two-equation turbulence model, second moment closure
model and LES studies for a square jet injected normally into the crossflow (Tyagi and
Acharya, 1999). Clearly, RANS modeling either at two-equation level or at the second
moment level is inaccurate for this stress component while LES can capture the flow
physics well.

Large eddy simulation technique is applied to the following problems of interest related
to external cooling of gas turbine blades (film-cooling or jets-in-crossflow).
- Effect of hole aspect ratio on the coherent structures and the turbulent stresses in the jets-in-crossflow is studied (Haven, 1996). Three different hole aspect ratios 0.5, 1.0 and 2.0 are analyzed.

- Effect of coolant jet injection angle on the various flow structures and their evolution is studied (Ajersch et al, 1995 and Findlay et al, 1996). Normal jet injection and inclined jet injection at 30° are simulated.

- Effect of freestream turbulence intensity levels on the flow structures and the turbulent stresses is studied. The freestream turbulence is assumed to correspond to the grid generated turbulence and is simulated using Gaussian pdfs. Turbulence intensity levels of 2% and 15% are studied.

- Effect of freestream turbulence length scales on the coolant-crossflow mixing and the dynamics of various flow structures is studied. The turbulent energy is maintained at 15% while the energy spectrum is altered. In the small scale case, the turbulence is generated using the Gaussian pdf. For the large scale case, the Von-Karman energy spectrum is used in the inertial range with a peak at the prescribed length scale (4D) corresponding to the low wave number regime.

- Film-cooling calculations are performed for UTRC experimental setup for two different blowing ratios (M = 0.5 and 1.0). The coolant jet issues out from a cylindrical delivery tube into the mainflow at an inclination angle of 35°. The Reynolds numbers based on the jet velocity and the diameter of the delivery tube are approximately 11100 (for M = 0.5) and 22200 (for M = 1.0).
5.1 Large Eddy Simulations of Rectangular Jets in Crossflow: Effect of Hole Aspect Ratio

The hole geometry is an important parameter that controls the development of the flow structures and the penetration and spreading of the jet (Haven (1996), Haven and Kurosaka (1997)). The vertical and lateral spreading of the jet, in turn, govern the film cooling effectiveness. This was studied by Haven (1996) using PIV technique that has limited information about the spatial and temporal characteristics of the flow. The objective of this study is to numerically investigate the effect of hole aspect ratio on the dispersion of film cooling jet. The numerical study will be done through time and space accurate simulations of the filtered Navier Stokes equations (Large Eddy Simulations). Although the focus of the study is on the flow field, a qualitative idea of the heat transfer can be obtained by examining the dispersion of the jet in the crossflow and by the shear stress distribution at the wall.

5.1.1 Problem Description

A schematic of the physical problem studied is shown in Figure 5.3. Three hole aspect ratios \( L/D = 0.5, 1.0 \) and \( 2.0 \) are investigated and are designated as hole A, B and C respectively. A uniform Cartesian grid of \( 122 \times 52 \times 32 \) points is used for a domain of \( 12D \times 5D \times 3D \) (Figure 5.3). At the inflow, a fully developed turbulent boundary layer profile is prescribed. The velocity field is specified at the jet inlet from the experiments of Ajersch et al (1995). The hole B case is the numerical simulation for the experiments of Ajersch et al (1995). The Reynolds number based on the jet velocity and the hole spanwise dimension is 4700. A periodic boundary condition is applied in the spanwise direction. The domain size is chosen such that free-stream conditions at the inlet can be used as the boundary conditions along the top plane. At the outflow, a convective boundary condition
is used where the wave speed is determined from a flux balance. Inlet planes are placed at $X/D = -3.25$ for hole A, $X/D = -3.5$ for hole B and $X/D = -3.0$ for hole C.

![Diagram of computational domain with labeled holes](image)

Figure 5.3 Schematic of the computational domain.

### 5.1.2 Results

Results are presented for three hole aspect ratio cases at various planes of the computational domain. The X-component of the mean velocity field and the root mean square fluctuations are presented in figure 5.4. The velocity profiles show a double peaked profile for all the holes; however, the location and magnitude of the peaks are different indicating that size and strength of flow structures obtained for three cases are different. For holes B and C, negative values of the meanflow profile at downstream locations represent large recirculation regions. The $u_{rms}$ profile at $X/D = 1.0$ shows that hole A has a maximum close to the wall which clearly indicates that for this case the vertical jet penetration is the smallest, and that the jet trajectory is closest to the wall. At
downstream planes, the location of $u_{rms}$ maximum shifts towards the leeward edge of the bent jet. For the case of hole B and C, there is a local maximum in $u_{rms}$ profile at downstream planes between the wall and the leeward edge of the jet indicating enhanced mixing.

Figure 5.4: Mean X component and $u_{rms}$ profiles at X/D = 1.0, 3.0 and 5.0 and Z/D = 0.0.

Presence of local maxima or minima in the spanwise direction for the streamwise component of velocity would tilt the $\omega_z$ into $\omega_x$. The local maximum in spanwise direction for the u-velocity component is designated as ‘concave warping’ and the local minimum is designated as ‘convex warping’ by Haven and Kurosaka (1997). In the instantaneous contours of $\omega_x$ at X/D = 1.0 (Figure 5.5), we observe anti-kidney vorticity over the CVP for hole A. It is also noted that the u-profiles at the leading and trailing edges of the holes warp in similar fashion as observed by Haven (1996). The local maximum around jet center plane in main flow profile along the spanwise direction will tilt the leading edge spanwise vorticity into anti-kidney pair. However, the warping of profile for hole A is not to the extent as observed by Haven (1996). This might be due to prescription of time averaged jet exit profile and higher jet Reynolds number as
compared to Haven (1996). The local minimum around the jet center plane would yield a kidney pair above the CVP that is formed by sidewall vorticity of holes.

Figure 5.5 Instantaneous velocity vectors and contours of $\omega_x$ at $X/D = 1.0$, Profiles of X-component of velocity at various Z-planes across the holes at $Y/D = 0.4$ to show the warping of mainflow.

One interesting observation is made about the relative strengths of horseshoe vortex and CVP for various cases. For hole A, the horseshoe vortex has higher levels of $\omega_x$ as compared to CVP. The velocity field induced by these vortices at the location over the CVP may result in the vorticity in the opposite sense of CVP. For hole C, the CVP is stronger than horseshoe vortex. The resultant induced velocity field would still have the same sense of vorticity as CVP except it peels off at a location somewhere above the CVP due to opposing nature of horseshoe vortex. At the trailing edge of hole the warping of reverse mainflow in the wake region gives rise to anti-kidney vorticity between the CVP (figure 5.5).
Figure 5.6 Time averaged velocity vectors and contours of $\omega_x$ at planes X/D = 1.0, 3.0, 5.0 and 7.0.

Contours of time averaged velocity vectors and $\omega_x$ for various cases are shown at X/D = 1.0, 3.0, 5.0 and 7.0 in figure 5.6. The asymmetry in the time-averaged results is attributed to the asymmetric boundary condition at the jet exit prescribed from the experimental data of Ajersch et al (1995). The jet penetration is least for hole A. The CVP weakens for this case more rapidly as compared to other cases. The time-averaged contours show the breakup of CVP into multi-decked CVP at downstream planes for hole B. The jet penetration is highest for hole C at all planes and the CVP degenerates into smaller structures with same sense of vorticity. These smaller structures interact amongst themselves in a highly unsteady manner at further downstream planes. As noted earlier, the unsteady anti-kidney vortex pair over the CVP (hole A) will inhibit the jet lift off near the hole region while the kidney vortex pair over the CVP (hole C) will assist the jet lift off. The lateral spread of the CVP also increases with the hole aspect ratio. The
contours of Reynolds stress $v'w'$ at various X-planes are presented in figure 5.7. The values of Reynolds stress $v'w'$ are high near the edges of CVP at X/D = 1.0 and 3.0. The nature of this stress component is to damp these vortical structures. The distribution of this stress component becomes patchy at the locations where the CVP degenerates into smaller vortices. The large values of $v'w'$ in the mainstream over the CVP at X/D = 1.0 for hole C corresponds to the stresses acting on the leeward edge of the jet.

Figure 5.7 Contours of Reynolds stress $v'w'$ at planes X/D = 1.0, 3.0, 5.0 and 7.0.

Instantaneous velocity vectors and contours of $\omega_y$ are presented at Y/D = 0.1 in figure 5.8. Reynolds stress $u'w'$ is presented for various cases at the same plane in figure 5.9. The reverse flow in the wake region of the holes is has the maximum around jet center plane. This leads to the formation of anti-kidney vortices at the trailing edges of the holes (figure 5.4). The footprints of wake vortices are confined below the CVP for
hole A. There is evidence of reorientation of horseshoe vortex close to jet corner around the trailing edge of hole B. For hole C, around X/D = 3.0, we observe strong signatures of $\omega_y$ near the periodic boundaries. Since, vorticity can not be generated within the flow or at the wall in the normal direction, one can deduce that these upright vortices are generated as a result of reorientation of horseshoe vortices (Kelso et al (1994), Fric and Roshko (1990)). The magnitude of $u'w'$ is largest for hole C as compared to other cases. This is because of enhanced crossflow entrainment into the wake of large aspect ratio hole. Large values of $u'w'$ in bigger spanwise extent for hole C also indicate the larger lateral spreading of the jet.

Figure 5.8 Contours of $\omega_y$ and instantaneous velocity vectors at Y/D = 0.1

Figure 5.9 Contours of Reynolds stress $u'w'$ at Y/D = 0.1
The Z-component of vorticity as well as instantaneous velocity field is presented at Z/D = 0.0 in figure 5.10. The Reynolds stress $u'v'$ for the various cases is presented on the same plane in figure 5.11. We observe a strong horseshoe vortex system in front of the jet for holes B and C and spanwise rollups of vortices at the leeward edge of the jet. The vertical penetration of the jet increases with the aspect ratio of the hole. There are events of reattachment and ejection along the wall below the roller vortices. Clearly, the coverage of the wall by jet changes dynamically and thereby changes the film cooling effectiveness. Reynolds stress $u'v'$ corresponding to the horseshoe vortex is largest for hole C while the patches of $u'v'$ at the leeward edge of the jet corresponding to rollup
vortices are largest for hole A and tendency of this stress is to damp these vortices. The positive values of $u'v'$ near the wall are observed at further downstream locations from jet center for hole A as compared to hole C. The negative values of $u'v'$ at the windward edge of the jet represents the mixing at the jet-crossflow interface.

Figure 5.12 Contours of $u_{rms}$, $v_{rms}$ and $w_{rms}$ at $Z/D = 0.0$.

In figure 5.12, the rms values of u, v and w fluctuation field are presented. The contours of $u_{rms}$ and $v_{rms}$ show local large values in front of the jet corresponding to the location of horseshoe vortex in the cases of hole B and C. The $u_{rms}$ values are largest at the leeward edge of the jet in the wake region for hole A and least for hole C. The values of $v_{rms}$ in the wake region along the leeward edge of the jet indicate that normal stresses result in enhanced mixing of coolant jet with the entrained wake. These normal stresses are largest for hole A and lowest for hole C in the near jet field. However, on further
downstream planes the trend is reversed. The value of \( w_{rms} \) is highest for hole C and lowest for hole A in the wake region. The value of \( w_{rms} \) on the windward edge of the jet for hole C is negligible as compared to other cases. The leeward edge of the jet is much farther into the mainflow for hole C as compared to the other cases where it is closer to the wall and entrained crossflow. Clearly, greater mixing is achieved for hole A and B as compared to hole C in the wake region of jet.

Figure 5.13 Contours of a) \( \omega_x \), b) \( \omega_z \) and c) wall shear stress at \( Y/D = 0.1 \)

Contours of \( \omega_x \), \( \omega_z \) and wall shear stress is presented in figure 5.13. The values of shear stress correlates with \( \omega_z \) in front of the jet and with \( \omega_x \) in the wake region. There is clear indication of the presence of \( \omega_x \) at the trailing edge of the hole A and B in the opposite sense of CVP. Moreover, most of the vorticity in CVP comes from the sidewalls of the holes and warped mainflow (reverse flow) at the trailing edge will distort and reorient the trailing edge \( \omega_z \) into X-direction. The contours of \( \omega_z \) in front of the jet for holes B and C indicate the presence of horseshoe vortex. Signature of horseshoe vortex
for hole A is seen after it bends around the jet in the contours of $\omega_x$. The value of shear stress increases locally in front of the jet due to horseshoe vortex. Using Reynolds analogy of heat transfer and skin friction coefficient, one may expect a higher heat transfer at this location. Large values are obtained along the sidewalls of holes. The low values indicate the separation events in the flow field.

5.1.3 Concluding Remarks

Large eddy simulations of the jet-in-crossflow for three rectangular holes were performed. The results obtained are consistent with the experimental observations of various researchers (Haven, 1996, Haven and Kurosaka, 1997, Andreopolous and Rodi, 1984, Fric and Roshko, 1994, Ajersch et al, 1995, Kelso et al, 1996). Following represent some of the major observations made in this study.

- The dispersion of jet in the mainstream is distinctly different for the three different hole geometries (Haven (1996) and Haven and Kurosaka (1997)). Dynamics and evolution of various flow structures is influenced by the hole geometry in the near jet region for the three hole aspect ratios investigated in this paper at this Reynolds number.

- For hole A, the streamwise velocity profile warps in same sense as observed by Haven (1996) (though not to the same extent). This leads to the evolution of leading edge as well as trailing edge vorticity that has the opposite sense between the CVP. The jet penetration is also smaller due to the counteracting induced velocity of this pair on the CVP.

- For hole B, the mixing in the wake region is enhanced and the jet penetrates farther into the mainflow as compared to hole A. The reverse flow in the wake region warps the crossflow profiles such that it leads to the evolution of trailing edge vorticity into anti-CVP vorticity. CVP breaks up into a multiple decked structure at downstream locations.
• For hole C, the horseshoe structure is dominant in front of the jet. The jet penetration is highest in this case. The CVP breaks down into multiple decked structure and these smaller vortices interact dynamically at further downstream locations. Large entrainment of the crossflow fluid is observed in the wake region for this case.

• The wake vortices are observed in all cases. However, the lateral migration of these vortices is largest for hole C and smallest for hole A.

• From the instantaneous snapshot at \(X/D = 1.0\), it is clear that the horseshoe is relatively stronger than the CVP for hole A while the CVP is stronger than the horseshoe for hole C. It may be conjectured that the induced motion over the CVP itself leads to the formation of anti-kidney pair above the CVP for hole A. Similar arguments can explain the kidney pair above the CVP for hole C.

• The shear stress \(u'w'\) magnitude correlates well with the lateral spreading of the jet. The shear stress \(v'w'\) acts to damp the secondary vortex motions while \(u'v'\) controls the jet penetration and the mixing at the jet-mainstream interface. Distinct distributions for various cases indicate differences in the dynamics and evolution of flow structures in corresponding cases.

• Contours of vorticity on a plane parallel to wall correlates well with the wall shear stress. The \(\omega_x\) correlates in the wake region with shear stress, while \(\omega_z\) correlates in front of the jet.

• For film-cooling applications, hole A will provide better film-coverage. For large aspect ratios, a slot type injection geometry will be obtained. However, there are issues regarding structural rigidity of blade surfaces with rows of slots.
5.2 Large Eddy Simulations of Jets in Crossflow: Effect of Jet Inclination Angle

The inclination angle of the jet is an important parameter that controls the development of the flow structures and the penetration and spreading of the jet. The effectiveness of film cooling is governed by the vertical and lateral spreading of the jet, and typically an inclination angle of around 30° has been found to be most desirable from the perspective of increasing film cooling effectiveness. The objective of this study is to numerically investigate the effect of the jet inclination angle on the various flow structures. Two jet-inclination angles are studied: 90° representing normal injection, and 30° representing simple-angle injection. The numerical study will be done through time and space accurate simulations of the filtered Navier Stokes equations (Large Eddy Simulations). Although the focus of the study is on the flow field, a qualitative idea of the heat transfer can be obtained by examining the dispersion of the jet in the crossflow and by the shear stress distribution at the wall.

5.2.1 Problem Description

A schematic of the physical problem studied is shown in Fig. 5.14a (for normal injection) and in Fig. 5.14b (for inclined injection). A uniform Cartesian grid of 122×52×32 points is used for a domain of 12D×5D×3D (Figure 5.14). At the inflow, a fully developed turbulent boundary layer profile is prescribed. In the normal jet case, the velocity field is specified at the jet inlet from the experiments of Ajersch et al. (1995). The Reynolds number based on the jet velocity and the hole dimension is 4700 for this case. For the inclined jet case, the velocity at jet inlet is prescribed from the experiments of Findlay et al. (1996). The Reynolds number based on the jet velocity and the hole spanwise dimension is 5000 for this case. For both configurations, a periodic boundary condition is
applied in the spanwise direction. The domain size is chosen such that free-stream conditions at the inlet can be used as the boundary conditions along the top plane. At the outflow, a convective boundary condition is used where the wave speed is determined from a flux balance.

Figure 5.14a: Schematic of the computational domain for normal jets.  
Figure 5.14b: Schematic of the computational domain for inclined jets.

5.2.2 Results

Results are presented for two different jet inclination angles at various planes of the computational domain. The X- component of mean velocity and rms fluctuations $\sqrt{(u'u')}$ and $\sqrt{(v'v')}$ are plotted at locations $X/D = 3.0$ and $5.0$ in the jet center plane, $Z/D=0$ (Figure 5.15a and 5.15b). A good agreement is obtained between the present predictions and the experimental data of Ajersch et al. (1995) for normal injection (Fig. 5.15b) and the data of Findlay et al. (1996) for $30^\circ$ simple injection (Fig. 5.15a). In the normal jets case (Fig. 5.15b), the mean profile at both $X/D$ of 3 and 5 exhibits three distinct regions: the shear-layer or jet region at the top, the wake region in the middle, and a near-wall boundary layer region induced by the entrained crossflow. The $u_{rms}$ appears to scale well
with $\frac{\partial U}{\partial y}$, with a maximum in the jet region and a minimum in the wake region. The $v_{rms}$ is highest in the jet region and is damped rapidly as the wall is approached.

At X/D=3.0 and 5.0, the peak $u_{rms}$ and $v_{rms}$ occur roughly around Y/D=1.2, while the vertical jet penetration distance is in the vicinity of Y/D=1.5. For the inclined jets (Fig. 5.15a), the jet penetration is considerably less (smaller than Y/D of 1), and wake effects are weaker. At X/D=3.0, the three regions noted in the normal injection case, although visible, are not as distinct. At X/D=5.0, the three regions are not visible, and wake recovery has progressed to a much greater degree than in the normal injection case. As for the normal injection case, the jet region appears to be associated with large rms values, but the magnitude of the rms values is nearly half of the values for the normal injection case. A notable exception is observed for the $u_{rms}$ which shows a large peak.
(with values as high as 0.4) close to the wall. This large near-wall value of $u_{rms}$ may be associated with the fact that the injected jet stays close to the wall, and its interaction with the entrained crossflow leads to significant production of turbulence near the wall.

The time-averaged vorticity contours ($\omega_x$) and velocity vectors are shown in Figure 5.16 at a location $X/D =5.0$ from the center of the jet. The contours of the turbulent stress $v'w'$ are presented at the same location in Figure 5.17. The CVP is an unsteady coherent structure and hence, time-averaged values, obtained after averaging over six flow-through periods, are shown to depict this organized coherent structure. The horseshoe vortex is also observed on the two sides of the CVP, and is associated with lower levels of $\omega_x$ in the opposite sense of the CVP. In comparing the flow structures for the normal and angled-injection cases, Fig. 5.16 shows results consistent with the velocity profiles in Fig. 5.15, that is, for the normal injection case, the jet penetration in the vertical direction, and the associated CVP structure, is much larger. The lateral penetration of the CVP is also more for the normal-injection case. However, the horseshoe vortex along the periodic boundaries appears to be considerably smaller for normal injection, and may be associated with the fact that the larger lateral growth of the CVP with normal injection compresses the horseshoe toward the periodic boundaries and restricts its growth in the lateral direction. The crossflow entrainment into the wake leads to the formation of the wall vortex structures (Figure 5.16) as also noted by Fric and Roshko (1994). These structures are seen for both jet inclinations. However, for the inclined jet case, due to the proximity of the jet to the wall, the interaction of the entrained crossflow with the jet is stronger leading to more complex near wall structures.
The $v'w'$ turbulent stress contours appear to correlate well with the CVP and the horseshoe vortex structures, implying that these large scale coherent structures in the flow are principally responsible for the Reynolds stresses. The locations of the peak stresses are shifted slightly upwards relative to the eye of the large structures, and correspond with the locations where the velocity gradient and the production of the turbulent stress is the greatest. In comparing the turbulent stress levels for 30° and 90° injection, it is observed that the magnitude of the peak stress levels is larger in 90° case almost by a factor of 7. Note that these shear stresses can be viewed as turbulent forces that influence the flow motion, and that in the right-half plane ($Z>0$), a positive stress value impedes the flow motion while a negative value aids the flow. The CVP motion is observed to be primarily damped by the opposing action of the turbulent stress $v'w'$. For the inclined jet, the horseshoe vortex motion is seen to also weaken due to this stress. The distribution of stresses is in the opposite sense on the edges of the CVP below the jet. This opposes the near wall motion induced by the CVP.
Vorticity contours of $\omega_y$ are shown at location $Y/D = 0.1$ from the wall (Figure 5.18). Significant entrainment of the crossflow into the wake region is evident, and the footprints of the coherent wake vortices can be clearly seen in the figure (Fric and Roshko, 1994). For the 90° injection, the blockage effect is greater, and the velocity vectors in Fig. 5.18 indicate that the approaching crossflow is deflected laterally to a much greater degree by the 90° jet compared to the inclined jet. Behind the jet, there is stronger crossflow entrainment into the wake region for 90° injection, and clear evidence of wake vortices or vertically oriented eddies are observed. For 30° injection, since the jet is much closer to the wall, evidence of wake vortices (with complete rollup) is not as strong in the instantaneous snapshot in Fig. 5.18. In both the 30° and 90° injection cases, it is observed that associated with the sweep of the crossflow boundary layer into the wake region, there is also an ejection of the wake fluid away from the jet centerplane. This behavior is more clearly observed in the 90° injection case. There is also greater excursion of the wake vortices in the lateral direction for the 90° injection and this is representative of the greater spreading of the jet. It is worth noting that in the instantaneous snap-shots, no symmetry is preserved along the jet centerplane, and crossflow entrainment is seen to cross over the centerplane from one half of the jet to the other half (particularly for the 90° injection). It is also observed that the flow structures in the wake are stronger at 90° injection, with the maximum vorticity magnitude for the 90° injection nearly 75% greater compared to the 30° injection. This is due to the stronger crossflow entrainment for the 90° injection case. Kelso et al. (1993, 1996) presented flow visualizations for these unsteady upright wake vortices. The vorticity at the jet-crossflow shear layer as well as the vorticity generated at the wall encounters the critical points in
the flow field downstream to the hole. The vorticity generated at the wall and within the hole is stretched and re-oriented by mean velocity gradients to give rise to such flow structures (Figures 5.26 and 5.27).

Contours of turbulent stress $u'w'$ that correspond to the vorticity contours in Fig. 5.18 are presented in Figure 5.19. As for the magnitude of $\omega_y$, the magnitude of $u'w'$ is lower for inclined jet, with peak values for the inclined jet lower by a factor of 2. Since $u'w'$ represents the turbulent stress in the x-z plane, its magnitude is indicative of the extent of turbulent transport of momentum in the lateral direction (Andreopoulos and Rodi, 1984). The greater lateral transport in the $90^\circ$ injection is evident from a comparison of the velocity vectors, where the influence of the jet clearly extends over a wider spanwise region for the $90^\circ$ injection. The $u'w'$ magnitude on this plane appears to be correlated with the magnitude of $\frac{\partial W}{\partial x}$ and $\frac{\partial U}{\partial z}$, indicating the applicability of the gradient approximation for the turbulent stress. The production of this stress is primarily due to $u'v'$ component working on $\frac{\partial W}{\partial y}$ and $v'w'$ component acting on $\frac{\partial U}{\partial z}$. Along the edges of the jet on the leeward side of hole, we observe the small regions of opposite signs for $u'w'$. After the jet exit, the presence of strong lateral gradient of streamwise velocity determines the local spreading of the jet around the hole. The magnitude of $u'w'$ at this location decreases as the injection angle is increased. Further downstream, the vertical gradient of spanwise velocity component comes into play and attributes to the higher levels of $u'w'$ and hence to the increase in the lateral spreading of jet.
Figure 5.18: Contours of $\omega_y$ and instantaneous velocity vectors at $Y/D = 0.1$ for jet inclination angles 30° and 90°.

Figure 5.19: Contours of Reynolds stress $u'w'$ at $Y/D = 0.1$ for jet inclination angles 30° and 90°.

Contours of the spanwise vorticity $\omega_z$ along the center plane of the jet ($Z/D=0$) are shown in Figure 5.20. Substantially greater vertical penetration for the 90° jet-injection case is evident in Fig. 5.20, with the jet penetrating upwards up to nearly 1.75D for the 90° jet-injection compared to less than 0.75D for the 30° jet-injection. The spanwise rollers on the leeward side of the jet are clearly evident in the 90° jet-injection case, consistent with the experimental observations of (Fric and Roshko, 1994 and Kelso et al, 1993, 1996). However, for the 30° jet-injection, the proximity of the jet to the wall damps the spanwise vorticity, and distinct spanwise rollers are not apparent in the instantaneous picture in Figure 5.20. The near-wall region clearly shows large scale sweep (instantaneous flow directed toward the surface) and ejection events (instantaneous flow directed away from the surface); this behavior is again more evident in the normal-injection case. Development of the horseshoe vortex upstream of the jet can be clearly seen in the 90° jet-injection case in the $\omega_z$ contours in Figure 5.20 and also in the $\omega_z$ contours in Figure 5.25. For this case, $\omega_z$ over the jet-hole region can be clearly observed.
in Figure 5.25, indicating the potential for crossflow ingestion into the hole region. For the 30° jet-injection case, the potential for such ingestion appears to be reduced.

Contours of the turbulent stress $u^'v^'$ are presented in Figure 5.21. Contours of $u^'v^'$ are mostly negative, and hence, the production of $u^'w^'$ becomes positive in the regions where the vertical gradient of spanwise velocity is positive. This happens predominantly at downstream location of hole around the CVP. The greater production of $u^'w^'$ at higher turbulence intensity levels enhances the lateral spread of the jet (Figure 5.18 and 5.19). The negative levels of $u^'v^'$ correspond to the mixing layer at the leeward side of jet. For normal injection of coolant jets, the positive levels of $u^'v^'$ are observed below the negative levels at downstream locations closer to the wall. These correspond to the wake region of the jet. Again, the magnitude of this stress is greater for 90° case.

![Contours of $\omega_z$ and instantaneous velocity vectors at Z/D = 0.0 for jet inclination angles 30° and 90°.](image1)

![Contours of Reynolds stress $u^'v^'$ at Z/D = 0.0 for jet inclination angles 30° and 90°.](image2)

Figure 5.20: Contours of $\omega_z$ and instantaneous velocity vectors at Z/D = 0.0 for jet inclination angles 30° and 90°.

Figure 5.21: Contours of Reynolds stress $u^'v^'$ at Z/D = 0.0 for jet inclination angles 30° and 90°.

Figure 5.22 presents the pressure contours along the jet centerplane (Z/D=0). The pressure gradients are expectedly larger for the 90° jet-injection case. The adverse pressure gradient upstream of the jet is clearly evident, and the signature of the horseshoe with a pressure-defect core can be seen. For 90° jet-injection, the spanwise rollers are
clearly identified by the series of closed pressure contours. The pressure contours for the 30° jet-injection case are relatively more uniform downstream of the jet hole, and no evidence of spanwise rollers can be seen in the pressure contours.

Figure 5.22: Contours of pressure and instantaneous velocity vectors at Z/D = 0.0 for jet inclination angles 30° and 90°.

Figure 5.23: Contours of skin friction coefficient at Y/D = 0.0 for jet inclination angles 30° and 90°.

Contours of skin friction coefficient on the wall are presented in Figure 5.140, and these are correlated with the magnitude of the $\omega_x$, $\omega_y$, $\omega_z$ vorticity above the wall shown in figures 5.24, 5.18 and 5.25 respectively (Chong et al., 1998). Low values of skin friction are representative of flow separation. In case of inclined jets, the flow separation region is small. For normal injection case, the horseshoe vortex in front of the jet is a very strong flow structure with a system of three eddies as shown in the $\omega_z$ vorticity contours. The skin friction coefficient distribution upstream of the jet appears to reflect this behavior. Similar correlation with $\omega_z$ can be seen upstream of the inclined jet. However, the distribution of skin friction coefficients in the wake of jets is predominantly aligned in the streamwise direction and correlates best with $\omega_x$ contours.
Figure 5.24: Contours of $\omega_x$ at $Y/D = 0.2$ for jet inclination angles 30° and 90°.

Figure 5.25: Contours of $\omega_z$ at $Y/D = 0.2$ for jet inclination angles 30° and 90°.

Figure 5.26: Particle traces originating from the location near the jet exit (near horseshoe vortex). ($X=0.3D,Y=0.3D,Z=\pm0.7D$)

Figure 5.27: Particle traces originating from the location of crossflow entrainment (near wall vortex). ($X=1.5D,Y=0.3D,Z=\pm0.7D$)

Particle traces are presented for the normal injection case to explain the unsteady upright wake vortices in Figures 5.26 and 5.27. These particle traces are rendered with Y component of vorticity. When the streamline is perpendicular to the wall and contains vorticity, we can deduce its contribution to the wake vortices. The contribution of the upright vortices also comes from the crossflow entrainment, which leads to the streamlines that are perpendicular to the wall (Kelso et al., 1993). As stated earlier, the
signatures of these wake vortices can be observed on a plane parallel to the wall (Figure 5.18). However, the vorticity of the jet and mainstream interface is also convected downstream in the wake.

For the particles closer to the jet exit (Figure 5.26), the streamlines are deflected in the wake to the upright position around 1.5D from the jet center. There is significant vorticity present along these almost vertical lines indicating the core of wake vortices. For the particle traces around the location of crossflow entrainment or the wall vortex, the streamlines bend to upright position close to the centerplane. The magnitude of the vorticity along the vertical portions of streamlines is smaller for these particle traces (Figure 5.27). Therefore, one can conclude that the major contribution to these upright wake vortices comes up from the re-orientation of streamlines in the vicinity of horseshoe vortex rather than the wall vortex which is generated by the crossflow entrainment.

5.2.3 Concluding Remarks

Large eddy simulations of the jet-in-crossflow for two jet injection angles were performed. The results obtained are consistent with the experimental observations of various researchers (Ajersch et al. (1995), Andreopoulos and Rodi (1984), Findlay et al. (1996), Fric and Roshko (1994) and Kelso et al. (1996)). Following represent some of the major observations made in this study.

- The dispersion of jet in the mainstream is distinctly different for the two different injection angles of 90° and 30° (Ajersch et al. (1995) and Findlay et al. (1996)). Significantly greater penetration and mixing of the jet with the crossflow is observed for the normally-injected jet.
• Large pressure gradients in the normal injection case indicate larger separation regions and lead to reduction in the wall shear stress. From film-cooling applications’ perspective, normal injection of coolant into hot crossflow would be detrimental to the film-coverage over blade surface. Therefore, most of the injection angles are around 30°-35° over the gas turbine blades. However, in the regions of crossflow stagnating on the blade near leading edge, it would be beneficial to have normal injection (It is a rather different situation involving stagnating flow instead of crossflow).

• Skin friction coefficient at the wall behind the jet correlates with the streamwise vorticity. For normal injection, the horseshoe vortex in front of the jet is responsible for low skin friction values upstream of the jet. Using analogy between higher skin friction regions and high heat transfer regions, one can estimate the distribution of heat transfer rates over the surface.

• The shear stress $u'w'$ magnitude correlates well with the lateral spreading of the jet. The shear stress $v'w'$ acts to damp the secondary vortex motions while $u'v'$ controls the jet penetration and the mixing at the jet-mainstream interface. Normal injection of jets is associated with greater magnitudes of these turbulent shear stresses, and is responsible for the observed greater penetration of the jet and greater mixing between the jet and the mainstream.

• Particle traces for normal injection case showed the entrainment of crossflow into the wake region of jet. Re-orientation of these particle traces in upright position explained the presence of crossflow fluid around wake vortices as observed in several experimental studies.
5.3 Large Eddy Simulations of Jets in Crossflow: Effect of Freestream Turbulence Intensity

5.3.1 Introduction

The ever-increasing demand for specific thrust and thermal efficiency of a gas turbine has resulted in turbine inlet temperatures exceeding the turbine blade material limits. To increase the life of the turbine blades, one has to keep the temperature of the blades below their melting point using some sort of cooling strategy. Film cooling is a technique in which rows of coolant jets are injected into the hot crossflow gas stream. These jets are deflected and strained by the crossflow and provide coverage of the blade surface from the hot gases. The interaction of the coolant jets with the crossflow involves complex and unsteady structures like the horseshoe vortex, the counter rotating vortex pair (CVP) and wake vortices. These structures control the jet penetration and its spreading rate. The effectiveness of film cooling is governed by the rate at which the coolant jets mix with the hotter surrounding flow. Important factors that influence the effectiveness are free-stream turbulence intensity levels, jet to mainstream momentum ratio and hole aspect ratio. The objective of this study is to investigate the effect of free-stream turbulence intensity on the various flow structures.

In order to reliably predict the jet-in-crossflow situation, the large scales have to be predicted accurately and the dynamics and interactions of small scales must be accurately predicted or modeled realistically. This requirement calls for either direct numerical simulations (DNS) or large eddy simulations (LES). Due to the overwhelming grid resolution requirements with DNS, in this study, the LES approach is adopted for the flow simulations.
The free-stream turbulence is characterized here as an homogeneous grid generated turbulence. Hence, a Gaussian distribution can be used to approximate the fluctuations (Batchelor, 1953). Gartshore et al (1983) studied the effect of external homogeneous turbulence on an initially turbulence-free region in which there is a mean velocity gradient. They showed that turbulence induces irrotational fluctuations in the sheared region, which interact with the shear to produce rotational velocity fluctuations and Reynolds stresses. These stresses extended into the shear region over the distance of the order of integral scale. Since, the production of Reynolds stresses involve the mean flow gradients and components of Reynolds stress tensor, these production terms are present at the jet and free-stream interface. However, the pressure-strain terms are due to rotation and distortion of eddies by the mean velocity gradients. Due to this distortion of eddies, the non-local pressure fluctuations are produced which can cause different fluctuation velocity components to be partly in phase and hence, govern the evolution of Reynolds stresses in an initially turbulence-free region. The evolution of Reynolds stresses is also governed by turbulent transport term (which is neglected in the rapid distortion theory analysis of Gartshore et al, 1983). Based on these observations, it would be expected that the homogenous turbulence levels above the film cooling jet would have a role to play in the dispersion of the coolant and on the near-wall heat transfer.

Ou et al (1990) investigated the effect of film hole row location on leading edge film cooling and heat transfer under grid generated high mainstream turbulence conditions. They observed that leading edge heat transfer increases with mainstream turbulence level for blowing ratio of 0.4 and 0.8. The mainstream turbulence adversely effects leading edge film effectiveness for blowing ratio of 0.4, but the effect reduces for
higher blowing ratios. Jumper et al (1991) reported that for low free-stream turbulence (0-4%) the cooling effectiveness is reduced by an interaction of the film with free-stream so as to increase film temperature with distance downstream of the injection. However, for high free-stream turbulence (14-17%), the cooling effectiveness is reduced not only due to increased film temperature but also due to destruction of cooling film by aggressive mixing with turbulent layer. Kohli and Bogard (1997) observed that with small free-stream turbulence, strong intermittent flow structures generated at the jet-mainstream interface disperse the jet by moving hot mainstream fluid into the coolant core, and ejecting the coolant fluid into the mainstream. They observed double peaked temperature p.d.f.s indicating the intermingling of distinct elements of fluid from free-stream and from the coolant. This penetration was observed up to Y/D = 0.1 at X/D = 3.0 for high free-stream turbulence, while the penetration of distinct elements of free-stream fluid is limited to a height of Y/D = 0.5 for low turbulence case. Burd et al (1998) measured the effect of jet hole length to diameter ratio under different free-stream turbulence levels. Clearly, the interaction of flow structures with free-stream turbulence conditions is an important parameter while conducting any study to vary the flow structures.

In this study results are obtained with two levels of homogeneous freestream turbulence: a 2% level representing low freestream turbulence, and a 15% level representing high freestream turbulence. The role of various components of Reynolds stress tensor in controlling the spreading and mixing of the jet with the mainstream and their dependence on free-stream turbulence intensity levels is studied.
5.3.2 Problem Description

A schematic of the problem of interest is shown in Figure 5.28, and consists of a row of square holes injecting fluid vertically into the crossflow. This physical configuration mimics the experimental configuration of Ajersch et al. (1985). The computational domain consists of a spanwise periodic module containing one injection hole. A uniform Cartesian grid of $122 \times 52 \times 32$ points is used for a domain of $12D \times 5D \times 3D$. The Reynolds number based on the jet velocity and the hole dimension is 4700. The jet to crossflow velocity ratio (called the blowing ratio) is chosen to be 0.5 corresponding to the experimental conditions of Ajersch et al. (1985).

Lund et al (1996) presented a methodology of generating turbulent inflow boundary conditions by introducing a buffer zone where coordinate transformations are applied to governing equations. The solution at the end of this buffer zone is rescaled at the inlet of the main computational domain. However, these simulations require extra effort due to the buffer zone. Alternatively, generating inflow conditions purely through random number generators will provide a turbulence field without significant computational effort. However, this field is not a solution of the governing flow equations at the inlet, but it evolves at downstream planes through the solution of the governing equations. Thus, a realistic field can be expected at downstream planes if the inlet fluctuations are generated from a proper probability distribution function.

At the inflow, a fully developed turbulent boundary layer profile is prescribed and a small scale turbulence field is superimposed on it. The Box-Muller method is used to generate the Gaussian distribution for the perturbation fields (Batchelor, 1953, Press et al, 1992). For this specific case of grid generated turbulence, the linear decay of streamwise
turbulent intensity with streamwise direction was predicted in accordance with theory (Figure 5.30). This agreement with the theoretical decay predictions validates the approach used here for generating freestream turbulence.

The velocity field is specified at the jet inlet from the experiments of Ajersch et al (1995). In the spanwise direction, the periodic boundary condition is applied. The domain size in the vertical direction is taken such that free-stream conditions at the inlet can be used as the top plane boundary conditions. At the outflow, a convective boundary condition is used where the wave speed is determined from a flux balance.

5.3.3 Results

Results are presented for two different turbulence intensity levels at various planes of the computational domain. The X- component of mean velocity and normal Reynolds stress are plotted in Figure 5.29 at streamwise locations $X/D = 1.0, 3.0, 5.0$ in the jet center plane i.e. $Z/D = 0$. For the mean velocity profiles, a good agreement is obtained between the experimental data of Ajersch et al (1995) and the computed results.
Figure 5.29 Comparison of X component of mean velocity and normal stress $u'u'$ at X/D = 1.0, 3.0, 5.0 respectively on the jet center plane (Z/D = 0.0)

The predicted magnitude of $u'u'$ for the 2% and 15% Tu cases are different in the wake region and the freestream region, but do not show significant differences in the jet-region. This implies that in the jet-region turbulence levels are primarily controlled by the gradient-production of turbulence, and diffusion or convection of freestream turbulence is not significant. However, in the wake region, the higher turbulence levels for the 15%Tu case implies that associated with the crossflow entrainment into the wake region there is significant convection and diffusion transport of the freestream turbulence. Similar results were reported by Bons et al (1996) who reported penetration of high freestream turbulence into the near wall region. It will be shown later that the production of $u'u'$ on the jet center plane is primarily due to $\partial U/\partial y$ in the vicinity of the jet-mainstream interface while the production is controlled by $\partial U/\partial z$ and $\partial U/\partial x$ at locations closer to wall in the wake region (Figures 5.36-5.39). The present simulations imply that the primary mechanism of the penetration of high freestream turbulence is the crossflow
entrainment into the wake region. Bons et al. (1996) reported that the net effect of the greater near-wall freestream turbulence levels is a reduction in the film-cooling effectiveness.

Figure 5.30 Decay of freestream turbulence intensity at Y/D = 4.5 and Z/D = 0.0

Figure 5.31 Freestream turbulence intensity for two cases at Z/D = 0.0

Figure 5.30 shows the turbulence intensity at the jet center plane at Y/D = 4.5. This Y/D location is sufficiently removed from the flat plate to represent freestream
conditions. The decay of turbulence is anisotropic and the decay rates are different for $u'u'$, $v'v'$ and $w'w'$. In the upstream region of the jet injection, all the normal stresses decay with different decay rates due to mean shear imposed by the boundary layer flow (Batchelor, 1953). However, unlike $v'v'$ and $w'w'$ components, the streamwise normal stress starts building up in freestream beyond the jet injection point. This is due to dominant production of $u'u'$ at the jet-crossflow interface and its subsequent diffusion into the freestream. Clearly, this process is much stronger for the Tu 15% case. Figure 5.31 shows the distribution of the turbulence intensity above the jet-crossflow interface (the intensities in the jet and wake region are not shown). Intensities much greater than the freestream turbulence are noted at the jet-crossflow interface implying that the primary mechanism for turbulence production is the jet-crossflow interaction. However, Figure 5.31 shows that the near-field of the jet hole is influenced by the freestream levels, with higher turbulence intensities associated with the higher freestream levels. It will be shown later that the wake of the jet is similarly influenced by the freestream turbulence levels.

Individual terms contributing to the production of all six components of Reynolds stresses for the two freestream turbulence cases are presented in Figures 5.32-5.35 at two downstream locations to jet injection: $X/D=3.0$, $Z/D=0.0$, corresponding to the jet centerplane and $X/D=3.0$, $Z/D=-0.5$, corresponding to the spanwise edge of the injection hole. Of specific interest here is the validity of the gradient approximation that is intrinsic to turbulence models.

For Tu 2%, the turbulent production of $u'u'$ in the wake region (close to the wall $Y/D < 0.6$) is dominated by $\partial U/\partial x$ (negative production) and $\partial U/\partial z$ (positive production).
The latter ($\partial U/\partial z$) is particularly important along the spanwise edges of the CVP (Fig. 5.33) as the crossflow is entrained towards the jet centerplane. The relative importance of $\partial U/\partial z$ implies that the gradient approximation for $u'u'$ is clearly invalid in the wake region. In the deflected jet region away from the wall ($Y/D > 0.75$), the production of $u'u'$ is governed primarily by $\partial U/\partial y$, which is again inconsistent with the gradient approximation. Near wall production for the 2%Tu case is observed to be negative in contrast to the always positive production predicted by the gradient assumption. In comparing the stresses for the two Tu cases, the general trends are identical, except for the differences in magnitude (somewhat higher stresses for the 15% case). Production of $v'v'$ is governed primarily by $\partial V/\partial y$ and the production of $w'w'$ is governed primarily by $\partial W/\partial z$ at both the stations. These are quite consistent with the gradient approximation. Notable exception to the gradient assumption is that the production is negative close to the wall. The effect of the high freestream turbulence intensity is again limited to altering the magnitude of the dominant production terms.

The turbulent production of $v'w'$ is observed to be primarily due to $w'w' \cdot \partial V/\partial z$ and $v'v' \cdot \partial W/\partial y$. Along the jet centerplane ($Z/D=0$) and the spanwise edge of the jet ($Z/D=-0.5$), the normal stress $v'v'$ is relatively low near the wall (due to wall damping), while $w'w'$ is relatively high (see Fig. 5.32 for relative production terms for these two stresses), thus reducing the production due to the $v'v'$ components. At locations away from the wall (in the jet region), $v'v'$ becomes significant while $w'w'$ reduces, and the most significant contribution to the production term comes from ($\partial W/\partial y$) and the shear stress $v'v'$. Note that the dependance of $v'w'$ on $\partial V/\partial z$ and $\partial W/\partial y$ reflects adequacy of the gradient approximation for this component of stress.
Figure 5.32 Production of Reynolds Stresses for $Tu = 2\%$ at $X/D = 3.0$ and $Z/D = 0.0$

Figure 5.33 Production of Reynolds Stresses for $Tu = 2\%$ at $X/D = 3.0$ and $Z/D = -0.5$
Figure 5.34 Production of Reynolds Stresses for Tu 15% at X/D = 3.0 and Z/D = 0.0

Figure 5.35 Production of Reynolds Stresses for Tu 15% at X/D = 3.0 and Z/D = -0.5
The turbulent production of $u'v'$ is primarily due to $\partial U/\partial y$, while the production of $u'w'$ is governed by $\partial U/\partial z$. These dependances are also consistent with the gradient assumption. However, Fig. 5.32 and Fig. 5.34 also shows modest influences on $u'w'$ due to other gradient terms including the $\partial W/\partial y$ and $\partial U/\partial y$ term. There is negative production of normal stress components for all cases. This feature cannot be captured by eddy viscosity assumption in which stress tensor is aligned with the mean strain rate tensor, hence always yielding a positive value for the modeled production of normal stresses. Again, the dependence of stress production on different velocity gradients than those used in eddy viscosity assumption is also observed.

Attention is turned next to the turbulent shear stresses and their role in controlling the secondary flow motions in the domain. Note that these stresses are computed as runtime averages of LES fields and can be related to RANS turbulent stresses provided the averaged SGS tensor contributions are known (Appendix I). For normal stresses, knowledge of SGS kinetic energy is required (which is usually lumped in pseudo pressure for incompressible flow calculations). The turbulent production and the sum of turbulent diffusion and dissipation of these stresses are also presented to explain the evolution of Reynolds stresses. The stresses represent turbulent viscous forces acting on the fluid, and influence the translational and rotational motions. The shear stress $u'w'$ controls the lateral mixing of the jet. The shear stress $v'w'$ acts to damp the secondary vortex motions while $u'v'$ controls the jet penetration and the mixing at the jet-mainstream interface. The direct dependence of these controlling factors on the free-stream turbulence levels results in the different mixing processes at low and high turbulence levels, and results are presented below with this perspective.
Time-averaged vorticity contours \((\omega_x)\) and velocity vectors are shown in Figure 5.36 at \(X/D = 3.0\). The contours of the turbulent stress \(v'w'\) are presented at the same location in Figure 5.37. The CVP is an unsteady coherent structure and hence, time-averaged values, obtained after averaging over six flow-through periods, are shown to depict this organized coherent structure. The horseshoe vortex is also observed on the two sides of the CVP, and is associated with relatively low levels of \(\omega_x\) in the opposite sense of CVP. In comparing the turbulent stress levels at 2\% Tu and 15\% Tu, it is observed that the magnitude of the stress levels are larger in the 2\% case along the upper edges of the CVP. These shear stresses resist the vortical motion of the CVP. For Tu 15\%, since the peak stresses are lower the peak vorticity in the CVP is larger and leads to further jet penetration into the mainstream at downstream locations.

The turbulent production of \(v'w'\) and the turbulent diffusion plus dissipation (obtained by subtracting viscous diffusion and production terms from the convective terms in the Reynolds stress \(v'w'\) budget), are presented in figures 5.38 and 5.39 respectively. In general, the production of \(v'w'\) appears to correlate well with the stress levels themselves. As observed earlier (Figure 5.32-5.35), the dominant contributions to the production of \(v'w'\) are due to \(v'v'\) and \(w'w'\) stresses. The value of \(v'v'\) is largest in the core of the CVP. The value of \(w'w'\) is largest below the CVP close to the wall \((Y/D < 0.3\) and \(-0.4 < Z/D < 0.4\)). These magnitudes of \(v'v'\) and \(w'w'\) are larger for 15\% Tu case. However, the difference in the mean field velocity gradients under different free-stream turbulence levels can change the production of \(v'w'\) significantly. Therefore, the turbulence intensity levels directly influence the evolution of CVP.
Vorticity contours of $\omega_y$ are shown at location $Y/D = 0.1$ from the wall (Figure 5.40). Significant entrainment of the crossflow into the wake region is evident, and the footprints of the coherent wake vortices (Fric and Roshko, 1994) can be clearly seen in the figure. The larger excursion of the wake vortices in the lateral direction at the higher turbulence level is representative of the greater spreading of the jet. The $u'w'$ contours (Fig. 5.41) have regions of significant magnitude on each side of the jet. At 15% intensity, the contours show larger magnitudes relative to the 2% case, indicative of the penetration of the freestream turbulence through the spanwise entrainment of the crossflow boundary layer. As noted earlier, this appears to be the primary mechanism by which the freestream turbulence influences the behavior of the jet and the wake. The higher velocity fluctuations entrained into the wake are responsible for the higher $u'w'$ values and the larger spanwise excursions of the wake vortices for the 15% $Tu$ case. The higher $u'w'$ at 15 %$Tu$ implies much greater mixing between the spanwise flow structures. These observations are consistent with those of Bons et al. (1996) who report greater turbulence levels and reduced film cooling effectiveness in the wake for the higher $Tu$ case.

Turbulent production of Reynolds stress $u'w'$ at $Y/D= 0.1$ is presented in figure 5.42. The turbulent diffusion and dissipation of the same stress component on the corresponding plane is presented in figure 5.42. Influence of the freestream turbulence intensity is evident from the larger regions of high production of stress for the higher $Tu$ level. On a plane close to the wall, the normal stress component $w'w'$ is observed to play the most important role in the production of $u'w'$. It is worth noting that along this plane the $u'w'$ stress appears to approximately correlate with the corresponding rate of strain,
and the assumption of aligning modeled stress along the strain rate tensor through an eddy viscosity may work for the $u'w'$ component of stress at this particular plane. However, in the jet region where $u'v'$ is significant, the production due to $\partial W/\partial y$ as well as other gradients to vertical gradients of spanwise velocity component becomes relatively important (Figure 5.32 and 5.34). Therefore, an eddy viscosity type assumption to align Reynolds stress tensor with strain rate tensor will lead to incorrect predictions of $u'w'$ in the jet region. There is also a significant difference between the turbulent production and the sum of turbulent diffusion and dissipation of Reynolds stress $u'w'$. In the proximity of the wall, the viscous diffusion becomes significant and hence, leads to this difference.

Vorticity contours of $\omega_z$ are shown at location $Z/D = 0.0$ which corresponds to the center plane of the jet (Figure 5.44). The contours of turbulent stress $u'v'$ are presented at the same location in Figure 5.45. The presence of stronger free-stream structures at the higher turbulence intensity levels enhances the mixing at the jet-mainstream interface and hence increases the penetration of the jet much further into mainstream at downstream locations to the hole. A similar observation was made based on the PIV measurement of jet (Gogineni et al, 1996). The vorticity generated at the jet mainstream interface rolls up analogous to a mixing shear layer. The extent to which these vortical structures penetrate into the mainstream can be used as the measure of mixing of jet with mainstream (Bons et al, 1996). Negative levels of $u'v'$ (Figure 5.45) correspond to the mixing layer on the leeward side of the jet. Positive levels of $u'v'$ are observed in the wake region closer to the wall and have higher magnitudes for the higher Tu case. These higher magnitudes in the wake region are again linked to the entrainment of the crossflow into the wake region.
Production of this stress component (Figure 5.46) depends primarily on $v'v'\cdot \partial U/\partial y$ in the jet region. This production, in the jet region, is relatively unaffected by the freestream turbulence intensity. Turbulent diffusion and dissipation (Figure 5.47) is however different for the two different turbulence intensity levels. Increased levels of turbulent diffusion and dissipation are observed for Tu 15% at the leeward edge of the jet as well as in the wake.

It was observed earlier that the production of normal stresses becomes negative in some regions. The turbulent production of $u'u'$ and $v'v'$ is negative below the CVP at $X/D=3.0$. The turbulent production of $w'w'$ is negative on the jet edges in the vicinity of hole. This too, can not be modeled by any eddy viscosity assumption. Usually, negative production in jets and wakes is associated with difference in maxima locations of mean strain rate and turbulent stresses. Also, negative eddy viscosity does not mean anything more than a backscatter of turbulent energy from smaller scales to larger scales in the spectrum (Tsinober, 2002). However, the sum of the production of all normal stresses is proportional to the production of turbulent kinetic energy and no region of negative production of turbulent kinetic energy was observed. In the governing equation for turbulent kinetic energy, it would lead to realizable solutions provided the energy drain due to turbulent dissipation is modeled correctly (Schumann, 1977). This observation may explain the partial success of two-equation turbulence models in predicting reasonable mean flow field but incorrect levels of Reynolds stresses, since these turbulence models solves turbulent kinetic energy equation while modeling turbulent stresses through a gradient type eddy viscosity model.
Figure 5.36 Contours of $\omega_x$ and time averaged velocity vectors at X/D = 3.0 for turbulence intensity levels of 2% and 15%.

Figure 5.37 Contours of turbulent stress $v'w'$ at X/D = 3.0 for turbulence levels of 2% and 15%.

Figure 5.38 Production of Reynolds stress $v'w'$ at X/D= 3.0 for turbulence intensity of 2% and 15%.

Figure 5.39 Turbulent diffusion and dissipation of Reynolds stress $v'w'$ at X/D= 3.0 for turbulence intensity of 2% and 15%.
Figure 5.40 Contours of $\omega_y$ and instantaneous velocity vectors at Y/D = 0.1 for turbulence intensity levels of 2%, and 15%.

Figure 5.42 Production of Reynolds stress $u'w'$ at Y/D= 0.1 for turbulence intensity of 2% and 15%.

Figure 5.41 Contours of Reynolds stress $u'w'$ at Y/D = 0.1 for turbulence levels of 2% and 15%.

Figure 5.43 Turbulent diffusion and dissipation of Reynolds stress $u'w'$ at Y/D= 0.1 for turbulence intensity of 2% and 15%.
Figure 5.44 Contours of $\omega_z$ and instantaneous velocity vectors at $Z/D = 0.0$ for turbulence intensity levels of 2%, and 15%.

Figure 5.45 Contours of Reynolds stress $u'v'$ at $Z/D = 0.0$ for turbulence intensity levels of 2%, and 15%.

Figure 5.46 Production of Reynolds stress $u'v'$ at $Z/D = 0.0$ for turbulence intensity of 2% and 15%.

Figure 5.47 Turbulent diffusion of Reynolds stress $u'v'$ at $Z/D = 0.0$ for turbulence intensity of 2% and 15%.
5.3.4 Concluding Remarks

Large eddy simulations of the jet-in-crossflow for two different free-stream turbulence intensity levels are performed. The results obtained are consistent with the experimental observations of various researchers (Andreopolous and Rodi, 1984, Ajersch et al, 1995, Gartshore et al, 1983, Gogineni et al, 1996). Following remarks can be made from this study:

• The role of turbulent stresses on jet penetration and lateral mixing is demonstrated. Turbulent shear stress $u'v'$ controls the vertical penetration of coolant jet. Turbulent stress $u'w'$ controls the lateral spread of the coolant over the surface. Turbulent stress $v'w'$ opposes the motion of mean CVP in the flow and increases the cross-plane mixing of the coolant and crossflow fluid.

• Freestream fluctuations are modeled as Gaussian noise with zero mean and variance equal to turbulence intensity. Box-Muller algorithm (Press et al, 1992) is used for generating Gaussian random numbers. Freestream turbulence intensity decays like isotropic homogeneous turbulence and due to lack of any structure to these fluctuations (Batchelor, 1953). Higher free-stream intensity is shown to lead to greater mixing between the jet and the crossflow both in the vertical and lateral directions.

• The processes of turbulent production as well as turbulent diffusion of Reynolds stresses are affected by freestream levels of normal stresses. Term by term decomposition of turbulent production of resolvable components of turbulent stresses demonstrated the pitfalls of aligning modeled turbulent stresses with the mean strain rate tensor through a positive scalar eddy viscosity relationship.
5.4 Large Eddy Simulations of Jets in Crossflow: Effect of Freestream Turbulence Length Scales

5.4.1 Introduction

Large eddy simulations of jets in crossflow are performed to study the effect of energy containing scales present in the freestream on the penetration and spread of the coolant jet. Two specific freestream turbulence conditions are examined, one corresponding to 15% small scale Gaussian turbulence, and the other corresponding to a 15% freestream turbulence that satisfies the Von-Karman spectrum and has its peak energy specified in the small wave number range (large scales). The small-scale freestream turbulence can be viewed to be similar to grid generated turbulence. The large scale freestream turbulence spectrum has energy peak at a small wave number (corresponding to a specified length scale taken to be 4 hole diameters in this study) and has energy in the inertial subrange for large wave numbers. In the present study, the jets are issued through a row of square holes into the main crossflow. The jet to crossflow blowing ratio is 0.5 and the jet Reynolds number is approximately 4,700.

The length scale of freestream turbulence is an important parameter that controls the development of the flow structures and the penetration and spreading of the jet. Typical turbulence intensities over a turbine blade can be quite high (15-20%); however, the majority of studies on film cooling have used low freestream turbulence levels (2-5%). Another important parameter of interest is the length scale of the freestream turbulence which has typically been represented in reported studies by grid generated turbulence. However, under realistic operating conditions, the blade experiences the flowfield emerging from the gas turbine combustor, and this flowfield is characterized by large coherent eddies and integral length scales that are 3-4 times the cooling hole diameter.
Therefore, to accurately assess the behavior of the film cooling jet, the influence of freestream turbulence levels and length scales must be appropriately studied.

A number of studies have looked at the effect of freestream turbulence intensity and length scales on the vane heat transfer with and without film cooling (Kadotani and Goldstein, 1979a, 1979b; Ou et al., 1990; Bons et al., 1996; Kohli and Bogard, 1998a, 1998b; Ames, 1997a, 1997b; Radomsky and Thole, 1998). The general conclusion is that high freestream turbulence increases the mixing and surface heat transfer, and reduces the film cooling effectiveness. However, Bons et al. (1996) observed that an increase in film cooling effectiveness could be obtained with increasing turbulence levels in the region mid-way between the holes. A clear understanding of the various mechanisms involved is clearly not available due to the limited data, and additional measurements and computations are necessary. One drawback in the experiments reported has been the difficulty associated with the ability to simultaneously control the intensity levels and the length scales. Such control of intensity level and length scales can be more effectively provided in a time-and space accurate simulation, and forms the basis of the present study.

The objective of this study is to numerically study the effect of energy containing scales in the freestream on the various flow structures. Two different characterization of freestream scales are studied: (1) grid generated turbulence representing small-scale turbulence, and (2) turbulence that has Von Karman spectrum with a peak at low wave number representing large-scale turbulence. Small scale grid generated turbulence represents the majority of the film cooling studies conducted in wind tunnels, while the typical flowfield at the exit of the combustor is characterized by the Von Karman
spectrum with inertial subrange isotropy (Ames, 1997a). In the present study, both these freestream length scales are simulated, and the effect of the differing length scales on the cooling jet dispersion is examined. In order to focus attention primarily on the length scale issue, the turbulence levels are maintained constant at 15% for the two cases.

The majority of the reported computational studies on the jet-in-crossflow configuration have primarily solved the Reynolds-Averaged-Navier-Stokes (RANS) equations, and due to the intrinsic time-averaging that is associated with these equations, the dynamical nature of the vortical structures can not be predicted. Further, turbulence models have to be introduced, and the accuracy of the time-averaged calculations is itself compromised by the validity of the model. Examples of RANS calculations are those of Patankar et al., (1977), Sykes et al. (1986), Kim and Benson (1992), and Garg and Gaugler, (1994, 1995). Since the dynamics of the large scale features are important, the large scales have to be predicted correctly and the interactions of small scales must be modeled accurately. This requirement calls for large eddy simulations (LES), where all structures beyond a certain filter size are resolved, and the unresolved scales are modeled. To minimize numerical dissipation, an accurate numerical scheme has to be employed. More recently, Muldoon and Acharya (1999) have presented time-and space-accurate Direct Numerical Simulations (DNS) for a normally injected jet. Jones and Wille (1996) and Yuan and Street (1996) have presented Large Eddy Simulations (LES) that resolve the dynamics of the large scales and model the small scales, for a normally injected jet, and observed some of the reported phenomena in the experiments.
5.4.2 Problem Description

A schematic of the physical problem studied is shown in Fig. 5.48 where a single row of square coolant holes is injecting vertically upwards into the crossflow. This specific configuration has been selected to correspond to the geometry and measurements conditions (with grid generated low freestream turbulence) reported by Ajersch et al (1995). Due to spanwise periodicity in the flow, the computational domain consists of a single coolant hole, and periodic boundary conditions are applied in the spanwise direction. A uniform Cartesian grid of 122×52×32 points is used for a domain of 12D×5D×3D (Figure 5.48). At the inflow, a fully developed turbulent boundary layer profile is specified for the mean velocity profile, consistent with that reported by Ajersch et al. (1995). Both the inflow profile and the freestream conditions are perturbed, as described later, to generate the desired length scales and intensity levels. The velocity field at the jet-hole exit is specified from the experiments of Ajersch et al (1995). The Reynolds number based on the jet velocity and the hole dimension is 4700. The top boundary of the computational domain is chosen such that freestream conditions at the inlet can be used as the boundary conditions along the top plane. At the outflow, a convective boundary condition is used where the wave speed is determined from a flux balance.

To specify grid generated turbulence at the inlet and the freestream, the velocity fluctuations at these locations are randomly specified from a Gaussian probability distribution function that is characteristic of grid generated turbulence. The random sampling is done using the Box-Muller method. This ensures that the random field has Gaussian probability density for a given variance level. The fluctuation levels are
normalized in order to give 15% turbulence intensity at the inlet and in the freestream in all three coordinate directions.

For the large scale turbulence case, the energy spectrum is prescribed by the following relation

\[ E(k) = \frac{A k^4}{(B + k^2)^{1/6}} \]

Figure 5.48: Schematic of the computational domain.

Figure 5.49: Representative random signals in freestream at X/D = -1.5, Y/D = 4.0 and Z/D = 0.0.
Figure 5.50: Spectrum of X-component of velocity field at three stations (X/D = -1.5, 1.5 and 4.5) at jet centerplane and Y/D = 1.0.

This spectrum behaves as $k^4$ in the limit as $k \to 0$ and as $k^{5/3}$ in the limit $k \to \infty$. The energy spectrum has a peak at the wave number $k_m$ that is given by

$$k_m = \frac{2\pi}{\Lambda}$$

where $\Lambda$ is the integral length scale of the energy containing eddies in free-stream (Kravchenko and Moin, 1997). This location of peak determines the constant $B$. Spectrum signal is multiplied by random phase. Scrambling of phase information does not affect the energy distribution amongst scales as it contains the information about the orientation of eddies only. The negative wave numbers contain the conjugates of complex entries corresponding to positive wave numbers. Taking the inverse Fourier transform of this randomized spectrum will give the desired real signal. This signal is scaled with its $rms$ value and multiplied by the given intensity. The turbulence intensity is chosen to be 15% to correspond to the small scale turbulence case, while the peak energy is specified at a $\Lambda=4D$. The choice of 4 hole diameters is based on typical values reported by Bons et al. (1996) and Kohli and Bogard (1998a,b).

At X/D=-1.5, Figure 5.49 shows the temporal-signature of the normal velocity
component for both the small-scale freestream turbulence and the large-scale freestream turbulence. The figure illustrates the differences between the small scale turbulence with high frequency (small length scales) oscillations compared to the low frequency (large length scales) signature from a signal satisfying the Von Karman spectrum at the inlet (X/D = -3.5) with peak energy at a wave number $\Lambda=4D$.

### 5.4.3 Results

In presenting the results, the focus is on examining the time-averaged results from the two simulations (small-scale freestream turbulence and large-scale freestream turbulence), and in highlighting the differences in the observed result. Comparisons with the published data of Ajersch et al. (1995) have been reported in Tyagi and Acharya (1999a, 1999b) and show good agreement. For comparison with experiments, the simulations were done with low freestream turbulence (2% turbulence intensity) to be consistent with the experimental configuration of Ajersch et al. (1995).

Figure 5.50 presents the spectrum of the X-component of velocity at X/D=-1.5 (upstream of the jet), and at X/D=1.5 and 4.5 (downstream of the jet). The spectra for both the large-scale and small-scale freestream turbulence are normalized by the maximum energy corresponding to the large-scale case. Upstream of the jet (X/D=-1.5), the peak in the u-spectrum at the small wave number corresponding to $\Lambda=4D$ can be clearly observed for the large-scale freestream turbulence case, while the inertial subrange (with $-5/3$ slope) is captured at the higher wave numbers. This peak at $\Lambda=4D$ is significantly larger than the peak in the small-scale freestream turbulence case. The spectrum for the small-scale freestream turbulence case represents isotropic turbulence corresponding to the inertial subrange. Downstream of the jet (X/D=1.5), in the jet region
(Y/D=1), the peak at the small wave number disappears, energy is transferred from this peak to the smaller scales, and the spectra for the two cases look similar, indicating the dominance of the jet-crossflow interaction in generating the scales of turbulence. However, the energy levels at all wave numbers are still somewhat higher for the large-scale freestream turbulence case reflecting the weaker role of dissipation in the presence of the larger scales. Further downstream in the jet region (X/D = 4.5, Y/D = 1.0), the spectra for the two cases are quite similar.

A quadrant analysis was performed at three X/D locations of –1.5, 1.5 and 4.5 and at Y/D = 1.0 (Brodkey et al., 1974). In a quadrant analysis (Q1: \((u' > 0, v' > 0)\), Q2: \((u' < 0, v' > 0)\), Q3: \((u' < 0, v' < 0)\), Q4: \((u' > 0, v' < 0)\)), the flow motions are decomposed into “ejection” (Q2), “sweep” (Q4), “jet-ward interaction” (Q1), and “wake-ward interaction” (Q3) events. The “ejection” event corresponds to the mixing of the wake fluid carrying low streamwise momentum into the jet. The “sweep” event corresponds to the mixing of the jet fluid with high streamwise momentum into the wake. The interaction events can also be identified similarly. These events are identified in the similar fashion as the original analysis of turbulent events close to the wall (Table 5.48).

At X/D = -1.5, all the events occur for almost equal duration with comparable contributions and hence the observed level of turbulent stress \(u'v'\) is negligible for both cases. However, the individual contributions of the various events are much larger for the large scale case. At X/D = 1.5, the “sweep” event contributes most for the small scale. At the same station, for the large scale case, the “ejection” occurs longer with greater average contribution (almost twice) than the small scale case. The total stress level for the large scale case is almost twice than that of the small scale case. Clearly, the mixing
between the jet and wake fluid is greatly enhanced for the large scale case at \( X/D = 1.5 \). The jet fluid disperses into the wake region for the longer duration for small scale case. For the large scale case, the wake fluid is carried away by the jet. Thus, the dispersion of the coolant jet by the crossflow is different for these two cases and may have different implications for the transport of passive scalar like temperature. At \( X/D = 4.5 \), the “ejection” and “sweep” events occur with much larger contributions for longer duration for the small scale case as compared to the large scale case. Hence, the overall stress level is higher for the small scale case. Therefore, the effect of length scales has reduced significantly at \( X/D = 4.5 \) and enhanced mixing is observed for both cases at \( X/D = 4.5 \).

Table 5.4.1 Quadrant analysis of truncated turbulent signals at three stations (\( X/D = -1.5, 1.5 \) and \( 4.5 \), \( Y/D = 1.0, Z/D = 0.0 \)) for the two cases. The number in the parenthesis is the percentage of the sampling time duration spent by the turbulent signal in the respective event. (S represents small scale turbulence case, L represents large scale turbulence case)

<table>
<thead>
<tr>
<th>Event</th>
<th>( X/D = -1.5 )</th>
<th>( X/D = 1.5 )</th>
<th>( X/D = 4.5 )</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>S</td>
<td>L</td>
<td>S</td>
</tr>
<tr>
<td>Q1: Jet fluid interaction</td>
<td>0.001 (22%)</td>
<td>0.027 (27%)</td>
<td>0.020 (26%)</td>
</tr>
<tr>
<td>Q2: Ejection</td>
<td>-0.001 (30%)</td>
<td>-0.025 (24%)</td>
<td>-0.078 (26%)</td>
</tr>
<tr>
<td>Q3: Wake fluid interaction</td>
<td>0.001 (20%)</td>
<td>0.029 (25%)</td>
<td>0.057 (17%)</td>
</tr>
<tr>
<td>Q4: Sweep</td>
<td>-0.001 (28%)</td>
<td>-0.024 (24%)</td>
<td>-0.087 (31%)</td>
</tr>
<tr>
<td>Total Stress</td>
<td><strong>0.000</strong></td>
<td><strong>0.003</strong></td>
<td><strong>-0.032</strong></td>
</tr>
</tbody>
</table>

Figure 5.51 show the contours of the time-averaged normal stresses (\( u'u' \), \( v'v' \), and \( w'w' \)), their turbulent production and the sum of turbulent diffusion and dissipation.
along the jet center plane (Z/D=0). Consistent with the observations in Figure 5.50, the $u'u'$ values are seen to be much higher in the presence of large scale freestream turbulence. This is, in part, due to the reduced role of dissipation at the larger length scales. Further, for the large scale freestream turbulence case, the coherent vortical structures in the near field (the shear layer vortices, the horseshoe vortex, and the wake vortices) are energized directly by the energy containing large scales in the freestream following the cascade mechanism. This does not happen for small scale freestream turbulence since the length scales in the freestream are smaller than those of the coherent structures in the flow. In particular, the magnitude of the streamwise turbulence intensity associated with the horseshoe vortex system (upstream of the jet where the large scales still show peak energy as seen in Fig. 5.50) is nearly doubled for the large-scale turbulence intensity case. The turbulent production of $u'u'$ is significantly higher for the large scale turbulence case, particularly in the horseshoe vortex and the leeward edge of the jet. The greater energy in the horseshoe vortex system has ramifications on the downstream development of the horseshoe (as will be seen later in Fig. 5.52). For the large scale case, significantly greater streamwise turbulence intensity can also be seen in the wake of the jet, which in turn, may lead to greater heat transfer at the surface. For the large scale case, since the production of $u'u'$ in the wake is not greater, the higher $u'u'$ in the wake region is associated with entrainment of the more energetic crossflow into the wake. Corresponding to high production values, large turbulent diffusion and dissipation values are observed in the large scale turbulence case. Note that the turbulent production of normal stress $u'u'$ is observed to be negative between the horseshoe vortex and the windward edge of the jet. However, an eddy viscosity type approximation will always
yield positive production of normal turbulent stresses. Thus, for such anisotropic and
inhomogeneous turbulent shear flow, the eddy viscosity approximation is noted to be
inaccurate as also pointed out by Davis (1974) and Troshkin (1995).

The $v'v'$ contours shown in Figure 5.51b, again indicate that large scales in the
freestream energize normal fluctuations, particularly in the early regions of jet-crossflow
interaction (-1<X/D<2). Higher $v'v'$ in the nearfield of the jet leads to enhanced jet-
crossflow mixing, while higher $v'v'$ near the wall leads to increased skin friction
(Fig.5.59) and is likely to lead to increased wall heat transfer. The turbulent production of
$v'v'$ is large along the edges of the jet, the level being higher for the large scale
turbulence case. Again, the region of negative production close to the horseshoe vortex is
also observed for the large scale case. The turbulent diffusion and dissipation is also
much higher for the large scale case. Several investigators have shown a strong
correlation between the surface shear and heat transfer with normal velocity fluctuations,
and this expectation is confirmed in the present study. The $w'w'$ contours show higher
values in the freestream and along the upper edges of the jet for the large scale case.
However, large values close to the wall (Y/D < 0.3 and 1.5 < X/D <4) in the wake region
are observed for the small scale case. This is in contrast to the observations for the $u'u'$
and $v'v'$ components. The production of $w'w'$ is greater for the small scale case. It may be
due to the enhanced transfer and reorganization of the turbulent kinetic energy into $u'u'$
and $v'v'$ components for the large scale case.

Time-averaged streamwise vorticity contours and the velocity vectors at three
transverse planes corresponding to X/D = 1.0, 3.0 and 5.0 are shown in Figure 5.52.
Asymmetry across the mid-plane can be observed, and is related to the asymmetry in the
jet exit boundary conditions specified from the experimental data of Ajersch et al. (1995). The horseshoe vortex system can be seen to be considerably stronger and larger in the presence of large freestream turbulence scales. Evidence of this was seen earlier in Figure 5.51a, where upstream of the jet, the streamwise and wall-normal velocity fluctuations in the horseshoe vortex were considerably greater for the large-scale case. At X/D=1.0 and 3.0, the CVP is also seen to be stronger and exhibits greater penetration for the large scale case; this behavior is again linked to the higher energies (\(u'u'\) and \(v'v'\)) in the jet and wake associated with the large scale freestream turbulence. At X/D = 8.0, the large scales and their effects seemed to have been considerably diminished, and little difference is observed in the time-averaged vorticity and vector plots. For both cases, the entrainment of the crossflow into the wake region below the CVP can be seen; this leads to wall vortices containing vorticity with the opposite sign to the CVP. As discussed by several investigators (Muldoon and Acharya, 1999; Kelso et al., 1996; Fric and Roshko, 1994) the crossflow entrainment and the development of wall vortices lead to the development of the wake vortex structure (see Fig. 5.56). The Reynolds stress contours \(v'w'\) in the transverse planes X/D = 1.0, 3.0 and 5.0 are shown in Fig. 5.53, and, for the large-scale freestream turbulence case, considerably higher stress levels in the near field (X/D = 1.0 and 3.0) are associated with both the CVP and the horseshoe vortex. The highest stresses are associated with the edges of the CVP and correspond to the positions of the maximum velocity gradients. The stress profiles at X/D=5 for the two cases look similar, and reflect a decrease in the effect of the large scales as already observed.

Details of all the stress components, their turbulent production, the turbulent diffusion and dissipation in the transverse plane X/D = 3.0 are presented in figures 5.54
and 5.55. The values of $u'u'$ are higher for large scale case almost everywhere on the plane. The significantly high levels of $u'u'$ in the core of CVP and locations close to the wall clearly demonstrates the effect of length scales on the evolution of coherent structures. However, the turbulence production is not markedly different for the two cases because the production is dominated by $u'v'\partial U/\partial y$ for $Y/D > 0.75$, by $u'w'\partial U/\partial z$ and $u'u'\partial U/\partial x$ for $Y/D < 0.5$. Note that the negative production of $u'u'$ in the wake region is due to $u'u'\partial U/\partial x$ and cannot be modeled by any approximation that aligns the turbulent stress tensor along the mean strain rate tensor. Again, turbulent diffusion and dissipation of $u'u'$ is large for small scale case at the edges of CVP. There is some evidence of enhanced turbulent diffusion of $u'u'$ around the legs of horseshoe vortex for the large scale case which may be attributed to larger amount of turbulent kinetic energy associated with this vortex in this case. The higher levels of $v'v'$ in the horseshoe vortex for large scale case clearly indicates the enhanced turbulent kinetic energy in this coherent structure. The turbulent production of $v'v'$ is greater in the CVP for the small scale case but it is negative in the wake region ($Y/D < 1.0$, $-0.5 < Z/D < 0.5$) for both cases (with larger magnitude in large scale case). The turbulent diffusion and dissipation of $v'v'$ is from the CVP into the freestream (for $Y/D > 1.0$) and from the lower edges of CVP into the wake region. The distribution of $w'w'$ shows higher levels in the CVP lobes for large scale case while the levels of $w'w'$ are higher in the wake region for the small scale case. There is negative production of $w'w'$ in the CVP and below the lobes for both cases indicating that mean strain rate tensor is not aligned with the turbulent stress tensor. Turbulent diffusion and dissipation is higher in the CVP for the small scale case.
The turbulent shear stress $u'v'$ is mostly negative in the core of the CVP as expected due to the mixing of the fluid elements with different momentum values for both cases. As explained earlier using the quadrant analysis, the contributions around $Y/D = 1.0$ are mainly due to the “ejection” and “sweep” events resulting in an overall negative value of the stress. At $X/D = 3.0$, the mixing in the CVP is comparable for both cases, however, there is evidence of some mixing in the legs of horseshoe vortex in the large scale case only. The production of $u'v'$ is primarily negative in the CVP and positive in the wake region while the turbulent diffusion and dissipation is in the opposite sense for both cases. Clearly, these processes are working towards a quasi-equilibrium state for such an anisotropic and inhomogeneous turbulent shear flow as the flow moves away from the jet injection location. The distribution of shear stress $v'w'$ is explained earlier in the figure 5. The production of $v'w'$ is primarily due to $v'v' \cdot \partial W/\partial y$ and $w'w' \cdot \partial V/\partial z$ terms. The sign of $v'w'$ depends directly on the above-mentioned gradients, both of which are anti-symmetric and hence, the production is almost anti-symmetric too. The turbulent diffusion is higher in the CVP for the small scale case while it is higher below the CVP for the large scale case. The lateral spread of the coolant jet is controlled by the shear stress $u'w'$. Similar distributions are obtained for both cases, except the levels of $u'w'$ are higher in the wake region ($Y/D < 0.5$) for the small scale case. This indicates that the crossflow fluid is entrained and mixed well for small scale case at this location (it will be noted later that the crossflow entrainment occurs closer to jet for the large scale case). The production of $u'w'$ is higher in the CVP lobes for the large scale case indicating that the lateral spread of the jet is effected by the length scale in the
freestream. Similar distributions are observed for the turbulent diffusion and dissipation (except the sign change) for both cases.

Figure 5.56a shows the contours of the vertical vorticity component and velocity vectors along a horizontal plane (Y/D=0.1) above the jet-exit. Figures 5.56b, 5.56c and 5.56d show the corresponding Reynolds stress contours $u'w'$, its turbulent production, turbulent diffusion and dissipation along this plane respectively. The large-scale case clearly shows evidence of the horseshoe upstream of the jet (with reversed velocity vectors), while the small scale case does not clearly exhibit this flow separation. For the large scale case significant levels of the shear stress $u'w'$ are generated along the edges of the horseshoe and the spanwise edges of the jet. These large values are again a consequence of higher levels of streamwise velocity fluctuations noted earlier for the case of large-scale freestream turbulence. Clear evidence of wake vortices is seen in both cases, and as noted earlier, is a consequence of the crossflow entrainment into the wake, and the upward reorientation of the entrained flow. The primary difference between the large scale and the small-scale case is that in the large scale case, there is evidence of stronger crossflow entrainment (larger magnitudes of the entrained crossflow velocities) that manifests itself earlier in the near wake region. This is best illustrated by particle traces shown in figure 5.57, where the path of a particle injected on either edge of the jet-exit is displayed. Two views are presented one looking down, and the other looking from the transverse edge along a streamwise plane. It can clearly be seen that the particle entrained into the wake is reoriented in the vertical direction, confirming the earlier observations that the crossflow entrainment into the wake is the source of the wake vortex system. For the large-scale case, the entrainment into the wake and the upward
reorientation begins immediately downstream of the jet exit \((X/D = 0.5)\), and for the two trajectories shown is completed by \(X/D = 3.0\). For the small-scale case, the upward reorientation is delayed to \(X/D = 2.5\) and is completed downstream of \(X/D = 3\). The earlier entrainment of the crossflow for the large scale case is also associated with the location of peak stress and its production being closer to the jet exit relative to the location of the peak stress for the small scale case. Higher values of production and diffusion of \(u'w'\) are observed in front of the jet injection for the large scale case (figs. 5.56c and 5.56d). Since the production of \(u'u'\) contains the term \(u'w'\partial U/\partial z\) that is significant in the stagnation region in front of the jet, the higher levels of \(u'u'\) associated with horseshoe vortex are observed for the large scale case.

Spanwise vorticity contours and velocity vectors along the jet centerplane are shown in Fig. 5.58a, while the corresponding shear stress \(u'v'\), its turbulent production, diffusion and dissipation in the vertical plane are shown in Figs. 5.58b, 5.58c and 5.58d respectively. For the large scale case, the spatial perturbations in the freestream velocity is evident in the vorticity contour and the velocity vectors in Fig. 5.58a. As already noted, Fig. 5.58 also shows substantially stronger horseshoe system obtained for large-scale freestream turbulence. Recirculation region is substantially smaller for the large-scale case due to the stronger crossflow entrainment in the near jet region. In both cases, spanwise vortices on the leeward edge of the jet are observed, and the highest velocity gradients \((\partial U/\partial Y \text{ and } \partial V/\partial X)\) and shear stresses \(u'v'\) are associated with these vortical structures. For the large-scale case, the region of negative shear stresses, associated with the jet region, is much closer to the wall than for the small-scale case, reflecting the much greater mixing between the jet and the wake for the large-scale case.

For this lifted jet configuration, the greater mixing of the lifted jet (cold) with the
entrained crossflow (hot) may lead to improvements in film cooling. This should, of course be balanced by earlier observations, where for the large scale case, greater crossflow entrainment into the wake region can lead to greater heat transfer and reduced cooling. Region of negative production is observed on the windward edge of the jet while the production is positive at the leeward edge of the jet for the large scale case. Therefore, the production of $u'u'$ due to $u'v' \partial V/\partial y$ and $u'v' \partial V/\partial x$ is enhanced in the horseshoe vortex and the wake region for the large scale case. Again, the production of $v'v'$ due to $u'v' \partial U/\partial x$ and $u'v' \partial V/\partial y$ is greatly enhanced because of streamwise stagnation in front of the jet and the bending of the jet for the large scale case. The turbulent diffusion and dissipation is also higher for the large scale case. Figure 5.59 presents the surface skin friction as well as the streamwise and spanwise vorticity components just above the surface. The skin friction is observed to correlate well with the streamwise vorticity. Recall that streamwise vorticity is associated primarily with the CVP, the wall vortex, and the horseshoe vortex. The peak skin friction below the CVP/wall vortex system is an order of magnitude higher (in the range of 2-3 in non-dimensional units) than at other locations corresponding to the horseshoe or the wake (in the range of 0.15-0.7). Skin friction for the large scale case is substantially higher in the near field ($X/D<3$) jet, and is consistent with the higher $u'u'$ and $v'v'$ observed near the wall for the large scale case. Higher wall friction is also likely to be associated with higher wall heat transfer. Note that in the region just downstream of the jet exit ($0.5<X/D<2$), the flow separation in the small scale case leads to very small values of wall shear, while in the large scale case the absence of any significant recirculation in this region, as seen in Fig. 5.58, leads to high values of the wall shear.
Figure 5.51 Contours of normal stresses, the turbulent production, the turbulent diffusion and dissipation at Z/D = 0.0 (a) $u'u'$ (b) $v'v'$ (c) $w'w'$
Figure 5.52: Contours of $\omega_x$ and time averaged velocity vectors at planes X/D = 1.0, 3.0 and 5.0

Figure 5.53: Contours of Reynolds stress $\nu'\omega'$ at planes X/D = 1.0, 3.0 and 5.0 (Dashed lines represent the negative contours levels)
Figure 5.54 Contours of normal stresses, the turbulent production, the turbulent diffusion and dissipation at X/D = 3.0 (a) $u'u'$ (b) $v'v'$ (c) $w'w'$
Figure 5.55 Contours of shear stresses, the turbulent production, the turbulent diffusion and dissipation at X/D = 3.0 (a) u'v' (b) v'w' (c) u'w'

135
Figure 5.56: Contours of (a) $\omega_y$, and instantaneous velocity vectors, (b) Reynolds stress $u'w'$, (c) its turbulent production and (d) its turbulent diffusion and dissipation at $Y/D = 0.1$.

Figure 5.57: Particle traces showing the development of wake vortices (trace release stations at $X/D = 0.7$, $Y/D = 0.2$ and $Z/D = \pm 0.6$)
Figure 5.58: Contours of (a) $\omega_z$ and instantaneous velocity vectors, (b) Reynolds stress $u'v'$, (c) its turbulent production and (d) its turbulent diffusion and dissipation at $Z/D = 0.0$.

Figure 5.59: Contours of $\omega_x$ and $\omega_z$ at $Y/D = 0.1$ and contours of wall shear stress.
5.51 Concluding Remarks

- The larger length scales have a substantial impact on the turbulent stresses in the near field of the jet. Significantly larger values of $u'u'$, $v'v'$, and the shear stress components are observed with the large-scale freestream turbulence in the region $1<X/D<3$. The spanwise normal stress $w'w'$ is observed to be lower in the wake region for the large-scale case.

- The effect of the large scales is significant in energizing the horseshoe vortex system, and in enhancing the crossflow entrainment into the wake region. The entrainment of the crossflow boundary layer into the wake is initiated earlier, and is stronger, for the large-scale case. Therefore, the recirculation region behind the jet is considerably diminished in the large-scale case. These large-scale effects are primarily responsible for enhancing the turbulence.

- Quadrant analysis of the resolved field signal indicated different behavior for the dispersion of the jet fluid due to the crossflow for these two cases. It needs to be conformed by solving the temperature (scalar) equation under these freestream conditions to verify such conjecture.

- The dynamics of various flow structures, resolved Reynolds stress tensor, its production and diffusion plus dissipation were presented. However, the production of the turbulent stresses in the wake region yields much larger levels than the freestream for both cases and hence, the stress fields and flow structures are similar at further downstream locations to the jet injection.

- The wall friction correlates well with the streamwise vorticity, and is substantially higher for the case of large-scale freestream turbulence.
5.5 Large Eddy Simulations for Film Cooling Flows (UTRC Cases): Inclined Circular Jet with Heat Transfer

5.5.1 Introduction

Advanced gas turbines are designed to operate at high turbine inlet temperatures. Increased temperatures improve the second law efficiency as well as the specific thrust obtained by the turbines. This poses a challenge to design better and efficient cooling methodology for gas turbine blades in the first few stages after the combustor. Film-cooling is used to maintain the turbine blade temperature below their melting point for increased blade life. In film-cooling, coolant jets are injected at an angle into the hot mainflow that deflects these coolant jets over the blade surface to provide a coolant film coverage. However, the increased amount of coolant injection can deteriorate the aerodynamic performance and the gas path temperature drastically. Therefore, the amount of coolant injected should be optimal.

In this study, large eddy simulations (LES) are performed to study the flow physics and heat transfer for the film-cooling of gas turbine blade surface. The coolant jet issues out from a cylindrical delivery tube into the mainflow at an inclination angle of $35^\circ$. The Reynolds number based on the jet velocity and the diameter of the delivery tube is approximately 11100 and 22200, therefore the jet to mainflow velocity ratio is 0.5 and 1.0 respectively. A stagnation type plenum is simulated for jet to mainflow velocity ratio of 0.5 while RANS solution is provided at the jet delivery tube inlet for other cases. Heat transfer calculations are also performed simultaneously to study the mixing of the passive scalar with the mainflow, evaluate film-cooling effectiveness and heat transfer predictions on the blade surface. The parameters in the simulation correspond to the experiments performed at UTRC (Lavrich and Chiappetta, 1990)
5.5.2 Governing Equations and Computational Method

The non-dimensional governing equations for conservation of mass, momentum and energy for an incompressible Newtonian fluid in LES methodology are as follows:

\[
\begin{align*}
\frac{\partial U_j}{\partial x_j} &= 0 \\
\frac{\partial U_i}{\partial t} + \frac{\partial U_i U_j}{\partial x_j} &= -\frac{\partial p}{\partial x_i} + \frac{1}{Re} \frac{\partial^2 U_i}{\partial x_j^2} + \frac{\partial \tau_{ij}}{\partial x_j} + f_i \\
\frac{\partial \Theta}{\partial t} &= \left[ 1 - \Phi \right] \left( -U_j \frac{\partial \Theta}{\partial x_j} + \frac{1}{Re \cdot Pr} \frac{\partial^2 \Theta}{\partial x_j^2} + \frac{\partial q_j}{\partial x_j} \right) + \Phi \frac{\Lambda}{Re \cdot Pr} \frac{\partial^2 \Theta}{\partial x_j^2}
\end{align*}
\]

where \( U_i \) is the filtered velocity field, \( f_i \) is the body force terms arising due to immersed solid surfaces, \( \Theta = \frac{T_j - T_\infty}{T_j - T_\infty} \) where \( T_j \) is the coolant jet temperature and \( T_\infty \) is the crossflow temperature, \( \Lambda \) is the ratio of thermal diffusivity of the immersed solid to thermal diffusivity of the fluid and \( \Phi \) is the indicator function for solving the unsteady diffusion problem in the immersed solid. The indicator function is 1 inside the solid and zero everywhere else. The SubGrid Scale (SGS) stress tensor and SGS scalar flux vector are given by \( \tau_{ij} \) and \( q_j \) respectively. In this study, Dynamic Mixed Model (DMM) is used to model these SGS stress tensor and scalar flux vector (Moin et al., 1991 and Vreman et al., 1994). DMM can represent the backscatter of energy through scale-similar part while it can drain the energy from large scales to small scales using an eddy viscosity part. It is therefore considered as the least common denominator from the physical and mathematical requirements on SGS models. The box filters are used in the Germano identity for the calculation of dynamic coefficient and for the calculation of Leonard stresses. The dynamic coefficient is test filtered to avoid numerical instabilities.

The momentum equations are solved using projection method. The temporal
scheme is explicit second order accurate Adams-Bashforth scheme. The spatial
descretization is done using fourth order central finite-difference schemes for all the
terms except the convective term \( \left( \frac{\partial U_j}{\partial x_n} \right) \) that is upwind-differenced with a third accurate
scheme. The pressure-Poisson equation is solved using a direct solver based on matrix
diagonalization. The Laplacian operator is approximated using 4-2 formulation i.e. the
gradient operator is fourth order central difference operator and the divergence operator is
second order accurate central difference operator. All the terms in energy equation are
fourth order centrally differentiated.

5.5.3 Blowing Ratio (M = 0.5)

The experiments were conducted with a large tube-length (approx. 5.5D) and a can-type
plenum. However, in this study, the plenum is simulated as a stagnation flow field below
the jet-delivery tube. The mass flow rate into the plenum is such that the velocity ratio of
0.5 is achieved at the coolant hole exit. The jet-delivery tube is short (approx. 1.74D).
These changes are made to simulate a reasonable computational domain that retains all
the essential physics. Such discrepancies are not expected to change the flow field
dynamics and heat transfer in this flow situation drastically. A uniform grid of
172×102×42 is used to model the computational domain of size 17D×5D×4D, where D is
the diameter of the coolant jet delivery tube (figure 5.60). The film-cooled surface is
placed at 1.5D from the bottom of the computational domain. The center of the jet
injection hole at the film-cooled surface is 5D downstream from the inlet plane. The jet
delivery tube is simulated as a cylindrical surface inclined at 35° in the streamwise
direction (X) using Immersed Boundary Method. A crossflow-stagnation type plenum is
simulated in this study for the coolant supply. The top wall for the plenum is placed at
0.5D from the bottom of the computational domain. The bottom plane of the computational domain is treated using symmetry boundary conditions. Top boundary of the computational domain is treated as freestream boundary. At the inlet, fully developed turbulent profile is specified. At the outflow, a convective boundary condition is used where the convection speed is obtained from the mass flux balance. The spanwise direction (Z) is assumed to be periodic.

Figure 5.60 Schematic of the computational domain

5.5.4 Results (M = 0.5)

Comparison of time-averaged LES results with experimental data of Lavrich and Chiappetta (1990) and Sinha et al (1991) is presented at various stations in the computational domain (figure 5.61a-d) for blowing ratio of half (M = 0.5). Time-averaged statistics is obtained as the run-time average from the computation over approximately ten flow-through time periods (flow-through: time taken by crossflow to sweep the computational domain from inlet plane to exit plane). Sinha et al (1991) conducted experiments with short coolant delivery tube (L/D =1.75) as compared to UTRC cases (L/D > 6). Thus, LES calculations approximate Sinha et al (1991)
experimental setup more accurately. Streamwise component of mean velocity field is underpredicted at $X/D = 5.0$ at the jet centerplane (figure 5.61a). Vertical component of velocity field as well as temperature is accurately predicted at this station (figures 5.61b and 5.61d). Also, the accurate prediction of spanwise component of mean velocity field at a plane passing through hole edge is an indicator that lateral spreading of mean flow is correct (figure 5.61c). Film-cooling effectiveness is accurately predicted for the short-delivery tube experiments of Sinha et al (1991) (figure 5.61d). For long delivery tube experiments, there is significant jet lift-off and dispersion leading to reduced film-cooling effectiveness. Comparison of LES calculations with experimental results at blowing ratio of half is accurate and provides the confidence in LES procedure to discuss complex flow physics and unsteady heat transfer processes.

![Graphs showing velocity components at different stations.](image)

**a) Streamwise component of velocity at $Z/D = 0$**

**b) Vertical component of velocity at $Z/D = 0$**

**c) Spanwise component of velocity at $Z/D = 0.5$**

Figure 5.61 Comparison of the computed time averaged streamwise component of velocity with experimental data at different stations.
d) Film-cooling effectiveness along centerline

Figure 5.61 Cont.

There is a recirculation region inside the tube at the leeward surface (Figure 5.62a). This effect has been reported in earlier RANS studied (Walters and Leylek, 1997). The stagnation flow field below the jet delivery tube as well as the recirculation regions in the tube are observed in figure 5.62b. The development of the vorticity field inside the delivery tube leads to complex internal structure to the counter-rotating vortex pair (CVP) or the kidney vortex in this flow situation (Fric and Roshko, 1994, Andreopoulos and Rodi, 1984). The recirculation region behind the jet on the wall is also noted.

![Figure 5.62](image-url) Details of the velocity field inside the coolant jet delivery tube, (a) velocity vectors at Z/D = 0.0 and (b) streamwise component of vorticity, \( \omega_x \) at X/D = 0.0.
In figures 5.63a-c, instantaneous vorticity field components are presented on the jet-centerplane (Z/D = 0.0). Streamwise component of vorticity, $\omega_x$ indicates location of the lobes of CVP and intertwining of this vorticity component is an indicator of crossplane mixing. Contours of $\omega_y$ correspond to upright wake vortices that are shed into the wake of deflected jet. Contours of $\omega_z$ show the vorticity generated inside the tube around recirculation region near plenum, is shed along with the vorticity generated at the leeward edge of the coolant jet. Interaction of the vorticity generated at windward edge of coolant jet inside the delivery tube and oncoming boundary layer vorticity is also observed. The delivery tube vorticity dynamics gives rise to complex and unsteady CVP.

Figure 5.63 Instantaneous vorticity field components (a-c) on the jet-centerplane.
In figures 5.64a-c, instantaneous vorticity field components are presented over the film-cooled surface ($Y/D = 1.5$). Streamwise component of vorticity, $\omega_x$ around the periphery of injection-hole shows the origin of CVP. Contours of $\omega_x$ are aligned with streaks formed due to the entrainment of crossflow into the wake region. Contours of $\omega_y$ show a complicated structure of coolant jet inside the injection hole. Upright wake vortices are shed from the edges of the coolant jet due to the interaction with the crossflow, entrained into the wake region and convected downstream. Contours of spanwise vorticity component, $\omega_z$ show the vorticity generated along the delivery tube walls. This component plays an important role in changing the structure of CVP in the near field of coolant jet. An explanation of these projected vorticity components in terms of three-dimensional coherent structures is provided later.

![Figure 5.64 Instantaneous vorticity field components (a-c) on the film-cooled wall surface.](image-url)
The details of instantaneous temperature field are given at several section of the computational domain (figure 5.65).

![Instantaneous temperature field at different computational domain sections.](image)

(a) Z/D = 0.0

(b) Y/D = 1.5

(c) X/D = 0.0

(d) X/D = 2.5

(e) X/D = 5.0

Figure 5.65 Instantaneous temperature field at different computational domain sections.

The centerplane corresponds to Z/D = 0.0 and shows the mixing of the mainflow and the coolant jet. The coolant jet temperature drops in the downstream direction, however the coherent structures in the wake region retain their scalar value further (figure 5.65a). The temperature distribution corresponding to adiabatic wall boundary conditions corresponds to film-cooling effectiveness too (figure 5.65b). The coolant jet provides good coverage immediately downstream of the injection hole, however the film-cooling
effectiveness decreases monotonically in the wake region along the centerplane. The development of the coolant jet along the streamwise direction is shown at three different X/D locations (figures 5.65c-e). The coolant jet is observed to have a well defined kidney shaped structure (Andreopoulos and Rodi, 1984, Fric and Roshko, 1994). The crossplane mixing of scalar leads to the decrease in scalar value in the core of coolant jet at downstream stations. Moreover, the jet is attached to the surface near the centerplane but is lifted off below the lobes of the kidney vortex.

### 5.5.5 Coherent Structures in Jets-in-Crossflow

There is still no consensus on the generation mechanisms and evolutionary dynamics of coherent structures in jets-in-crossflow configuration. In an attempt to explain the flow physics better, the coherent structures are extracted from the time-dependent turbulent flow fields. The hairpin vertical structure is identified as the basic element of this flow configuration. The evolution of hairpin structure can explain the persistence of far-field structures and the unsteady vortices in various projection planes. Comparative assessment of experimental visualizations and coherent structures extracted from numerical datasets is presented here. Also, the generation of these vortices depends crucially on several flow parameters and it can rationalize the view of numerous parametric studies.

Hairpin vortices are believed to be the building block of turbulent boundary layers. The region near the surface is a complex environment dominated by the presence of a myriad of vortices that are believed to be predominantly of the hairpin type. Such vortices constitute moving lagrangian disturbances which carry concentrated vorticity in the core region that diffuses progressively outward with time. These vortices are embedded in a highly sheared background flow near the surface; over time they can be
expected to distort into complex shapes, as well as to interact with one another, resulting in the evolution of complicated vorticity topologies. Smith et al (1991) studied the evolution of such hairpin vortices that are generated by impulsively injecting fluid into a subcritical laminar boundary layer (Figure 5.66). Clearly, one can expect that similar hairpin structures should be present in the jet-in-crossflow situation with significantly increased complexity of both the background shear flow as well as the topology of lagrangian disturbance issuing out of the jet.

![Figure 5.66](image)

(a) Plan view  
(b) Side view

Figure 5.66 Picture of dye-marked single hairpin vortex generated using controlled injection through a narrow streamwise slot into a subcritical laminar boundary layer (Smith et al, 1991).

Zhou et al (1999) studied the evolution of a single hairpin vortex in the mean turbulent field of a low Reynolds number channel flow using direct numerical simulation. The initial flow structure was a viscous hairpin vortex structure extracted from the full two-point correlation tensor of a low Reynolds number channel flow database using the linear stochastic estimation (LSE) procedure. This initial vortex structure in a clean turbulent mean flow environment is studied to visualize the complex evolution and subsequent autogeneration of new hairpin vortices. They observed that new hairpins generate downstream of the primary hairpin above a threshold strength of this structure, thereby forming, together with the upstream hairpins, a coherent packet of hairpins that propagate coherently (Figure 5.67). These vortices were observed to pass low-speed fluid
from the downstream vortex to its upstream neighbor and so on over several hairpin vortices to form near-wall low-speed streaks of length significantly longer than the streamwise lengthscale of any single hairpin vortex. Since the wake region of jet-in-crossflow is populated periodically with such hairpin vortices that can autogenerate further, this mechanism is very critical in understanding the dynamics and influence of these vortices on the skin-friction and near-wall heat transfer for the film-cooling applications.

Figure 5.67 Packets of hairpin vortices in wall turbulence formed by autogeneration (Zhou et al, 1999). PHV, primary hairpin vortex; SHV, secondary hairpin vortex; THV, tertiary hairpin vortex; DHV, downstream hairpin vortex; QSV, quasi-streamwise vortices.

Fric and Roshko (1994) characterized the vortices in jets-in-crossflow using flow visualizations. They identified counter-rotating vortex pair (CVP), horseshoe vortex, jet shear layer vortices and upright wake vortices as the main coherent structures present in the flow (Figure 5.68). They concluded that upright wake vortices are formed due to re-orientation and entrainment of the crossflow boundary vorticity and almost no jet fluid was present in these vortices (Figure 5.69). On the contrary, Eiff and Keffer (1997)
argued that there is evidence of the jet fluid in these wake vortices (Figure 5.70). Smith and Mungal (1998) showed that for large jet-to-crossflow velocity ratio (~20), there is jet fluid in the wake vortices and for lower jet-to-crossflow velocity ratio (<10) there is no jet fluid. Clearly, the lagrangian disturbances issuing out of jet are hairpin shaped loci of local pressure minima. The jet fluid wraps around the head and streamwise oriented legs of these structures. The crossflow boundary layer is entrained into the wake region and is lifted upright around the vertical legs of hairpin structures.

Figure 5.68 Schematic of coherent structures in jet-in-crossflow configuration (Fric and Roshko, 1994)

Figure 5.69 Flow visualizations of the upright wake vortices in the jet-in-crossflow by seeding the crossflow boundary layer with smoke (Fric and Roshko, 1994).

Eiff and Keffer (1997) studied a jet issued from an elevated stack into crossflow and stressed that the jet-wake structures contain jet fluid; their vorticity originates in the jet’s shear layer (see Smith and Mungal, 1998). They observed that the Karman-like
vortex structures in the stack-wake are locked to jet-wake vortices that extend to just below the centerline of the jet on each side of the symmetry plane of the flow. To satisfy the solenoidal condition on vorticity field, these vortices must remain connected which is provided by the cross-link configuration (Vortex lines can not terminate in a flow but must be interconnected or end on a solid or free surface). The two central aspects of the cross-link configuration are that the vortices split in the jet-wake and that they are linked to each other just below the centerline of the jet. Coherent structure extraction from the time-dependent three-dimensional turbulent fields shows similar hairpin vortices (Tyagi and Acharya, 2001). The mean or phase averaged projection of these hairpin structures on different planes can explain experimentally observed phenomena in a unified fashion.

Figure 5.70 View of two orthogonal planes intersecting a typical jet-wake vortex convecting past the planes (Eiff and Keffer, 1997). Mean measurements in a vertical plane reveal a mean counter-rotating vortex pair with streamwise component (x) of vorticity. Pattern recognition technique results in a horizontal plane below the centerline of the bent-over jet revealing a pair of time-dependent counter-rotating vortices with wall-normal component (y) of vorticity.

Haven and Kurosaka (1997) studied the formation of kidney and anti-kidney vortex pairs over the stable counter-rotating vortex pair in the jets-in-crossflow issuing vertically from different hole geometries. They concluded that the hole geometry can significantly alter the crossflow strain rates (referred as warping) in the near-field of the jet-issuing hole. The subsequent evolution and break-up of the jet vorticity can result in the observed differences for various cases. Tyagi and Acharya (1999) observed that the
horseshoe vortex could have significantly different strength for different hole geometries. Induced strain rate fields due to jet vorticity (CVP in the near-field) and horseshoe vortex can also explain the different evolution of unsteady vortices over the CVP. The generic schematic of the jet vorticity is shown in figure 5.71 (Haven and Kurosaka, 1997). The deformation and strength of this jet vortex ring primarily depends on the jet Reynolds number, injection angle, jet-to-crossflow velocity ratio, upstream crossflow-to-jet boundary layer thickness ratio and hole geometry. The deformed jet vortex ring can lead to observed hairpin vortices for inclined circular jets in crossflow.

Figure 5.71 Schematic of the issuing jet vorticity and warping of the crossflow (Haven and Kurosaka, 1997).

Smith and Mungal (1998) determined the scaling laws for the different regimes of jets-in-crossflow for range of jets-to-crossflow velocity ratios \(10 < R < 200\). They noted that there is absence of jet fluid (and hence vorticity) in the upright wake vertical structures for velocity ratios less than 10 (in agreement with Fric and Roshko (1994)). However, for large ratios, there is significant jet fluid in these structures (as suggested by Eiff and Keffer (1997)). A plausible explanation can be provided in terms of wake vortices that are identified as the upright legs of the loci of pressure minima (hairpin
shapes) generated at the jet tube and convected into the wake region. The crossflow fluid wraps around these local pressure minima near the wall and lifts up to form wake vortices. At sufficiently high velocity ratios, the jet fluid leaks into these structures.

Figure 5.72 Instantaneous plan-view images comparing the wake structures at (a) M = 20 and (b) M = 10 for Re_{jet} = 33000 (Smith and Mungal, 1998).

Kelso et al (1996, 1998) presented flow visualizations for jets-in-crossflow and proposed the mechanism for the generation of shear layer vortices and the CVP by the warping, tilting, folding and reorientation of jet vorticity through K-H kind mechanism (Figure 5.74). They also identified vertical streaks and complex structure of CVP for large jet-to-crossflow velocity ratio. The folding of the jet-crossflow interface along the sides of the jet forms vertical streaks. However, they could not identify any well-organized vortex structure for the CVP. Careful examination of flow-visualization pattern (Figure 5.75) at this jet-to-crossflow velocity ratio shows the evidence of hairpin structure right above the jet exit that disintegrates into a highly randomized and mixed interface at the leeward side of the jet.
Figure 5.74 Shear layer evolution in the transverse jet: (a) isometric view of the jet shear layer vortex rings, showing how they tilt and fold as they convect downstream, and (b) schematic diagram of the reorientation of the shear layer vorticity, leading to the folding of the cylindrical vortex sheet (Kelso et al, 1996).

Yuan and Street (1998) reported that entrainment of crossflow fluid is the primary mechanism by which the jet changes course downstream of the injection (characterized with a power-law fit of jet trajectories to a single curve). Yuan et al (1999) attributed the origin of ubiquitous far-field counter-rotating vortex pair from a pair of quasi-steady ‘hanging’ vortices. These vortices form in the skewed mixing layer that develops between jet and the crossflow fluid on the lateral edges of the jet (K-H instability). Axial flow through the hanging vortex transports vertical fluid from the near-wall boundary layer of the incoming pipe flow to the backside of the jet. These hanging vortices

Figure 5.75 Flow pattern obtained when dye is injected from a small injection hole below the edge of the pipe exit into the upstream side of the shear layer, and also from a circumferential slit to mark evenly the cylindrical shear layer of the jet. The Reynolds number is about 750 and the velocity ratio is about 5 (Kelso et al, 1998).
encounter an adverse pressure gradient and breaks down. The vortex diameter expands dramatically after the breakdown, and a weak counter-rotating vortex pair is formed that is aligned with the jet trajectory. This view is in contrast with the explanation for origin of CVP in the issuing jet vorticity. The upright or wake vortices are formed when the streamwise vortices near the wall are reoriented by the strain field directly behind the jet. Although the reorientation of streamwise vortices into upright position can explain the wake vortices, it cannot explain the presence of jet fluid in these structures at large velocity ratios.

![Figure 5.76](image)

Figure 5.76 (a) Vortical structures and (b) pressure iso-surfaces in jet in crossflow simulations of Yuan et al (1999).

Blanchard et al (1999) explained the origin of CRVP in terms of elliptic instability contrary to the generally accepted view of K-H instability. The high selectivity of the dominant frequency of appearance of transverse structures (usually referred as shear layer vortices, rib vortices or ring like vortices) is not characteristic of K-H instability. Therefore, these vortices are a result of the global instability of the CVP due to their elliptical shape. The spatial gap between two successive structures is interpreted as the longitudinal wavelength of the global instability. The schematic of these structures is shown in figure and is mostly in agreement with our simulations.
Rivero et al (2001) used hot wire as well as planar laser induced fluorescence (PLIF) to identify the structures present in a jet in crossflow. Three different structures – folded vortex rings, horseshoe vortices and handle-type structures were identified. The handle-type structures link the boundary layer vorticity with the counter-rotating vortex pair through the upright tornado-like vortices. The upstream lateral (or spanwise) vorticity in the wall boundary layer is lifted as the crossflow approaches the jet exit and senses the adverse pressure gradient imposed by the jet. This pressure field is also responsible for the generation of vorticity of opposite sign that forms near the wall at the upstream side of the jet and constitutes the horseshoe vortex system. The azimuthal vorticity in the inner pipe walls is reoriented as the jet exits and penetrates the above-mentioned pressure field; the adverse pressure gradient at the upstream side slows down the associated vortex rings and the opposite occurs at the downstream side where the vortex is lifted. The fast homogenization of the pressure field from the exit plays an important role in the vortex breakdown that occurs on the downstream side of the jet, which results in the small-scale turbulence production. The alternate shedding in the leeward side of the jet is due to the lateral separation of the wall boundary layer giving place to the upright vortices in the pseudo-wake that develops downstream of the jet. Figure 5.77 Schematic diagram of the three-dimensional shape of the unsteady structures (Brizzi et al, 1995, ref: Blanchard et al, 1999)
They concluded that the overpressure both at the front and inside the jet exit is the main mechanism responsible for the distortion and reorientation of vorticity. The pressure field around the jet exit plays a dominant role in the formation of the far-field structures. In our opinion, Rivero et al (2001) have identified for the basic elements of the flow as the different parts of single hairpin structure and their explanation would then be consistent for different regions of the hairpin structure.

Figure 5.78 Sketch over an instantaneous photograph of the significant vorticity lines and instantaneous structures currently defined in the jet in crossflow (Rivero et al, 2001). The following are observed: the vortex lines in the wall boundary layer upstream of the jet; the vortex rings as they are folded and wrinkled; the vertical horseshoes from the bottom of the wall to the top of the upper jet interface; and the ‘handle-type’ structures similar to the horseshoes, but located along the lower jet interface, where the cores of the CVP are placed.
Figure 5.79 Side view of the evolution of the transverse jet: $U_{\text{jet}}/U_{\text{cf}} = 2.5$, (a) $t = 3.0$, (b) 4.0, (c) 5.0, (d) 6.0, (e) 7.0, and (f) 8.0. Vortex filaments with initially 24 nodes and core size $\sigma^2 = 0.1$ are introduced in the flow at each time step $dt = 0.02$ (Cortelezzi and Karagozian, 2001).

Cortelezzi and Karagozian (2001) performed three-dimensional vortex elements simulation to understand the vortex ring roll-up, interactions, tilting and folding in the near field of a jet in crossflow. In their study, folding of the vortex rings is the primary
mechanism responsible for the initiation of the counter-rotating vortex pair as well as the stretching and deformation of the later-forming vertical structures. The deformation and interaction of successive vortex rings can be delayed with an increased upstream boundary layer thickness and this period can be decreased with increasing jet-to-crossflow velocity ratio. The evolution of vortex rings in transverse jet is shown in figure. These numerical observations are in qualitative agreements for most of the jet-to-crossflow velocity ratios and can be extended to even lower velocity ratios. Topology of deformed vortex rings is very complex but one can identify tilted hairpin shapes around the leeward side of the jet.

Camussi et al (2002) analyzed PIV vector fields and flow visualizations to characterize the effect of jet-to-crossflow velocity ratios on the formation and evolution of large-scale vortices at very low jet Reynolds numbers. They observed that destabilization mechanisms are driven by the counter-rotating vortex pair (CRVP) and the jet flow is unable to induce Kelvin-Helmholtz instabilities on the shear layers. At low jet-to-crossflow velocity ratios (R<3), the longitudinal vorticity dynamics is dominated by the so-called wake-like structures (associated with negative vorticity outside the jet) connected to the streamwise CRVP that drive the destabilization of the jet flow. This destabilization leads to the formation of ring-like vortices (RLV) that are dominated by the jet-axis curvature and pairing of the counter-rotating vortices (Figure 5.80). The RLV are tilted and folded under the influence of the cross-stream and entrain the wall fluid. At large jet-to-crossflow velocity ratios, vortices with positive and negative vorticity are coupled together (Figure 5.81). For high velocity ratios, the curvature effects and the CRVP evolution are observed to play a less stringent role for the RLV formation.
Although the experimental visualization supports the hairpin structures in this flow configuration, the generation mechanism through destabilization of CRVP does not account for the jet vorticity at the leeward edge of the jet-delivery tube.

Figure 5.80 Schematic diagram to show the instability process leading to the formation of the ring-like vortices (a) Destabilization of the CRVP, (b) Pairing and formation of the RLV (Camussi et al, 2002)

Figure 5.81 Top-view of the jet flow visualized with dye at jet-to-crossflow velocity ratio, \( R \approx 2 \) (Camussi et al, 2002).

Coherent structures are extracted using positive pressure Laplacian criterion (Figure 5.82). It is very interesting to note that the shape of these vortices are in the form of hairpins or arches (similar structures were observed experimentally by Acarlar and Smith (1987) during the breakdown of the low-speed streak structures beneath the turbulent boundary layers). It is proposed that these vortices are the building blocks for this configuration. Previous studies have reported projected vorticity components on two-dimensional planes and attributed them to different coherent structures in the flow-field. Streamwise counter-rotating vorticity distribution is attributed to counter-rotating vortex
pair (CVP). Spanwise vorticity is attributed to roller vortices around the jet and crossflow interface. Wall-normal vorticity is attributed to upright wake vortices. The projection of a hairpin vortex on a streamwise plane will show a counter-rotating vorticity distribution corresponding to the legs of the hairpin structure. The projection of this structure on a spanwise centerplane will depict the vorticity associated with the head or arch of hairpin structure and will appear as a roller in a two-dimensional view. The projection of hairpin structures on the wall normal plane will show the signatures of the upright legs of hairpins connecting the arch. This unified perspective of the coherent structures simplifies and/or clarifies the understanding of the origin of these vortices and their subsequent evolution.

Figure 5.82 Coherent structures indicating packets of hairpin vortices in the wake of film-cooling jet a) Three-dimensional view, b) Top view.
5.5.6 Passive Scalar Mixing Interface

To understand the mixing and entrainment processes, the iso-surface for scalar dissipation rate are extracted from the time-dependent data sets (Figure 5.83). The level of the iso-surface is chosen to encompass the underlying hairpin structures, thus forming a mixing interface between the coolant and crossflow. The bumps on this surface correspond to the arch or head of the hairpin structure underneath. To detail the unsteady influence of these hairpin structures on mixing, a small station is chosen downstream of injection hole with dimensions just larger than a typical hairpin structure (~2D×2D×2D).

The time-dependence of the mixing interface defined by the scalar dissipation rate iso-surface can be evaluated through various geometric properties of this surface. The quantities of interest are the surface area of the mixing interface, average curvature of the interface, wrinkling of the surface and the entrainment across the interface. These details are presented for blowing ratio of 1.0 and are therefore, omitted here to avoid repetition.

Figure 5.83 Animations of the scalar dissipation rate iso-surface formed as the envelope to the hairpin coherent structures.
5.5.7 Blowing Ratio (M=1.0)

A uniform grid of $172 \times 102 \times 62$ is used to model the computational domain of size $17D \times 5D \times 6D$, where $D$ is the diameter of the coolant jet delivery tube. The film-cooled surface is placed at 1.0D from the bottom of the computational domain. The center of the jet injection hole at the film-cooled surface is 5D downstream from the inlet plane. The jet delivery tube is simulated as a cylindrical surface inclined at 35° in the streamwise direction (X) using Immersed Boundary Method. The body force terms are evaluated prior to the pressure-Poisson equation in the fractional step approach. These terms enforce no-slip conditions on the delivery tube surface (Yusof, 1996). The bottom plane of computational domain is treated using boundary conditions from a RANS study. The turbulence levels are generated using a Gaussian random number generator with variance corresponding to RANS turbulent kinetic energy. Top boundary of the computational domain is treated as freestream boundary. At the inlet, fully developed turbulent profile is specified. At the outflow, a convective boundary condition is used where the convection speed is obtained from the mass flux balance. The spanwise direction (Z) is assumed to be periodic.
5.5.8 Results (M = 1.0)

a) Streamwise component of velocity at Z/D = 0

b) Vertical component of velocity at Z/D = 0

c) Spanwise component of velocity at Z/D = 0.5

d) Non-dimensional temperature at Z/D = 0

Figure 5.84 Comparison of time-averaged LES predictions (lines) with experimental data (symbols) at different stations.
Comparison of time-averaged LES results with experimental data of Lavrich and Chiappetta (1990) is presented at various stations in the computational domain. The time-averaged statistics is obtained as the run-time average from the computation over approximately ten flow-through time periods (flow-through: time taken by crossflow to sweep the computational domain from inlet plane to exit plane). Streamwise component
of mean velocity field is underpredicted at X/D = 5.0 at the jet centerplane (figure 5.84a). Vertical component of velocity field as well as temperature is accurately predicted at this station (figures 5.84b and 5.84d). Also, the accurate prediction of spanwise component of mean velocity field at a plane passing through hole edge is an indicator that lateral spreading of mean flow is correct (figure 5.84c). Temperature field is overpredicted at X/D = 10 station near the film-cooled surface (for Y/D < 0.5). However, few discrepancies are noted and are attributed to insufficient time-averaging at farther downstream stations (X/D = 10).

To simplify the understanding of unsteady dynamics, different components of vorticity field are presented at respective projection planes (Figure 5.85a-d). Spanwise component ω₂ at Z/D = 0 shows roller vortices (negative vorticity patches) along the leeward edge of coolant jet. These roller vortices are shed regularly into the wake region and are convected downstream. Vorticity generated along the windward surface of hole delivery tube is weak (Figure 5.85a). To visualize upright vortices on the wall, vertical component ω₃ at Y/D = 0 is presented (Figure 5.85b). A symmetric street of vortex-pairs with opposite vorticity is observed in the wake region of coolant jet. This is clearly in contrast with the wake of a bluff body where such a pattern alternates. Streamwise component ω₁ is presented at X/D = 5 and 10 (Figure 5.85c-d) to show the counter rotating vortex pair (CVP). This coherent structure persists in the far field of jets-in-crossflow and is the only organized pattern in time-averaged mean velocity fields. Similar distribution of this vorticity component on streamwise projection planes leads to “almost always additive” contribution to the mean flow field structure. It is worth noting that most dominant contribution in the dynamics of these coherent structures is the
vorticity associated with jet boundary layer in the delivery tube. Roller vortices are attributed to the leeward edge of this boundary layer. CVP is associated with side edges of coolant jet boundary layer. Upright vorticity in wake vortices is primarily issued out of hole exit, however, there are evidences of contribution from crossflow entrainment in the downstream vicinity of jet injection (Figure 5.85b).

In an attempt to explain the flow physics better, coherent structures are extracted from time-dependent turbulent flow fields using positive iso-surface of Laplacian of pressure field (Wray and Hunt, 1989, Tanaka and Kida, 1993, Dubief and Delcayre, 2000). There are a number of criteria available to extract coherent structures from turbulent flow fields. Since the vortex cores are associated with strong vorticity and local pressure minima, it can be readily shown that positive surfaces of pressure Laplacian ($p_{kk}$ = $(\omega_i \cdot \omega_i)/2 – S_{ij} S_{ji}$) satisfy these requirements for identification of coherent structures. For incompressible flows, it is directly related to second invariant of the velocity gradient tensor. Simplicity and robustness of pressure Laplacian criterion for various problems is advocated here. There is some ambiguity associated with the value of positive levels since that depends on the problem. However, this ambiguity is easy to resolve through the analysis of flow fields and is not of much concern. Moreover, other methods involve much more computations for extracting coherent structures from time-dependent turbulent fields. A positive value of 0.7 of $p_{kk}$ yields packets of hairpin coherent structures (see figure 5.87 for more details). Therefore, a hairpin coherent structure is identified as the basic element of this flow configuration (Figure 5.86). The evolution of hairpin structure can explain the persistence of far-field structures and the unsteady vortices in various projection planes.
Figure 5.86 Details of the flow field in the vicinity of a hairpin vortex

To explain the morphological details and their impacts on evolutionary dynamics of hairpin coherent structure, several projected views as well as vorticity associated with them is presented in figure 5.86. The mean averages of such structures in the wake region on YZ projection plane explain the counter-rotating vortex pair (CVP) (normal in streamwise direction, Front view in figure 5.86). To illustrate this structure further, velocity vectors are shown at X/D = 5.7 plane. Also, helicity (\(=U_i \cdot \omega_i\)) associated with the legs of hairpin structure is presented. Although helicity is non-Galelian invariant property, yet it can be used to provide details of flow physics in inertial frames (as is the case with present computation). It provides the sense of rotation of fluid parcels as they move along with streamlines. The streamwise CVP is clearly associated with the legs of
hairpin structures. At almost all time instances, these legs are more or less located at similar \((y,z)\) coordinates on YZ projection planes. Therefore, mean or time-averaged fields as well as experimental visualizations will capture CVP. However, only phase averaged flow field can capture the upright wake vortices on XZ projection plane (normal in vertical direction, Top view in figure 5.86). These structures also explain symmetric shedding of vortices in the jet wake as compared to alternate shedding of vortices in the solid cylinder (or any bluff body) wake. Also, the entrainment of crossflow into the wake region right behind the jet injection is around these upright legs of hairpin structures. These legs can entrain crossflow streamwise vorticity and re-orient it into vertical component as the fluid parcel swirls around them (Yuan et al, 1999). Again, the phase average of flow fields would yield the roller vortices around the head of hairpin structures on XY projection plane (normal in spanwise direction, Side view in figure 5.86). As presented earlier, the train of roller vortices in figure 5.85a is a signature of heads of hairpin coherent structure packets in the wake region (figure 5.87). The entrainment of the crossflow fluid around the head of hairpin structures is expected to be a dominant contribution to the mixing processes in the wake region (also see figure 5.90). Note that absence of such rib or roller shaped vortices at the windward edge of jet is primarily due to the inclined injection of jet along the crossflow. The crossflow impacts the windward side of coolant jet and partially blocks this portion of jet vortex ring issuing out of delivery tube boundary layer. Velocity field induced by the arch of hairpin (head and upright legs) generates a backflow between the legs and below the head of hairpin. This velocity field can be evaluated from the vorticity field using Biot-Savart law (Saffman, 1992). It is similar to the magnetic field generated in the core of a solenoid when electric
current is passed through it. The backflow generated is the main mechanism of velocity
deficit in the wake region of coolant jet.

Generation of these hairpin vortices depends crucially on several flow parameters
and it can rationalize the view of numerous parametric studies. The deformation and
strength of issuing crossflow jet vortex ring primarily depends on the jet Reynolds
number, injection angle, jet-to-crossflow velocity ratio, upstream crossflow-to-jet
boundary layer thickness ratio and hole geometry (Haven and Kurosaka, 1997,
Andreopoulos and Rodi, 1984). The deformed jet vortex ring can lead to observed hairpin
vortices for inclined circular jets in crossflow. The lagrangian disturbances issuing out of
jet are hairpin shaped loci of local pressure minima. The streamwise spacing of these
disturbances and the Strouhal frequency of shedding of hairpin vortices can be related
through convection speed of these structures. The jet fluid wraps around the head and
streamwise oriented legs of these structures. The crossflow boundary layer is entrained
into the wake region and is lifted upright (i.e. re-oriented) around the vertical legs of the
hairpin structures. The experimental visualizations and measurements support the hairpin
structures in this flow configuration (Smith et al, 1991, Eiff and Keffer, 1997, Blanchard

Hairpin structures evolve while convecting downstream in the wake region and
their impact on entrainment and mixing of crossflow fluid with coolant fluid is
substantial. Mixing of temperature field is achieved around these hairpins due to large
temperature gradients created by these coherent structures (explained later, figure 5.90).
Hairpin structures entrain crossflow fluid into the wake region of jet and lead to the
formation of “hot spots” on the film-cooled surface. Migration of “hot spots” on the film-
cooled surface implies an intermittent coverage provided by the coolant jet. The time-averaged value of film-cooling effectiveness will be subjected to the variation in adiabatic wall temperature. Also, growth rate of individual hairpin coherent structure is related to its size and strength at the time of inception. Influence of these large scale coherent structures on the surface heat transfer process is presented in a time sequence (figure 5.87a-e). Again, coherent structures are extracted as a positive iso-surface of pressure Laplacian (= 0.7). Contours of non-dimensional temperature on the wall (Y/D = 0) are presented (Red = 1 and Blue = 0).

At t = t₀, five hairpin structures can be clearly identified (A-E), while hairpin structure F is in nascent stage. Hairpins are labeled alphabetically in the time-sequence of their generation. Thus, last coherent structure in computational domain (exit plane ~ 7D) is labeled A and this convention is followed while identifying hairpins towards coolant hole exit. Also, note that some of the hairpin structures (C and F) are not extracted completely by a single value of pressure Laplacian iso-surface. There is a “hot spot” on wall beneath the legs of hairpin D. Coolant jet is close to wall beneath hairpins A-C.

At t = t₀ + T, hairpin A has left the domain under consideration. Morphology of hairpins B and C has changed, while hairpin D has grown in size. “Hot spot” under the legs of hairpin D is traversing to the right of computational domain, phase-locked with coherent structure. There is a new “hot spot” generating beneath legs of hairpin E and head of hairpin F. Hairpin F has developed further and is followed by another hairpin G (nascent stage).

At t = t₀ + 2T, hairpin B has left the computational domain under consideration and hairpin C has moved close to this exit plane. Hairpin D has grown further in size and “hot
spot” beneath it has intensified. Hairpin E has grown in size and convected downstream. Also, the “hot spot” beneath head of hairpin F has intensified. Hairpin G has evolved to a well-formed structure and is followed closely by hairpin H.

At \( t = t_0 + 3T \), hairpin C has left the computational domain and hairpin D has moved to right along with “hot spot” underneath it. Hairpin E has convected to right and has grown in size. “Hot spot” beneath hairpin G has intensified. Note that a dense (streamwise closely placed) packet of hairpins can generate stronger “hot spot”. The influence of such packets of hairpins on entrainment process across an envelope around them will be investigated later (figure 5.91d). Hairpin H closely follows hairpin G and there is another hairpin I forming around 1D downstream of the coolant hole exit.

At \( t = t_0 + 4T \), hairpin D is at exit plane. Hairpin E has grown further in size. It is interesting to note that generally large hairpins are followed by small hairpins that do not grow in size (there are evidences of small “unlabeled” hairpin between D and E in all time instances). Hairpin F is following hairpin E but has moved over from the “hot spot” first generated beneath it. As explained earlier, large hairpin structures can generate substantial backflow and create velocity deficit in the wake region. Thus, small hairpin structures convect faster as compared to larger hairpin structures (hairpin C moved faster than hairpin D, hairpin F moved faster than hairpin E). Hairpin G has developed further and the entrainment around it has intensified the “hot spot” that is traversing to right below this coherent structure. Hairpin H moves to right, over the “hot spot” generated by preceding vortices. Hairpin I is now well-formed and there are evidences of formation of the head of another hairpin structure J (nascent stage).
Figure 5.87 Unsteady dynamics of coherent structures and their influence on wall heat transfer at different time instants a) $t_0$ (arbitrary), b) $t_0 + T$, c) $t_0 + 2T$, d) $t_0 + 3T$ and e) $t_0 + 4T$. (time gap $T$ is equal to 300 time steps (=1.5D/$U_j$). Arrows are tracking hairpin $E$ from one snapshot to another.
Animations of the coherent structures extracted from the flow-fields at higher blowing ratio (M= 1.0) show similar hairpin vortices (Figure 5.88). These vortices originate from the coolant delivery tube and therefore are comprised of coolant primarily. Their evolution in the downstream direction is a result of mixing of crossflow fluid with the coolant. These vortices are shed into the main flow and form packets of hairpins in the wake of the coolant jet. The interaction of the crossflow with coolant and the influence of these vortices on each other lead to the deformation via stretching and folding of these vortices. However, the time-averaged envelop of these packets will appear as a kidney shaped surface around the coolant which is commonly known as counter-rotating vortex pair (CVP) in the earlier studies.

Figure 5.88 Animations of the packets of hairpin vortices.
5.5.9 Passive Scalar Field Description

The details of instantaneous temperature field are given at several projected planes of the computational domain (figure 5.89a-d). The centerplane corresponds to \( Z/D = 0.0 \) and shows the mixing of the mainflow and the coolant jet. The coolant jet temperature drops in the downstream direction, however the coherent structures in the wake region retain their scalar value further (figure 5.89a). Coolant jet is lifted off the surface and there is crossflow fluid entrained beneath the jet downstream of the injection hole. The billows in the coolant-crossflow interface correspond to the heads of hairpin coherent structures. The temperature distribution corresponding to adiabatic wall boundary conditions also corresponds to film-cooling effectiveness (figure 5.89b). Again, the immediate decrease in film-cooling effectiveness is expected just downstream of the jet injection because coolant jet is lifted off the surface. However, the coolant associated with legs of hairpin structures would yield an increase in film-cooling effectiveness away from centerline. The coolant jet is closer to wall at farther downstream stations \( (X/D > 5.0) \) leading to a recovery in film-cooling effectiveness. The development of the coolant jet, its vertical penetration and lateral spread is shown at two different \( X/D \) locations (figures 5.89c-d). The coolant jet is observed to have a well defined kidney shaped structure with local maxima close to the core of CVP and large gradients near the crossflow-coolant interface. The crossplane mixing of scalar leads to the decrease in scalar value in the core of coolant jet at farther downstream stations.
Figure 5.89 Instantaneous non-dimensional temperature field on different projection planes.
To investigate mixing and entrainment process due to a single hairpin structure, an iso-surface of scalar dissipation rate ($= \Theta_k \cdot \Theta_k$) is extracted. The value ($=0.01$) is chosen such that this iso-surface forms an envelope over hairpin coherent structure (Figure 5.90). However, this measure of scalar dissipation rate does not account for subgrid stirring and mixing of scalar field and should be treated as a macro-scale mixing measure (Southerland et al, 1995). Also the scaling factor ($=1/(Re \cdot Pr)$) is omitted from the definition of scalar dissipation rate since it does not change any interpretation of results. Gradient of scalar field ($= \Theta_i$) is presented as vectors on different projection views of hairpin structure. In the top view, these vectors are seen to be directed just
beneath the head of hairpin structure. Similarly, the vectors converge beneath the hairpin head in side view. Absence of scalar gradients in crossflow fluid is observed in the front view around a projection plane around hairpin legs. Clearly, heat flux directed along these vectors on various projection planes indicates that there is a focus of heat flux below these large scale structures and that mixing is enhanced beneath the head of hairpin and between its upright legs.

Figure 5.91 Relative change in geometric properties from their respective mean value over observation period; a) Surface Area, b) Average Curvature, c) Wrinkling and d) Entrainment across “mixing interface” vs. Time.
A control volume of the size of a hairpin coherent structure is defined around X/D = 5 over the film-cooled surface. As packets of hairpin structures convect beneath this envelope in this control volume, they deform the scalar-dissipation iso-surface. The geometric properties of this “mixing interface” are presented as a function of time in figure 5.91a-d. Different geometric properties such as surface area of mixing interface, average curvature and wrinkling of the interface are evaluated (Geurts, 2002). The surface area of “mixing interface” increases whenever the head of hairpin is underneath it. Hairpin packets usually are formed as clusters with smaller hairpins following large hairpin coherent structure and there is some gap between such clusters (figure 5.87). Such clusters would lead to peaks and valleys in surface area of “mixing interface” as time progresses with a local minimum between such clusters (figure 5.91a and fig. 5.92). Note that local peaks of surface area diminish in size as hairpin cluster passes beneath this interface suggesting that growth of following hairpin vortices may be hindered by the leading hairpin vortex in the cluster. Average curvature of this surface (defined as the surface integral of local curvature) changes substantially (from -75% to +75% of mean value) during this time interval. The event that corresponds to this change is the relaxation of “mixing interface” between two hairpin clusters followed by its subsequent stretching by the following hairpin cluster. The entrainment across this interface can be evaluated approximately from difference in flux contributions using surface integral over the control volume that encompasses this “mixing interface” around hairpin structures (This is simply a result of Gauss divergence theorem to evaluate surface integral in a divergence-free (incompressible) flow field). The above-stated event of change in curvature is associated with reversal of entrainment process across interface (figure
5.91d). Also, the entrainment achieves local maximum over time interval corresponding to gap between hairpin clusters. Wrinkling (defined as the surface integral of absolute value of local curvature) is a relatively stable geometric property (usually varies between -10% to +10% of average mean value). It is a measure that is insensitive to “convexity” or “concavity” of mixing interface and implies that this interface maintains corrugations at almost all times. It is an important measure because small scale mixing is insensitive to curvature of mixing fronts and depends only on the absolute value of curvature.

Figure 5.92 Coherent structures underneath the mixing interface corresponding to respective time instants on the figure above (Figures corresponding to instants C and D yield similar information and hence not presented).
5.5.10 Conclusion

Large eddy simulations are performed for a simplified geometry representing film-cooling of a gas turbine blade surface and simulates an experimental study of Lavrich and Chiappetta (1990). Heat transfer calculations are also performed in a conjugate heat transfer mode to study the heat transfer on film-cooled wall. Following remarks summarize this study:

- Comparison of time-averaged LES predictions with experimental data of Lavrich and Chiappetta (1990) shows the adequacy of LES approach for film-cooling flows. Few discrepancies are noted at farther downstream stations. Insufficient averaging of time-dependent fields as well as uncertainty associated with boundary conditions can be regarded as main reasons for these deviations.

- Flow physics is explained in terms of components of vorticity field on respective projection planes in computational domain. All previously reported vortical structures i.e. CVP, roller vortices and upright wake vortices are identified (Kelso et al., 1996, Haven and Kurosaka, 1997, Fric and Roshko, 1994). Jet boundary layer vorticity is identified as the source of these vortices in the inclined jet in crossflow.

- Coherent structure extraction from instantaneous three-dimensional fields revealed packets of hairpin shaped vortices in the coolant jet. A unified perspective of previously reported vortices on different projection planes in this flow field is presented in terms of these basic hairpin coherent structures. CVP is shown to be associated with the legs of hairpin structure while roller vortices are linked to the head of hairpin structures. The upright legs are identified as the

- The dynamics of packets of hairpins in the wake region of injected jet and their influence on the unsteady wall heat transfer is presented. Generation of “hot spots” and their migration on the film-cooled surface is associated with the entrainment due to hairpin structures. Transient behavior of wall heat transfer under the influence of coherent structures reveals the inadequacy of any steady state RANS simulation even if it matches time-averaged film-cooling effectiveness.

- Scalar field distribution on different projection planes of computational domain revealed the correspondence with large scale coherent structures. Details of gradients of scalar field around a hairpin coherent structure showed the dynamical significance of such large scale vortices on the mixing process.

- Several geometric properties as surface area, average curvature and wrinkling of a “mixing interface” around hairpin coherent structures are presented to illustrate and quantify their impact on entrainment rates and mixing processes in the wake region.
Chapter 6  Internal Cooling of Gas Turbine Blades

6.1 Introduction

Modern gas turbines operate at very high turbine inlet temperatures for better second law efficiency and specific thrust. However, such increased thermal loads can deteriorate the blade life in a rotating environment. These blades are internally cooled by using the serpentine channels with turbulators inside the blade to enhance the heat transfer (Figure 6.1). The increment in heat transfer due to rib turbulators as compared to the increased pressure drop in the channel is a crucial design parameter (Morris, 1981). The problem is complicated further due to the interplay of Coriolis forces and buoyancy forces. Several experimental investigations to study the effect of centrifugal buoyancy, rotation number and Reynolds number have been performed (Wagner et al., 1992). However, a numerical study can provide much more detailed information on flow physics as well as heat transfer in such situations. The secondary flow in non-circular duct without rotation is generated due to the anisotropy in turbulent stresses. In rotating ducts, the Coriolis forces give rise to secondary flow as well. Again, buoyancy forces can generate secondary flow field to enhance the crossplane mixing. Iacovides and Launder (1995) reviewed CFD studies related to internal cooling passages of gas turbines and concluded that low-Reynolds number modeling for the sublayer region is essential for such flows. Turbulence modeling using two-equation models can not capture essential physics due to isotropic nature of modeled normal turbulent stresses. Morris and Rahmat-Abadi (1996) conducted experimental investigation on rotating ribbed circular ducts and proposed that Nusselt number correlations should depend on the quotient of buoyancy parameter and rossby number to uncouple the effect of Coriolis forces from centrifugal buoyancy forces.
Naimi and Gessner (1997) calculated fully developed turbulent flow in rectangular ducts with ribs on opposite walls using three different turbulence models and noted some spurious secondary flow features in predictions as compared to the experimental data. Bredberg (1997) documented a literature survey of experimental as well as numerical studies on turbine blade internal cooling ducts. Iacovides (1998) presented a comparison of several low-Reynolds number eddy viscosity models with low-Reynolds number second moment closure models for internal coolant passage flow and heat transfer. Second moment closure has some promise in that direction (Iacovides and Raisee (1999), Saidi and Sunden (1999), Hermanson et al. (2001), Jang et al (2001)). Bonhoff et al (1999) performed stereoscopic PIV for 45° ribs in coolant channels and compared it with several turbulence model predictions. Murata et al (2000, 2001) performed a series of LES studies to understand the unsteady dynamics of various flow structures on the heat transfer in internal coolant ducts. Pallares et al (2001) analyzed and simplified LES momentum budgets for the flow field in rotating square ducts. Roclawski (2001) conducted PIV measurements for channel flow with multiple rib arrangements and Roclawski et al (2001) presented modeling based on Discrete Dynamical System (DDS) concepts for such flows. Yamawaki et al. (2002) presented local heat transfer measurements using thermochromic liquid crystals on a flat plate subjected to rotation and analyzed turbulent stress equations for their influence on mean momentum transport. Miyake et al (2002) carried out DNS of a channel with one ribbed wall and presented the evolution of coherent structures in the vicinity of rough wall. In this research study, Large Eddy Simulations (LES) were performed to study the flow physics and heat transfer in a rotating ribbed duct. This numerical study simulates the experiments conducted to
investigate the effects of buoyancy and Coriolis forces on heat transfer in a turbine blade internal coolant passages (Wagner et al., 1992).

Figure 6.1 Typical turbine blade internal cooling configuration (Wagner et al., 1992)
6.2 Governing Equations

The non-dimensional governing equations for conservation of mass, momentum and energy for an incompressible Newtonian fluid in LES methodology are as follows:

\[
\frac{\partial U_j}{\partial x_j} = 0
\]

\[
\frac{\partial U_j}{\partial t} + \frac{\partial U_i U_j}{\partial x_i} = -\frac{\partial p}{\partial x_i} \delta_{ij} + \frac{1}{Re} \frac{\partial^2 U_j}{\partial x_i^2} + \frac{\partial \tau_{ij}}{\partial x_j} - 2Ro\epsilon_{ijk}\Omega_j U_k + Bo\left(1 - \frac{\Theta}{\Theta'^{\prime}}\right)e_{ijkl}\delta_{il}\delta_{kj}
\]

\[
\frac{\partial \Theta}{\partial t} + \alpha \frac{\partial \Theta}{\partial x_j} = \frac{1}{Re Pr} \frac{\partial^2 \Theta}{\partial x_j^2} + \frac{\partial q_j}{\partial x_j}
\]

where \( U_j \) is the filtered velocity field, \( \Theta = \frac{T - T_{ref}}{T_w - T_{ref}} \) where \( T_w \) is the wall temperature and \( T_{ref} \), \( T_{ref} \) are yet undefined reference temperatures. The mean pressure gradient in flow direction is \( dP/dz \). Therefore, \( p \) is the periodic component of the pressure field. \( \delta_{ij} \) is the Kronecker delta tensor. \( \epsilon_{ijk} \) is the alternating tensor. The distance vector can be written as \( r_i = R_m \delta_{ij} + x_i \), where \( R_m \) is the mean radius of the periodic module from the rotation axis. The important parameters for such flows are Reynolds number \( (Re = U_m D_h/\nu) \), rotation number \( (Ro = \Omega D_h/U_m) \) and centrifugal buoyancy number \( (Bo = \frac{\beta(T - T_m)}{\nu \alpha} \Omega^2 R_m D_h^3 \left( \frac{\alpha}{\nu} \right) \left( \frac{\nu}{U_m D_h} \right)^2 = \frac{Ra}{Pr Re^2} \) \). The SubGrid Scale (SGS) stress tensor and SGS scalar flux vector are given by \( \tau_{ij} \) and \( q_j \) respectively. In this study, Dynamic Mixed Model (DMM) is used to model these SGS stress tensor and scalar flux vector (Moin et al., 1991, Vreman et al., 1994). The box filters are used in the Germano identity for the calculation of dynamic coefficient and for the calculation of Leonard stresses. The dynamic coefficient is test filtered to avoid numerical instabilities.
Treatment of the non-dimensional temperature in the periodic direction needs special attention. Patankar et al. (1977) described a method to solve the uniform heat flux (UHF) and uniform wall temperature (UHT) problems in the ducts with periodic cross-sections for steady situations. Wang and Vanka (1989) presented an iterative procedure to calculate $\lambda$. However, this parameter can be calculated directly for explicit schemes. Most of the simulations were performed using non-dimensionalization with respect to friction velocity and uniform heat flux case. As it will be explained later, that renders the sink terms in momentum and energy equations, i.e. $dP/dz$ and $\lambda$, constant. In experiments, usually the mass flow rate and wall temperatures are control parameters, therefore reference velocity should be the average velocity and $\lambda$ is no more constant. For unsteady heat transfer calculations in periodic geometries, the following simplifying assumption is usually invoked

$$T(x_i + L\delta_{j_3}, t) = (1 - \lambda)T_{ref} + \lambda T(x_i, t)$$  \hspace{1cm} (6.2)

$T_{ref}$ is a reference temperature or flux (in appropriate units) for the problem (figure 6.2). The scaling factor $\lambda$ can at most be function of time. The non-dimensional temperature variable can now be defined as follows

$$\Theta = \frac{T - T_{ref}}{T_{wall} - T_{r2}}$$  \hspace{1cm} (6.3)

Here $T_{wall}$ can be function of time and wall-tangential directions and $T_{r2}$ is another constant reference temperature (it relates wall phenomenon in post-processing of non-dimensional field, thus it is not a crucial parameter except it should not be set to $T_{wall}$).
Figure 6.2 Relationship between non-dimensional temperature field at periodic stations

For constant wall temperature, we can see that $T_{ref}$ is equal to wall temperature and it leads to a simple homogeneous boundary condition for $\Theta$ (i.e. zero on the wall). Also the denominator is merely a constant. Calculation of bulk temperatures at various streamwise locations show that $\Theta_b = \frac{T_b - T_{ref}}{T_{wall} - T_{r2}}$

Since $T_b(z + L, t) = (1 - \lambda)T_{ref} + \lambda T_b(z,t)$, it can be shown that

$$\frac{\Theta_b(z + L)}{\Theta_b(z)} = \lambda \quad (6.4)$$

At geometrically periodic planes, the following relation is obtained

$$\frac{\Theta(x_i + L\delta_{ij}, t)}{\Theta(x_i, t)} = \lambda \quad (6.5)$$

For constant heat flux, we can see that $T_{ref}$ is equal to $\frac{q_w D}{k}$ and it leads to a simple homogeneous boundary condition for $\frac{\partial \Theta}{\partial \eta}$ (i.e. zero on the wall). The calculation of $\lambda$ is done in a similar fashion as described above. Thus, this non-dimensional temperature
assumes existence of a reference temperature and a reference driving potential in the form of heat flux or applied temperature drop. Therefore, simple energy balance and periodicity of surface phenomenon can yield the relations for these reference values.

Boundary conditions for non-dimensional temperature in the periodic direction is written as (using 6.4 and 6.5)

$$\frac{\Theta^0}{\Theta_b^0} = \frac{\Theta^L}{\Theta_b^L}$$  \hspace{1cm} (6.6)

where superscript indicates the z-location and subscript \(b\) denotes the bulk non-dimensional temperature. Differentiating the periodic boundary condition in the wall-normal direction we get

$$\frac{1}{\Theta_b^0} \left( \frac{\partial \Theta^0}{\partial X} \right) = \frac{1}{\Theta_b^L} \left( \frac{\partial \Theta^L}{\partial X} \right)$$  \hspace{1cm} (6.7)

This is equivalent to enforcing periodicity on the Nusselt number in a periodic geometry.

- **Uniform Heat Flux (UHF) case:**

$$\left( \frac{\partial \Theta^0}{\partial \eta} \right) = \left( \frac{\partial \Theta^L}{\partial \eta} \right) = q_w$$  \hspace{1cm} (6.8)

$$\therefore \Theta_b^0 = \Theta_b^L$$

Clearly, setting \(Tr_2\) to \(T_b\) will render the denominator as a constant. Moreover, the independence of non-dimensional bulk temperature from periodic direction implies that \(T_{ref}\) is equal to \(T_b\). Therefore, the scaling at the inlet plane and periodicity of Nusselt number can uniquely determine the non-dimensionalization and the sink term in the energy equation. Also, this sink term is independent of time because heat addition to the domain is constant at all time instants.
Uniform Wall Temperature (UWT) case

From the energy balance, one can write

$$
\rho c_p \bar{U}_{\text{avg}} \left( T_b^L - T_b^0 \right) A_c = \int_S q_w dS
$$

$$
\int_S q_w dS \approx \sum_{\text{in}} q_w dS + \frac{\sum (q_w^0 + q_w^L) dS}{2}
$$

$$
\rho c_p \bar{U}_{\text{avg}} \left( T_w - T_2 \right) \left( \Theta_b^L - \Theta_b^0 \right) A_c \approx k(T_w - T_2) \mu \left[ \sum_{\text{in}} \left( \frac{\partial \Theta}{\partial \eta} \right)_w^0 dS + \sum \left( \frac{\partial \Theta}{\partial \eta} \right)_w^L dS \right] + \sum \left( \frac{\partial \Theta}{\partial \eta} \right)_w^0 + \frac{\left( \frac{\partial \Theta}{\partial \eta} \right)_w^L}{2} dS
$$

Here $\eta$ is the wall normal direction and $dS$ is the differential area element on the wall.

For the square channel, we use Nusselt number periodicity to define the flux at the inlet in terms of the flux at the exit as

$$
\left( \frac{\partial \Theta}{\partial \eta} \right)_w^0 = \left( \frac{\partial \Theta}{\partial \eta} \right)_w^L
$$

$$
\tilde{\lambda} = \frac{\Theta_b^0}{\Theta_b^L}
$$

Using these relations in the energy balance, we get

$$
\left( \Theta_b^L - \lambda \Theta_b^L \right) A_c \approx \left( \frac{1}{\text{Re} \cdot \text{Pr}} \right) \left[ \sum_{\text{in}} \left( \frac{\partial \Theta}{\partial \eta} \right)_w^L dS + \sum \left( \lambda \left( \frac{\partial \Theta}{\partial \eta} \right)_w^L + \left( \frac{\partial \Theta}{\partial \eta} \right)_w^L \right) dS \right]
$$

$$
\therefore \lambda \approx \left\{ \Theta_b^L + \left( \frac{1}{\text{Re} \cdot \text{Pr}} \right) \left[ \sum_{\text{in}} \left( \frac{\partial \Theta}{\partial \eta} \right)_w^L dS + \sum \left( \frac{\partial \Theta}{\partial \eta} \right)_w^L \right] \right\}
$$

To enforce the validity of scaling relation up to the wall, we choose $T_1$ equal to $T_w$. Therefore, the non-dimensional temperature is zero at the wall and the scaling ensures the periodicity of the Nusselt number in the periodic geometries.
6.3 Problem Description and Computational Method

The first set of computations are performed at a Reynolds number ($Re$) of 12,500 based on average velocity in the duct and the hydraulic diameter of the square duct. The rotation number ($Ro$) is 0.12 and the inlet coolant-to-wall density ratio ($\Delta \rho / \rho$) is 0.13. The rib height-to-hydraulic diameter ratio ($e/D$) is 0.1 and the rib pitch-to-height ratio ($P/e$) is 10. The ribs are square in cross-section and are placed transverse to the flow in the duct (figure 6.3). This numerical study simulates experiments of Wagner et al (1992) conducted to study the effects of buoyancy and Coriolis forces on heat transfer in turbine blade internal coolant passages.

Figure 6.3 Schematic of the computational domain
6.4 Results and Discussion

The three-dimensional spectrum of the instantaneous flow field is shown in figure 6.4a. The grid resolution is sufficient to capture the energy producing events as well as the portion of the inertial subrange. A peak in energy spectrum is also observed around the wave number corresponding to a length scale \(l/D = 0.16\). Clearly, this can be attributed to the energy production by vortex shedding behind the ribs \((e/D = 0.1)\). Figure 6.4b shows the variations of the instantaneous flow rate. Note that, to maintain an average flow rate, a mean constant (in space) pressure gradient is applied. Superimposed on this mean pressure gradient are temporal variations corresponding to the dominant Strouhal frequency. Instantaneous pressure fluctuations are linked to vortex shedding behind ribs and due to other sources of unsteadiness. This pulsation causes the flow rate to vary in time with the variation dominated by the vortex shedding frequency (figure 6.4b). However, the average flow rate is always maintained very close to 1.0 as desired.

![Figure 6.4 (a) Three dimensional energy spectrum of the flow field (b) Flow rate vs time.](image)

193
The influence of Coriolis forces and Centrifugal buoyancy on the ensemble averaged fields is presented in table 6.1 on plane normal to the generators of ribs. Also, the leading terms in turbulent stresses and turbulent scalar flux are listed for the present configuration. The effect of Coriolis force is to direct secondary flow from the leading wall to the trailing wall. In a duct, such a distribution will result in mean counter-rotating vortices along the flow direction. The Coriolis force contribution for normal stresses is produced by the turbulent shear stress which in turn is produced by the difference in the anisotropy of normal stresses. It is an interesting balance because turbulent shear stress contribution is reducing the difference between turbulent normal stresses. The mean scalar field is affected through turbulent scalar fluxes only. Centrifugal buoyancy terms are significantly larger in streamwise mean momentum as compared to normal to the leading or trailing wall (They are still smaller than Coriolis contributions for the parameters chosen for this problem).

Table 6.1 Influence of Coriolis force and Centrifugal buoyancy on different mean variables.

<table>
<thead>
<tr>
<th>Mean Variable</th>
<th>Coriolis Force</th>
<th>Centrifugal Buoyancy</th>
</tr>
</thead>
<tbody>
<tr>
<td>W</td>
<td>$2\Omega U$</td>
<td>$-\beta\Omega^2 z\Theta$</td>
</tr>
<tr>
<td>U</td>
<td>$-2\Omega W$</td>
<td>$-\beta\Omega^2 x\Theta$</td>
</tr>
<tr>
<td>$\Theta$</td>
<td>_</td>
<td>_</td>
</tr>
<tr>
<td>$-u'w'$</td>
<td>$2\Omega (w'w'-u'u')$</td>
<td>$\beta\Omega^2 (xw'\theta'+zu'\theta')$</td>
</tr>
<tr>
<td>$w'w'$</td>
<td>$4\Omega u'w'$</td>
<td>$-2\beta\Omega^2 zw'\theta'$</td>
</tr>
<tr>
<td>$u'u'$</td>
<td>$-4\Omega u'w'$</td>
<td>$-2\beta\Omega^2 xu'\theta'$</td>
</tr>
<tr>
<td>$-w'\theta'$</td>
<td>$-2\Omega u'\theta'$</td>
<td>$\beta\Omega^2 z\theta'^2$</td>
</tr>
<tr>
<td>$-u'\theta'$</td>
<td>$2\Omega w'\theta'$</td>
<td>$\beta\Omega^2 x\theta'^2$</td>
</tr>
</tbody>
</table>
The instantaneous snapshots of the streamwise component of vorticity field at three different streamwise stations \((z/D = 0.25, 0.50 \text{ and } 0.75)\) depict the complex flow field in the duct (figure 6.5). At \(z/D = 0.25\), the rib is placed on the leading wall and sheds vortices into the mainflow with significant streamwise vorticity. The boundary layer on the trailing wall is highly turbulent and intensified. The local increase in velocity magnitude near the trailing wall is due to decrease in cross-sectional area as well as body forces. At the streamwise centerplane \(z/D = 0.50\), the recirculation region behind the rib on the leading wall contains intense vortices along the flow. The trailing wall vortices are gathered towards the center of the duct due to secondary flow in the crossplane of the duct. At \(z/D = 0.75\), the rib is placed on the trailing wall and interacts with the oncoming turbulent boundary layer vortices. In this snapshot, the vortices are pushed towards the center of duct by the rib as well as the secondary flow. The counter-rotating vortex over the rib enhances the entrainment of coolant from core of duct to trailing wall (see figure 6.10). The leading wall boundary layer shows a lot of activity too. The vortices at the front and the back wall do not penetrate into the core flow to the similar extent as the leading and trailing wall vortices.

![Figure 6.5 Streamwise component of instantaneous vorticity \(\omega_z\) at three XY planes](image)

Figure 6.5 Streamwise component of instantaneous vorticity \(\omega_z\) at three XY planes
Time-averaged velocity vectors at \( Y/D = 0.5 \)

Figure 6.6 Time-averaged velocity vectors and details of flow field near the ribs at the \( Y/D = 0.5 \)

Time-averaged velocity field shows the skewed profile (Figure 6.6). The boundary layer on the trailing wall (unstable) is much steeper than on the leading wall (stable). The details near the ribs show the difference in the size of recirculation regions in front and behind the ribs. The front recirculation region for rib B is smaller than rib A. The flow attaches over the top of rib B (on the trailing wall) while it remains detached on the top face of rib A (on the leading edge). This effect is primarily due to Coriolis forces that are directing mean flow towards the trailing wall. The impingement of oncoming flow on the front of ribs results in high heat transfer rates. The lack of coolant fluid in recirculation behind the ribs will create “hot spots”. However, these recirculation regions are accompanied with enhanced crossplane mixing and the flow re-attaches to the walls between the ribs.
Time-dependent animations of non-dimensional temperature field are shown at the first cell node over the corresponding walls (Figure 6.7). There is more coolant accumulation on the trailing wall as compared to the leading wall (note the difference in the range). The heat transfer is enhanced on the trailing wall by a factor of two approximately. The temperature field show streaks correlated with the streamwise component of vorticity on these walls. The coolant fluid in front of the ribs increases heat transfer in the stagnation (front recirculation) region. In the leeward recirculation regions, the non-dimensional temperature is close to wall temperature. The temperature distribution on the front and back wall is similar. However, there is more coolant near the back wall as compared to front wall (and it is observed throughout the computational duration). This might be caused by a low frequency mode in the coreflow. Again, the temperature field correlates with the streamwise streaks of coolant fluid on these walls. A time-sequence for wall-normal vorticity dynamics and temperature field is presented for the extended module to demonstrate that time-dependent flow indeed allows larger than pitch wavelengths (Figure 6.11a-e). Therefore, the need to elaborate on such time-sequence descriptions for single module is deemed unphysical.

Figure 6.7 Animations of temperature field on the walls of the duct a) Trailing wall, b) Leading wall, c) Back wall and d) Front wall.
Comparison with experimental data is fair (noting that few discrepancies remain and the error range is quite large for the experimental setup, Table 6.2). It is interesting to note that low frequency perturbations of the flow field can render the difference on the sidewall averaged Nusselt numbers. The possibility of such frequencies corresponding to rotation number can not be ruled out and therefore, very long averaging periods are needed.

Table 6.2 Comparison of the Averaged Nusselt Number with Wagner et al (1992).

<table>
<thead>
<tr>
<th>Average Nusselt Number</th>
<th>Computed</th>
<th>Experimental</th>
</tr>
</thead>
<tbody>
<tr>
<td>Leading Wall</td>
<td>53.24</td>
<td>56.51(±20%)</td>
</tr>
<tr>
<td>Trailing Wall</td>
<td>101.73</td>
<td>124.5(±15%)</td>
</tr>
<tr>
<td>Side Wall 1</td>
<td>85.57(?)</td>
<td>66.48(±15%)</td>
</tr>
<tr>
<td>Side Wall 2</td>
<td>75.07</td>
<td>63.16 (±15%)</td>
</tr>
</tbody>
</table>
6.5 Simulations in Larger Computational Module

As next step of this study, the computational module is extended in the streamwise direction to resolve the issue concerning the size of large coherent structures in a single periodic “geometric” module. Evolution of coherent structures in this larger computational module clearly shows the pitfalls of periodic assumption for the unsteady simulations in a single pitch module. The computational module is extended to double the size in streamwise direction to study the influence of large coherent structures on heat transfer and possibly eliminating the low-frequency oscillations that will be resolved on this computational domain. As expected, large computational domain convincingly established the presence of larger wavelengths than one pitch size. However, the statistical averages are not significantly different for the computational runtime to observe the effect of large wavelength modes. Distribution of temperature in the duct depends strongly on mixing generated by large coherent structures (see table 6.1). These coherent structures are in turn subjected to body forces due to Coriolis force and centrifugal buoyancy. Coherent structures evolve dynamically as they convect down through the duct. Moreover, these structures could possibly be subjected of Taylor-Gortler like instability due to concave curvature of mean streamlines. To understand the flow physics and heat transfer processes in detail, time-sequences of temperature field and vorticity component normal to different cross-sectional projection planes of the duct are presented.

In figure 6.10a-e, streamwise component of vorticity and temperature field are presented in a time-sequence. Various walls of the duct are labeled as LW: leading wall, TW: trailing wall, LSW: left side wall and RSW: right side wall. Starting time instant $t_0$
is arbitrary and time gap, $T$ equals 150 timesteps ($=0.15\tau$) in this time sequence. At time $t_0$ (figure 6.10a), there is abundance of intense vortices near trailing wall and thermal boundary layer is also thinner near trailing wall. The accumulation of coolant near trailing wall is a direct consequence of Coriolis forces in this plane. Note that the streamwise vorticity appears as counter-rotating vortices over wall layer vortices. Thermal plume in the center of duct near trailing wall is associated with this counter-rotating vortex pair. These counter-rotating vortices increase mixing on this cross-sectional plane. Counter-rotating vortex pair on this plane entrains coolant from the core of duct towards the trailing wall. Also, these vortices are traveling along mean streamlines with concave curvature and hence, are subjected to Taylor-Gorter like instability. At time $t_0 + T$ (figure 6.10b), counter-rotating vortex pair has intensified and its influence on thermal field is noted through the deformation of plume in the center of duct. At this instant, thermal boundary layer on leading wall has also grown. It is associated with the penetration of vortices into the core region. The upward motion of positive vorticity along right side wall is attributed to the Coriolis forces in this plane. At time $t_0 + 2T$ (figure 6.10c), thermal plume on the center of trailing wall is well mixed by counter-rotating vortex. There is another such pair near left side wall that can be associated to the formation of second thermal plume on trailing wall. On leading wall, there is accumulation of vortices around the center. This results in growth of thermal plumes around the corner of leading wall due to lack of mixing. These plumes appear as fingers penetrating into the coolant core.
Figure 6.10 Time-sequence of vorticity dynamics and temperature field on cross-sectional plane at $Z/D = 1.0$. Streamwise vorticity (left) and temperature field (right).
At time $t_0 + 3T$ (figure 6.10d), counter-rotating vortex has diminished in intensity and as a result, thermal plume around the center of trailing wall has increased temperature. Secondary vortex pair has moved to the corner closer to left side wall. Coolant is closer to left side wall at this instant. As noted at earlier time instant, vorticity accumulates between the center and corners of leading wall. At time $t_0 + 4T$ (figure 6.9e), vortices near the corner of left side wall and trailing wall are moving upwards due to Coriolis forces resulting in the growth of thermal boundary layer on left side wall. Counter-rotating vortex on the center of trailing wall has diminished further and thermal plume has intensified due to lack of mixing there. Animations of streamwise vorticity component and temperature field are presented in figures 6.14d and 6.15d respectively.

Attention is next turn to the dynamics of spanwise vorticity and temperature field on a cross-section plane through the center of duct at $Y/D = 0.5$ (figure 6.11a-e). In this time-sequence, time-gap is 375 timesteps ($=0.375\tau$). At time $t_0$ (figure 6.11a), spanwise vorticity is mostly concentrated in the roller vortices that are shed from the ribs. There is a roller vortex on the top of first rib on trailing wall. Ribs on the leading wall generate spanwise vorticity in the opposite sense and thermal plumes emanating from ribs in the separation region indicate low heat transfer rates. Temperature field distribution shows the presence of more coolant on trailing wall, resulting in thin thermal boundary layer and large heat transfer rates. There is an impingement of coolant in front of the first rib on trailing wall. Most of the LES studies performed for internal cooling problems utilize only a single pitch module. However, it is demonstrated here that for unsteady simulations, enforcing spatial periodicity to be the same as geometric periodicity for low-blockage ratio ducts may not be a physical assumption.
Figure 6.11 Time-sequence of vorticity dynamics and temperature field on cross-sectional plane at $Y/D = 0.5$. Spanwise vorticity (left) and temperature field (right). Arrows on the snapshots track vortices and entrainment interface of scalar field.
At time $t_0 + T$ (figure 6.11b), roller vortex is between the ribs on trailing wall in the core of duct (below the second rib on leading wall). Most of the vortices shed from ribs on trailing wall remain close to the wall due to Coriolis body force (directed from leading to trailing wall on this plane). Vortices that reach the core of duct enhance mixing of scalar field there. Engulfment of coolant near the first rib on leading wall is observed and is related to the entrainment associated with vortices that were shed from a rib in preceding such module (periodicity in streamwise direction is enforced, therefore coherent structures that leave the domain are equivalent to the ones that would enter this domain from a preceding module). At time $t_0 + 2T$ (figure 6.11c), reference roller vortex is above the second rib on trailing wall and has diminished in intensity. There is impingement of coolant in front of both the ribs on trailing wall. Entrainment around leading wall appears as fingers penetrating into the core of duct and these fingers have convected downstream to the station between the ribs on leading wall. At time $t_0 + 3T$ (figure 6.11d), reference vortex has “re-entered” the computational domain due to the streamwise periodicity. Physically, it represents a similar vortex shed from a preceding ribbed module. The reference location on entrainment interface is below the second rib on leading wall. The interface is more diffuse than previous time instants due to mixing of scalar field. At time $t_0 + 4T$ (figure 6.11e), reference vortex has completed one flow-through in duct and has faded in intensity (It was possible to track this vortex at this instant only by observing complete animation frame-by-frame). Entrainment interface fingers (or billows) are past the second rib on leading wall and completely diffused due to mixing. Animations of spanwise vorticity component and temperature field are presented for longer duration in figures 6.14c and 6.15c.
Figure 6.12 Time-sequence of vorticity dynamics and temperature field over trailing wall at X/D = 0.125. Wall-normal vorticity (left) and temperature field (right)
Time sequence of evolution of wall-normal vorticity component and temperature field above the trailing wall is presented in figure 6.12a-e. Since these vortices decorrelate very rapidly in time, time gap between snapshots is chosen as 225 timesteps (≈0.225τ) (see animations). At time $t_0$ (figure 6.12a), distribution of vorticity on the windward surface of the ribs suggest that there is convergence region around the trailing wall centerline. Coolant moves along rib face towards the rib and side-wall junction corners and the centerline of trailing wall. Convergence of fluid around centerline and reorientation of wall-normal vorticity into streamwise direction over the ribs results in counter-rotating vortex pairs (figure 6.11a-e). There is large amount of coolant in front of the first rib on trailing wall at this instant. Although there are separation and recirculation regions behind the ribs, these regions are very turbulent and active zones for fluid mixing. At time $t_0 + T$ (figure 6.12b), coolant accumulation in front of the ribs is attributed to impingement of core due to Coriolis body force. Coolant fluid traverses over trailing wall and breaks the roller vortices generated by ribs around the centerline. At time $t_0 + 2T$ (figure 6.12c), coherent structures have lifted off the surface leading to disappearance of projected vorticity component on trailing wall at this instant. This also results in decreased coolant amount on the trailing wall. At time $t_0 + 3T$ (figure 6.12d), wall-normal component of vorticity evolves in front of second rib on trailing wall in a similar fashion as in front of first rib at earlier time instants. Migration of “hot-streaks” on trailing wall is linked to motion of coherent structures (Figure 6.16). At time $t_0 + 4T$ (figure 6.12e), coolant amount in front of first rib has decreased while it has increased in front of second rib. Animations of wall-normal vorticity dynamics and temperature field on trailing wall are presented in figures 6.14a and 6.15a.
Figure 6.13 Time-sequence of vorticity dynamics and temperature field over leading wall at X/D = 0.975. Wall-normal vorticity (left) and temperature field (right)
Time sequence for the evolution of wall-normal vorticity component and temperature field above the leading wall is presented in figure 6.13a-e. Since these vortices decorrelate very rapidly in time, they can be tracked meaningfully only for short durations (see animations). Same time gap and initial time instant $t_0$ is chosen here as in figure 6.12a-e. At time $t_0$ (figure 6.13a), streaks of wall-normal vorticity are associated with the presence of coolant. Coolant is present right in front of the ribs on leading wall. There are “hot-streaks” on the leeward side of ribs due to coolant flow separation. Rib and side-wall junctions are dead zones for the fluid in rotating ribbed duct. Vorticity on windward surfaces of the ribs shows that the flow is converging from rib corners towards rib face center. Convergence of flow along with stagnation in streamwise direction would result in flow over the rib around windward face center of ribs. At time $t_0 + T$ (figure 6.13b), vortices in front of the second rib are accumulating towards center. Breakup of wall-normal vorticity streaks into small blobs (or patches) results in growth of thermal boundary layer. At time $t_0 + 2T$ (figure 6.13c), influence of coolant has diminished further, however there is coolant impingement in rib-side wall corner in front of the first rib. Coherent structures are lifted off the leading wall while migrating over the surface. This leads to disappearance of blobs and breakup of streaks of projected vorticity component on leading wall. At time $t_0 + 3T$ (figure 6.13d), coolant is present around centerline of leading wall behind the second rib. Note that vorticity in the separation region just behind the ribs has slower evolution as compared to the distribution in front of the ribs. Clearly, unsteady heat transfer on leading wall is important in these regions only. At time $t_0 + 4T$ (figure 6.13e), coolant noted at earlier instant “re-enters” computational
domain due to enforced streamwise periodicity. Vorticity patches have decorrelated by
this time instant owing to their disappearance from this plane.

Figure 6.14 Animations of normal component of vorticity to the corresponding planes a) Trailing Wall, b) Leading Wall, c) Spanwise Centerplane and d) Streamwise Centerplane
Figure 6.15 Animations of non-dimensional temperature field on a) Trailing wall, b) Leading Wall, c) Spanwise centerplane and d) Streamwise centerplane.
For the sake of completeness, animations of normal component of vorticity field and temperature field on respective planes in computational domain are also presented (figures 6.14 and 6.15). These animations illustrate projected vorticity of large coherent structures and associated scalar mixing during the evolution of coherent structures on these planes for a longer duration of time.

Concentration of vortical structures near the trailing wall increases the heat transfer dramatically. The vortices are shed from the ribs and subsequently distorted by the mainflow and the secondary flow to result in arch or hairpin shape vortical structures. Coherent structures extraction from the time-dependent flow-fields is done by rendering positive iso-surfaces of pressure Laplacian (Figure 6.16). For the sake of clarity, different positive levels were chosen for the leading and trailing walls. Top views are also presented for these two walls. There is myriad of coherent structures due to shear layer of ribs and wall boundary layers. The complexity of coherent structure dynamics is increased further by Coriolis force, Centrifugal buoyancy and strong variations of pressure gradients between the ribs. Separation and recirculation regions behind the ribs change the morphology and evolution of hairpin shaped structures beneath the roller vortices of shear layer. Coherent structures are shed from the ribs in the shape of roller vortices. These structures near the leading and trailing walls are broken around spanwise centerplane due to secondary flow (Coriolis forces). However, the influence of side walls and secondary flow provides different convection velocity along spanwise length of these roller vortices. These coherent structures converge towards spanwise centerplane and breakup into smaller structures in front of the rib of the next module. Moreover, some smaller vortices are also seen beneath these large structures. These structures are
produced primarily at the wall and evolve under the influence of induced flow field of large structures. Size of coherent structures can be of the order of rib-pitch. Therefore, most of the single rib-pitch module calculations can not capture such large energy containing scales accurately. These coherent structures must be resolved in a larger computational domain. Evolution of these structures on leading and trailing wall result in migration of “hot-streaks” on walls, entrainment of coolant from the duct core and mixing of scalar field in the core of these vortices.

Figure 6.16 Unsteady dynamics and evolution of large scale coherent structures extracted from the flow fields using pressure Laplacian criterion.
6.6 Proper Orthogonal Decomposition

To analyze the low-dimensionality of this system, proper orthogonal decomposition (POD) is applied on two hundred snapshots from the flow field (For discussion on POD, see chapter 4). POD is a projection of turbulent fields on an optimal basis with some structure to these underlying flowfields. The optimality lies in the fact that for a given number of modes, the POD modes capture the most amount of “energy” of the turbulent fields. Mathematically, one solves an eigenvalue problem for the covariance or auto-correlation matrix. This matrix is constructed using method of snapshots (Sirovich, 1987). The eigenvectors would then be POD modes with eigenvalues representing the amount of “energy” captured by the respective mode. Clearly, first 75-80 modes capture almost 99% of the total turbulent energy (Figure 6.17).

![Figure 6.17 Energy distribution in the POD modes calculated from 200 snapshots.](image-url)
The identification of the first two POD modes with flow events shows that the event like impinging of flow on the trailing wall contains most “energy” (Figure 6.18, top row). In this event the streamwise component of velocity field is highest in front of the trailing wall rib (about quarter of the module pitch in front of the rib) and the velocity vectors are directed towards the trailing wall. Moreover, there is large recirculation pattern behind the rib on the leading wall. However, next most “energetic” event corresponds to impingement of flow behind the trailing wall rib (almost half of the module pitch behind the rib).

Figure 6.18 First two POD modes extracted from 200 snapshots for single module flow fields.

No attempt is made here to construct a low-dimensional ODE system emulating the dynamics of the Navier-Stokes PDE system. This is deferred for the future work.
6.7 Conclusion

Large eddy simulations are performed for a rotating square duct with normal rib turbulators to enhance the heat transfer. The Coriolis force as well as centrifugal buoyancy parameter has been included in this study. A direct approach is presented for the unsteady calculation of non-dimensional temperature field in periodic domains. The complex flow field shows dominant secondary flow vortices that enhance mixing of the thermal boundary layers on the duct walls with coolant fluid in the core. The temperature field is highly unsteady and may contain low frequency mode that allows the coolant to adhere to either front or the back wall of the duct. Simulations are performed in larger computational modules to understand coherent structure dynamics. Time-sequences of vorticity components and temperature fields are presented to understand flow physics and heat transfer processes in unsteady fashion. Coherent structures are extracted using a simple pressure Laplacian criterion. These shear layer (or roller) vortices evolve under the influence of Coriolis forces, centrifugal forces, variations of pressure gradients and other secondary flows. Proper orthogonal decomposition (POD) of 200 snapshots indicates a low dimensionality of this system. Almost 99% of turbulent energy can be captured by first 80 POD modes. First two most energetic modes are related to dynamical events of flow impingement due to Coriolis force. Further numerical studies are needed to resolve issues regarding discrepancies between computational and experimental results.
Chapter 7 Large Eddy Simulations of Trapped-Vortex Combustor

7.1 Introduction

High combustor inlet temperatures and airflow velocities, as well as lean combustion requirements, impose very stringent requirements for the development of stable, compact combustor systems with low emissions. Conflicting performance parameters at the high and low power operational conditions present a challenge to the historical manner in which combustors have been designed. Modern combustor aerodynamics is focused on enhancing fuel/air mixing for low emissions without degrading stability or exciting instabilities in the system. There are on-going efforts directed at re-designing the aerodynamics needed to improve flame stability and emissions. The Trapped Vortex Combustor (TVC) is a unique turbine engine combustor concept that offers reduced emissions and improved performance. The concept of a TV combustor was first presented by Hsu et al (1995). They showed that the recirculation zones could stabilize the flame and provide the sites for mixing, ignition and burning.

![Schematic and concept of the TVC](from Mancilla, 2001)

Figure 7.1 Schematic and concept of the TVC (from Mancilla, 2001).
Katta and Roquemore (1996, 1998) performed two-dimensional numerical studies for the experiments of Hsu et al (1995). Hosokawa et al (1996) performed RANS calculations using two-equation turbulence model to study the effect of flame holder shape on vortex shedding. In a recent study, Stone and Menon (2000) used 2-D LES to simulate the fuel-air mixing and combustion in a TVC. Benefits from the TVC concept can be realized for aircraft propulsion as well as marine, industrial, and electrical power generation applications. The TVC has proven to be a great advancement in combustor technology.

The integrated diffuser injector flameholder (IDIF) supplies the main air and fuel flows to the combustor. The current IDIF and fuel injection system, although performing well, can be enhanced significantly, yielding further reductions in NOx and better combustion efficiency over a wider fuel-to-air ratio range. To increase volumetric heat release, this novel method is being integrated with a practical fuel-injection design that makes it possible to shorten supersonic flame length in particular combustion geometry (Baurle and Gruber, 1998, Gruber et al, 1999 and Mathur et al, 1999). The trapped-vortex (TV) concept, previously explored for subsonic gas turbines, is being explored on ramjet/scramjet operation and combined with unique supersonic mixing enhancement features (Ben-Yakar and Hanson, 1998). Successful integration of advanced concepts will provide the scientific basis for developing compact combustors with a small length-to-diameter ratio. If successful, this integrated concept will be applicable not only for the scramshell combustor but to other ramjet/scramjet designs for hypersonic technology. In addition, achieving and dual-mode scramjet operation is essential for precision attack as it relates to time-critical targets, uninhabited aerial vehicles, and low-cost prevasion weapons. In the latest test series, ignition and lean blowout (LBO) tests were conducted.
on a TVC configuration. These tests characterized the ignition and LBO performance at pressures ranging from 3 psia, which simulates an altitude of approximately 40,000ft, up to 75 psia. Compared to previous TVC configurations, the results indicated the combustor stability was improved by approximately 15%. In addition, ignition was improved by nearly 40%. The F-414 Trapped Vortex Combustor is designed to reduce oxides of nitrogen emissions in the exhaust. However, back fitting all engines is cost prohibitive. It is more likely this technology will be incorporated into the phased new design F-414 replacement.

In a trapped vortex (TV) combustor, a properly sized cavity is used to trap a vortex, which provides the flame stability, reduces emissions and improves performance. A simple schematic presented in figure 7.1 shows the concept of the TVC to ensure good-mixing of air and fuel inside the cavity of the flame-holder (Mancilla, 2001). Large eddy simulations (LES) are performed for this numerical study. While 2D-LES results were reported earlier by Stone and Menon (2000), it should be noted that in two-dimensional simulations, the energy cascade via vortex stretching and folding or tilting mechanism is absent. This leads to large energy retaining eddies. However, in real situations, these vortices supply the energy to smaller scales, which dissipate the energy away due to viscosity. The resolution requirement up to dissipative scales can be very large and hence, beyond the computational resources. Moreover, the universal character of these dissipative scales deems their resolution non-economical. Large eddy simulations (LES) resolve the large eddies and model the energy drain in smaller eddies or subgrid scales. In this research effort, two simulations are performed that approximate the TVC geometry in a Cartesian framework first by two-dimensional planes and next by using Immersed
Boundary Method (IBM). These time-resolved three-dimensional simulations present the dynamics of the trapped vortex accurately compared to two-dimensional simulations that lacked the mechanism of vortex stretching and folding/tilting. Details of the time-averaged flow field and the turbulent stresses are presented at selected sections of the computational domain. The dynamics of TV influences the mixing inside the cavity strongly. The flow field is essentially 3-D in the presence of fuel and air injections. However, reaction and passive scalar mixing issues are not addressed in these simulations and hence, the fuel is treated with the same material properties as that of air.

7.2 Problem Description (Approximation of TVC with Cartesian Geometry)

A uniform Cartesian grid of 92×72×25 points is used for a domain of 45.5×35×12.5 mm³ (Figure 7.2). All the dimensions are selected to approximate the experimental setup of Hsu et al (1995) with Cartesian geometry. The periodicity of the geometry is exploited by putting half of the jet injection at the spanwise edges of the computational domain. At the inflow, fully developed turbulent profile is prescribed. At the walls, no slip boundary conditions are imposed. Uniform injections of air and fuel are applied at the respective holes. Periodic boundary conditions are applied in the spanwise (z) direction. At the outflow, a non-reflective convective scheme is applied to convect away the flow structures out of the computational domain without any spurious reflections. The wave speed is calculated to maintain the mass flux balance in the whole domain. Ratio of hole injection velocity to the inflow velocity is 0.5. Reynolds number based on the air injection velocity and air hole dimension is 5000 for these simulations.
7.3 Results

The numerical results obtained from the LES of the TV cavity are presented at various planes of computational domain to explain the dynamics and evolution of the trapped vortex. The time-averaged velocity field shows the fluid motion inside the cavity in the mean sense. Instantaneous snapshots of the velocity field at the centerplanes of air and jet injections indicate that this mean motion is induced by a strong vortex inside the cavity.
which is moving around in the cavity (Figure 7.3). The interaction at the mainflow and cavity interface is also averaged out in the mean field. Such interactions can be predicted only through a time-accurate simulation. Instantaneous fields show the flow separation due to the adverse pressure gradients at the top wall. Time averaged fields do not show this separation at the outflow. Flow unsteadiness at the vertical wall at $x = 0.0$ (forebody) is influenced by the location of the center of trapped-vortex.

![Instantaneous Velocity Vectors at $z = 0.0$ mm](image1)

![Time-averaged Velocity Vectors at $z = 0.0$ mm](image2)

![Instantaneous Velocity Vectors at $z = 6.25$ mm](image3)

![Time-averaged Velocity Vectors at $z = 6.25$ mm](image4)

Figure 7.3 Comparison of the instantaneous flow fields with the time averaged flow fields at the centerplanes of air and fuel injections.
Figure 7.4 Instantaneous snapshots of the velocity vectors and $\omega_z$ contours at the centerplane of air injections at time $t = 0.0$, $\tau$, $2\tau$ and $3\tau$ ($\tau = 3.3$ m.sec)

The time sequence of the velocity and spanwise component of vorticity is presented in figure 7.4. The negative $\omega_z$ region at the center of the trapped vortex can be clearly identified in all instants. The locus of this vortex-core region follows the mean flow trajectory (figure 7.3). This cyclic process of the motion of the center of trapped vortex is a slower process as compared to the mainflow-cavity interactions near the forebody i.e. shear layer vortices roll-up process. Therefore, it can be expected that the entrainment of the mainflow air into the cavity is achieved during the oscillation of trapped vortex. Towards the afterbody, there is a persistent separation region on the top-
wall. The flow accelerates over the afterbody due to this reduced cross-sectional area creating low-pressure region in the mainflow. The ejection of fluid mixture in the cavity can thus be achieved around this region. Pereira and Sousa (1994) showed that the flow unsteadiness can be captured reasonably well by LES for a turbulent flow over a cavity. They related this observed unsteadiness to the oscillatory pressure field inside the grooved channel forming the cavity. The magnitude of vorticity in the trapped vortex changes due to the stretching mechanism in the spanwise direction (Note that trapped-vortex core is intensified when it is near the injection jets and at later moment, it diffuses and loses intensity due to possible compressive modes in spanwise direction. In this discussion it is assumed that viscosity has relatively weaker effect on the dynamics as compared to the stretching/tilting mechanism. This latter process governs instantaneous kinetic energy transfer between the mean flow and the fluctuations. In simulations incorporating combustion chemistry and transport of scalar fields, it will be important to resolve this mechanism). Earlier simulations of Katta and Roquemore (1996) were two-dimensional and hence did not incorporate this important mode of vorticity transport.

The contours of Reynolds stress $u'v'$ and spanwise component of instantaneous vorticity $\omega_z$ are presented at the centerplanes of air and fuel injections in figure 7.5. The vorticity is mainly concentrated at the center of the vortex inside the cavity. The generation of positive vorticity at the top wall is observed. The structures generated by the interaction of the mainflow and the cavity are analogous to that of a mixing layer. These shear layer vortices convect downstream over the cavity resulting in the entrainment of the mainflow air into the cavity near the fore-body. Generation of vorticity along the edges of jet injections is also observed. The motion of these jets inside the
cavity assist the entrainment of mainflow into the cavity and results in mean trapped-vortex flow. Distribution of Reynolds stress $u'v'$ inside the cavity indicates that this turbulent shear stress component is resisting the mean fluid motion in the cavity. However, the fluid in the lower-left corner of the cavity (near the foot of fore-body) is mostly laminar. Clearly, in such situations a dynamic SGS model is required that can identify these regions and evaluate the model coefficient properly.

Figure 7.5 Contours of Reynolds stress $u'v'$ and $\omega_z$ at the centerplanes of air and fuel injections

The contours of normal turbulent stress components $u'u'$ and $v'v'$ are presented at the centerplanes of air and fuel injections in figure 7.6 ($w'w'$ is order of magnitude
smaller than these two components). High levels of normal stresses are present in the vicinity of the jet injections and the mainflow-cavity interaction region. In other parts of the cavity, the flow is unsteady but laminar. In the centerplane of air injection, the magnitude of normal stress $v'v'$ is large below the mean trapped vortex. In the centerplane of fuel injection, the normal stress $v'v'$ is large at the downstream side below the vortex. Large levels of normal turbulent stresses along the edges of mean trapped vortex indicate that fluid mixing is enhanced inside the cavity by exchanging the outer cavity fluid with interior of mean trapped-vortex.

![Reynolds Stress $u'u'$ at $z = 0.0$ mm](image1)

![Reynolds Stress $v'v'$ at $z = 0.0$ mm](image2)

![Reynolds Stress $u'u'$ at $z = 6.25$ mm](image3)

![Reynolds Stress $v'v'$ at $z = 6.25$ mm](image4)

Figure 7.6 Contours of normal turbulent stresses $u'u'$ and $v'v'$ at the centerplanes of air and fuel injections
The instantaneous velocity vectors, streamwise (x) vorticity component $\omega_x$ and Reynolds stress $v'w'$ are presented at a plane close to jet injections near the afterbody in figure 7.7. There is clearly a downwash of fluid into the cavity close to this plane. Complex flow structures develop between the air and fuel injections. This results in enhanced mixing of fuel and air before it gets entrained into the trapped vortex. There is indication of the development of hairpin vortices at the top wall. These vortices can be identified as the alternate patches of $\omega_x$ at the wall. Around this location, the top-wall boundary layer experiences adverse pressure gradient and the flow turns towards the afterbody away from top-wall. The turbulent stress is significant only close to the jet injections indicating that the flow is mostly laminar away from these injections. Damping of turbulence near the solid walls of afterbody can result in this observed behavior of turbulent stresses.

Figure 7.7 Instantaneous velocity vectors, contours of $\omega_x$ and Reynolds stress $v'w'$ at YZ plane (X = 42.75 mm)

The instantaneous velocity vectors, vertical (y) component of vorticity $\omega_y$ and Reynolds stress $u'w'$ are presented at a plane parallel to bottom wall through the lower air
injection in figure 7.8. Near the injection, there are secondary vortices that give rise to large levels of $\omega_y$. Distribution of turbulent shear stress at this plane opposes the motion of these vortices. Though there are lots of unsteady vortices close to forebody, the flow is essentially laminar there because of insignificant levels of Reynolds stress $u'w'$. 

Figure 7.8 Instantaneous velocity vectors, contours of $\omega_y$ and Reynolds stress $u'w'$ at XZ plane (Y = 4.75 mm)
7.4 Problem Description (Approximation of TVC using Immersed Boundary Method)

A uniform cartesian grid of $92 \times 57 \times 117$ points is used for a domain containing the upper half of the TV combustor. All the dimensions are selected to approximate the experimental setup of Chakka et al (1999) with Immersed Boundary Method on Cartesian grid (figure 7.9). Ratio of air injection velocity to the mainflow velocity is 2.2. Reynolds number based on the annular mainflow velocity and air hole dimension (D) is 3400 for these simulations. The radii of the forebody, the connecting tube, the afterbody and the outer shell are 24.5D, 3.7D, 23D and 27.5D respectively. The lengths of the forebody, the connecting tube and the afterbody are 12D, 30D and 12D respectively. The periodicity of the geometry is exploited by putting half of the jet injections around the bottom of the computational domain with the boundary conditions obtained by the rotational symmetry about the axial direction. Therefore, only half of the fuel injection holes at the meridional planes $0^\circ$ and $180^\circ$ are simulated. This corresponds to the largest azimuthal wavelength of size $\pi D_{TV}/2$ in the domain, where $D_{TV}$ is the diameter of the trapped vortex in the cavity. However, periodicity due to four injections implies the fundamental mode of wavelength $\pi D_{TV}/4$. Thus, stability of the trapped vortex can be analyzed with respect to very large scale disturbances in this computation. At the inflow, fully developed laminar profile along with fluctuations is prescribed. The fluctuations are assumed to be Gaussian and are calculated using Box-Muller algorithm. At the walls, no slip boundary conditions are imposed using immersed boundary method. Uniform injections of air and fuel are applied at the respective holes. Periodic boundary conditions are applied in the spanwise (z) direction. At the outflow, a non-reflective convective scheme is applied to convect away
the flow structures out of the computational domain without any spurious reflections. The wave speed is calculated to maintain the mass flux balance in the whole domain.

Figure 7.9 Details of the flame-holder and the afterbody fuel and air injection ports (Mancilla, 2001)

7.5 Results

The objective of this research effort is to investigate the dynamics and evolution of the trapped-vortex through a time and space-resolved three-dimensional large eddy simulation. No attempt is made to address the issue of combustion here. The numerical results obtained from the LES of the TV cavity are presented at various planes of computational domain to explain the dynamics and evolution of the trapped vortex. The meridional plane ($\theta = 90^\circ$) shows the flow on the centerplane through the computational domain. The meridional planes ($\theta = 0^\circ$ and $180^\circ$) shows the flow on the bottom plane of the computational domain. These two planes pass through the center of the fuel injections. The axial plane ($X/D = 40.8$) shows the flow around the fuel and air injections in front of the afterbody (This plane is 1.2D away from the afterbody injections inside the
cavity). First the unsteady evolution and dynamics of coherent structures is explained on these planes. Then, the mean flow behavior and the turbulent stresses are presented to explain their effects on the mixing process.

7.6 Unsteady Dynamics of Trapped Vortex

The instantaneous snapshots of velocity vectors and stream traces at the meridional planes ($\theta = 90^\circ$, $\theta = 0^\circ$ and $180^\circ$) are shown in figure 7.10. The stream traces at $\theta = 90^\circ$ show the presence of a large recirculation region above the fuel injection. A small recirculation region is also formed at the junction of the forebody and the fuel-air delivery pipe (referred to as connecting tube earlier). The instantaneous velocity vectors show the complex structure of the large recirculation region comprising of smaller recirculating regions within it. The differences of the flow field at these meridional planes clearly illustrate the strong three-dimensionality in the flow. The instantaneous stream traces show complex 3-D trajectories of the fluid parcels in the cavity at these planes. It will be shown later through the sequence of flow field snapshots that the TV is subjected to the strong axial, azimuthal as well as radial disturbances (Figure 7.11-7.13). The stability of trapped vortex to such complex variations can be very important regarding the mixing inside the cavity. For example, the unstable disturbances can breakup the toroidal (dough-nut shaped) trapped vortex into several large eddies inside the cavity in a quasi-periodic fashion, thus enhancing the homogeneity of the mixture inside the cavity. The corner vortices near the junction of the connecting tube and the forebody are formed due the stagnation of the flow. Streamtraces as well as the vector snapshots capture the corner vortices. The interaction near the forebody lip and the cavity is illustrated by the strong bending of the streamtraces around that region. The flow separates over the afterbody due
to strong adverse axial pressure gradient. The stream traces wrap around the TV and induce a nearly uniform core flow in the cavity. This induced flow is in the same direction as the fuel and air injections.

Figure 7.10 Instantaneous snapshots of the velocity vectors (left column) and the streamtraces (right column) at the meridional planes 90° (top row), 0° and 180° (bottom row).

At $\theta = 90^\circ$, the pressure is lower at the core of the TV which is about 10D away from the afterbody. The flow experiences strong adverse pressure gradient in front of the afterbody. There are small low pressure regions around the injections. The recirculating flow inside
the cavity is subjected to adverse gradients close to the forebody. The pressure gradients are large close to the core of TV, injections and separation region over the afterbody as expected (Figure 7.11). At $\theta = 0^\circ$ and $180^\circ$, the core of the TV is identified by the low pressure region at axial location close to the center of the cavity in figure 7.12. The distribution of pressure gradients is similar at these meridional planes except the location of the core of the TV. The adverse pressure gradients in front of the afterbody leads to the entrainment of the annular mainflow inside the cavity. At $X/D = 40.8$, the pressure is higher near the rim of afterbody and lower close to injections leading to radially inward pressure gradients (Figure 7.13).

Figure 7.11 Instantaneous snapshot of pressure field at $\theta = 90^\circ$
Figure 7.12 Instantaneous snapshot of pressure field at $\theta = 0^\circ$ and $180^\circ$

Figure 7.13 Instantaneous snapshot of pressure field at $X/D = 40.8$
Figure 7.14 Details of flow-field around trapped-vortex low-pressure iso-surface.

Figure 7.15 Large and small scale variations of the pressure fluctuations during the transient at \( \theta = 0^\circ \), \( R \sim 14 \, D \), \( X/D = 39 \)
An iso-surface of low-pressure level around the core of trapped vortex is visualized as a dough-nut shaped tube in the cavity (figure 7.14). The details of the flow field around this surface clearly shows the velocity vectors wrapping around the surface. The conventional criteria (positive surfaces of Laplacian of pressure) yields vortices around shear layers and jet injections. A typical signature of pressure fluctuation at meridional plane $\theta = 0^\circ$ and axial location in front of the fuel injection show both large and small scale variability (Figure 7.15). Clearly, the small scale fluctuations depict the turbulent diffusion at this location and the large scale variability can be associated with motion of the TV inside the cavity.

The unsteady dynamics of coherent structures is explained using the four instantaneous flow fields at time instants $t_0$, $t_0 + 5T$, $t_0 + 10T$ and $t_0 + 15T$, where $T$ is the non-dimensional time scale ($T = D/V$) and $t_0$ is an arbitrary instant when the flow is turbulent in the cavity. Since the focus of this study is motion of trapped vortex rather than the dynamics of injection jets, it seems reasonable to take time instants where the small scale dynamics as compared to trapped-vortex dynamics are de-correlated. The motion of trapped vortex inside cavity is a slower process as compared to the mainflow and cavity interaction near the forebody lip. The large scale and small scale variations of the pressure fluctuations are shown in figure 7.15 to illustrate this.

The instantaneous velocity vectors and spanwise component of vorticity $\omega_z$ are presented at the meridional plane $\theta = 90^\circ$ in figure 7.16. This component of vorticity is the same as the azimuthal component at that plane. The vorticity is mainly concentrated at the center of the vortices inside the cavity. The generation of positive vorticity at the top wall (outer cylinder) is observed. The structures generated by the interaction of the
annular mainflow and the cavity around the forebody lip are analogous to that of a mixing layer. Generation of vorticity along the edges of jet injections is also observed.

Figure 7.16 Instantaneous velocity vectors and the spanwise component of vorticity field $\omega_z$ at four time instants to show the evolution of coherent structures inside the cavity.

At $t_0$, the vortices in the mixing layer region are at the outer radial locations with respect to the afterbody and the separation region over the afterbody is small. At $t_0 + 5T$, the mixing layer vortices have been entrained inside the cavity, the smaller vortices around the jet edges have been convected towards the core of TV and the separation region over the afterbody is large. At $t_0 + 10T$, the vortical structures around the jets breakup due to instabilities and the adverse pressure gradients inside the cavity and the separation region over the afterbody is decreasing. At $t_0 + 15T$, there are a bunch of small vortices inside the core of TV, the mixing layer vortices are broken into smaller vortices and the
separation region over the afterbody is large. Clearly the separation bubble over the afterbody is more dynamic than the flow inside the core of TV. Thus, the mixing layer vortices are subjected to oscillatory adverse pressure gradient in front of the afterbody (in the annular region). The entrainment of the annular mainflow inside the cavity and the ejection of mixture inside the cavity around this region can be related to this dynamic process. Moreover, the mixing layer around the forebody lip enhances the mixing process in that region.

The instantaneous velocity vectors and the vertical (y) component of vorticity $\omega_y$ are presented at the meridional planes $\theta = 0^\circ$ and $180^\circ$ in figure 7.17. Note that at these selected meridional planes the vertical component of vorticity is the same as the azimuthal component if one performs the calculations in cylindrical coordinates. The distribution of vorticity is nearly anti-symmetrical at these planes as expected. At all time instants, the distribution of vorticity at the meridional planes $\theta = 0^\circ$ and $180^\circ$ differs significantly from the distribution at $\theta = 90^\circ$. All of these planes pass through the centerplane of fuel injections and should be identical for the axisymmetric case. Clearly, the problem must be solved in 3-D to understand the fluid mechanics of TV and mixing processes. At $t_0$, the fuel jets convect the vortices inside the core of TV and the mixing layer vortices are shed into the annular mainflow. The evolution at later time instants reveals a slow process of vortex breakup at these planes around the axial location close to the center of the cavity. The instantaneous center of TV at $\theta = 90^\circ$ is near the afterbody while the center of TV at $\theta = 0^\circ$ and $180^\circ$ is near the cavity center (see figure 7.14). Therefore, TV has a doughnut shape structure with the centerline as a function of axial,
radial and azimuthal parameters. These variations on TV generate high vorticity regions around TV envelope and therefore, mixing and dissipation will be higher at this surface.

Figure 7.17 Instantaneous velocity vectors and the vertical component of vorticity $\omega_y$ at four time instants.

To identify the vertical structures associated with the injection jets, the helicity (defined as scalar product between vorticity and velocity) is used. As expected, the positive and negative tubes of helicity surfaces can be identified inside the cavity and the helicity associated with the boundary layer on the outer shell of the cavity. Two cross-stream
planes are presented to show the velocity vectors around these injection jets. Radial ingestion of the annular fluid into the cavity is seen along these jets.

Figure 7.18 Details of flow-field near the injection jets from the afterbody

The instantaneous velocity vectors and the axial (x) component of vorticity \( \omega_x \) are presented at the axial plane X/D = 40.8 in figure 7.19. At \( t_0 \), the vorticity between the fuel and air injections is generated due to the flow entrainment near the connecting tube and the injection jets mix with each other. The later time instants depict the evolution of these unsteady and complex structures. The vorticity generated around the edges of the fuel and air injections is convected along the jets into the cavity. Some of the vortices from the annular mainflow are ingested radially inwards into the cavity at this plane. The flow
unsteadiness between the injections is high and is a faster process than the motion along the TV. Therefore, the mixing of fuel and air must occur very rapidly around this plane. The non-axisymmetric nature of the flow field is clearly represented in these snapshots.

Figure 7.19 Instantaneous velocity vectors and the axial component of vorticity field, $\omega_x$ at four time instants.

7.7 Time-Averaged Flow Fields

Attention is next turned to the time averaged flow behavior inside the cavity over a non-dimensional time duration of 110T. The streamtraces are presented at the meridional planes $\theta = 90^\circ$, $0^\circ$ and $180^\circ$ (figure 7.20 and 7.21). At $\theta = 90^\circ$, the large recirculation region is formed between the annular mainflow and the fuel injections. The unsteadiness of the mixing layer and the motion of TV inside the cavity has been averaged out (compare with Figure 7.10). A small recirculation near the forebody and spindle junction
is observed. Separation of flow over the afterbody due to adverse pressure gradients is also noted. At $\theta = 0^\circ$ and $180^\circ$, the streamtraces are asymmetric implying that a longer averaging period is needed to completely eliminate the slow variations. Moreover, the axisymmetry is also absent at these meridional planes. The axial view of the streamtraces in the cavity is shown in figure 7.22. TV is a doughnut shaped structure inside the cavity (figure 7.14, Note that an instantaneous iso-surface of low-pressure associated with the TV is closer to afterbody, while time-averaged streamlines indicate that the core of TV is closer to fore-body). The injection jets from afterbody ingest the annular fluid radially inwards and impart azimuthal momentum to fluid during this process. Between the injections, it is therefore expected for the fluid parcel to have a longer residence time and variable pitch of the spiral formed by streamtraces. The azimuthal motion of the streamtraces along the surface of the TV is absent in all the previous 2-D studies. The three-dimensional nature of these streamtraces is shown in figure 7.23. However, the core of TV is mostly irrotational. Positive surfaces of pressure Laplacian are significant close to jet-injections and shear layers. Low-level of pressure and lack of vorticity in the core of TV suggest that trapped-vortex is a flow pattern and not a coherent structure. The curvature of a vortex sheet produces an altered flux of the vorticity parallel to the sheet so that the speed of fluid parcel moving with the circulation is changed. The variation of thickness of vortex sheet induces a motion normal to itself. Thus, the stability of TV as an axisymmetric vortex ring subjected to perturbations can be analyzed (Saffman, 1992).
Figure 7.20 Streamtraces at $\theta = 90^\circ$ (tracers are released along radial positions at several axial locations).

Figure 7.21 Streamtraces at $\theta = 0^\circ$ and $180^\circ$ (tracers are released along radial positions at several axial locations).
Streamtraces from an axial plane inside the cavity

Figure 7.22 Axial view of the trapped vortex

Figure 7.23 Isometric view of the trapped vortex (all the solid surfaces are removed)
7.8 Turbulent Stresses

The role of turbulent stresses on the evolution of TV and mixing inside the cavity is presented at the various sections of the computational domain (figures 7.24-7.26). Since the calculation of turbulent stresses in LES will invariably invoke an SGS model, it should be noted that Reynolds stress referred to here are calculated from the time averages of filtered fields (Appendix I). At $\theta = 90^\circ$, the normal stress component $u'u'$ is large in the annular mainflow over the cavity center, in the separation region over the afterbody and away from the afterbody along the jet injections (Figure 7.24). The normal stresses $v'v'$ and $w'w'$ are large near the injections. The normal stress $v'v'$ is also large in the core region of the TV and along the face of the afterbody. The turbulent shear stress $u'v'$ inside the cavity is mostly generated by the production terms ($u'u'\partial V/\partial x$ and $v'v'\partial U/\partial y$) around the mixing layer region. Note that at this azimuthal plane, $x$ is along axial direction, $y$ is along radial direction and $z$ is out-of-plane azimuthal component. The anisotropy of normal stress components is evident and can be attributed to large scale structures associated with shear layer and injection jets. The high axial momentum fluctuations of the fluid parcels are correlated with radially inward fluctuations around this region. This implies the turbulent mixing and diffusion is enhanced between the cavity and annular mainflow by this stress component. The shear stress $u'v'$ is also large along the jet injections and is responsible for the jet spreading and mixing inside the cavity. However, the levels of turbulent stresses indicate that flow is mostly laminar close to the forebody and connecting tube junction. It can be expected that the modeling of this shear stress component using an eddy viscosity approximation may work here since the shear stress depends on the corresponding mean strain rate tensor component only.
Figure 7.24 Components of the Reynolds stress tensor at $\theta = 90^\circ$

At $\theta = 0^\circ$ and $180^\circ$, the normal stress $u'u'$ is large along the jet injections around the cavity center (Figure 7.25). At these planes, $x$ is directed along the axial direction, $z$ is the radial direction for $\theta = 0^\circ$ (negative for $\theta = 180^\circ$) and $y$ is the out-of-plane azimuthal component for $\theta = 0^\circ$ (in-to-plane component for $\theta = 180^\circ$). The normal stress $v'v'$ is negligible inside the cavity at these planes. The normal stress $w'w'$ is large behind the forebody lip. The distribution of turbulent stresses is primarily in-plane at this location. As noted earlier, this anisotropy of normal stresses on the injection center-planes is primarily due to large scale structures in these planes. It is clear that the entrainment process between annular shear layer and trapped-vortex is going to be strong here.
Figure 7.25 Components of the Reynolds stress tensor at $\theta = 0^\circ$ and $180^\circ$

The turbulent shear stress $u'w'$ inside the cavity is mostly generated by the production terms $(u'u'\cdot\partial W/\partial x$ and $w'w'\cdot\partial U/\partial z)$ around the jets in the cavity and the mixing layer region. This distribution indicates enhanced mixing of fluid parcels inside the core of TV. The distribution of $u'w'$ near the jet injections is dictates the radial spread of the jets along the axial direction in the cavity. Moreover, the dependence of the shear stress on
the corresponding mean strain rate tensor component implies that eddy viscosity approximation may work here. It may be noted that these azimuthal planes ($\theta = 0^\circ, 90^\circ$ and $180^\circ$) are geometrically equivalent and hence should yield statistically similar behavior for the time-averaged quantities. Turbulent stresses and time-averaged streamlines confirm this expectation.

At $X/D = 40.8$, the normal stress components $v'v'$ and $w'w'$ are highest at the injection locations (figure 7.26). The larger circles correspond to the fluctuations in the fuel jet and the smaller circles correspond to the fluctuations in the air jet injections (see figure 7.18). Large magnitude of $v'v'$ around $\theta = 90^\circ$ above the jet injections is also observed. This indicates the large radial fluctuations of the ingested annular mainflow. The normal stress $w'w'$ is greater around $\theta = 0^\circ$ and $180^\circ$. Again, at these locations the radial fluctuations are along the $z$-direction. Clearly, the radial fluctuations are high at different azimuthal or meridional planes. The normal stress $u'u'$ is mostly negligible due to suppression of fluctuations normal to the face of the afterbody. The only significant levels are above the afterbody in the annular region. The distribution of the shear stress $v'w'$ along the jets is primarily due to turbulent production ($v'v'\partial W/\partial y$ and $w'w'\partial V/\partial z$). The distribution in the annular region seems to attain the periodicity between the fuel injections (corresponding to the fundamental mode of wavelength $\pi D_{TV}/4$). The larger wavelength corresponding to the computational domain size is averaged out. Again, the modeling of shear stress by eddy viscosity assumption may work here. Non-linear eddy viscosity models can describe such anisotropic distribution in normal stresses (Speziale, 1991).
7.9 Concluding Remarks

Results show that the center of TV moves along mean flow trajectories. This leads to periodic oscillation of low-pressure region inside the cavity and change in vorticity magnitude of TV due to vortex stretching mechanism. Ingestion of annular mainflow in front of afterbody separation region is the main mechanism of flow entrainment inside TV cavity. Fluid mixture in the cavity is ejected radially outwards due to the pressure gradients near afterbody. Turbulent stresses enhance mixing between cavity fluid and annular mainflow and in the vicinity of air and fuel injections. The three-dimensional nature of time-averaged TV depicts a doughnut shaped structure inside the cavity.
• Unsteady dynamics of the coherent structures inside the cavity is a slower process as compared to the separation region over the afterbody and the mixing layer region behind the forebody lip.

• The instantaneous axial and radial locations of the TV are different at different meridional planes. However, time-averaged streamtrace patterns seem to converge towards a TV with simpler geometrical features.

• The ingestion of annular mainflow in front of the afterbody separation region is the main mechanism of the flow entrainment inside the cavity. The mixture in the cavity is ejected radially outwards due to the pressure gradients there.

• The three-dimensional nature of the time-averaged TV depicts a doughnut shaped structure inside the cavity. The streamtraces along this structure have azimuthal component of velocity and hence form a dense spiral around it. Azimuthal variability in the pitch of spiraling streamtraces is also noted.

• Turbulent stress $u'w'$ enhances the mixing inside the core of the TV. Turbulent stress $u'v'$ enhances mixing along the annular mixing layer behind the forebody and over the cavity. Turbulent stress $v'w'$ governs the radial spread of jets inside cavity as well as the mixing of cavity fluid with entrained annular mainflow at axial locations close to the injections.

• Dependence of the turbulent stresses on the in-plane normal stresses and the corresponding mean strain rate tensor component implies that the turbulence modeling using eddy viscosity assumption may work. However, the anisotropy of normal stresses would require a non-linear eddy viscosity model.
Chapter 8  Concluding Remarks and Future Directions

This research effort has addressed the following issues:

Development of LES methodology for complex geometries of industrial interests. Of particular interest, the external and internal cooling of modern gas turbine blades is studied using this methodology.

Simulation of high Reynolds number turbulent flows of industrial results is a daunting task. In principle, a true Large Eddy Simulation (LES) methodology can alleviate the issue of ever-increasing computation resource requirement with Reynolds number. However, there are several issues regarding the modeling and filtering in complex geometries that have not been addressed appropriately yet. Immersed Boundary Method (IBM) can potentially be applied to almost any complex and moving geometry. As a demonstration of capability of the method for complex moving geometry, a stator-rotor configuration is studied. Though, the flow conditions are of academic interests only, yet it shows the superiority of the methodology over any sliding mesh or re-gridding procedure to simulate this flow. There are some open issues regarding the resolution near the solid surfaces. In this research effort, a combination of these two powerful ideas is presented as the direction to take for the simulations of industrial turbulent flows with complex moving geometries.

Investigated parametric effects on the flow and heat transfer in simple jets-in-crossflow situations and film-cooling flow situation. Parameters explored included freestream turbulence intensity, freestream length scales, jet injection angle, hole geometry, blowing ratio and plenum effects.
1. The influence of hole aspect ratio \((AR = L/D)\) on the coherent structures in the near field was found to be significant. For small aspect ratios, the horseshoe vortex formed upstream of the jet is relatively weak. The jet lift-off is diminished for the low aspect ratio case by unsteady counter-rotating structures formed over the counter rotating vortex pair (CVP). For the large aspect ratio case, the horseshoe vortex is very strong and is observed to induce another unsteady co-rotating secondary CVP over the main kidney vortex.

2. The jet injection angle also affects the flow structures, their vorticity contents and the pressure gradients. Two injection angles were studied: a normal \((90^\circ)\) injection and an inclined \((30^\circ)\) injection. The stronger pressure gradients in the normal injection case lead to a larger recirculation region behind the jet. This is expected to adversely effect the film cooling effectiveness. For the normal injection, enhanced mixing of the jet-fluid and the mainstream is observed near the wall. This can severely decrease the film effectiveness.

3. The effect of freestream turbulence intensity levels \((Tu = 2\% \text{ and } 15\%)\) was studied to understand the influence of freestream fluctuations on the evolution of various coherent structures and the corresponding turbulent stresses. The major effect of the freestream turbulence intensity level was through the entrainment of the crossflow into the wake region. For the higher \(Tu\), higher turbulent stresses were noted in the near wall region. This is likely to lead to increased heat transfer.

4. Under realistic engine conditions, the freestream contains a spectrum of energy containing scales with the most significant portion of energy concentrated in the scales larger than the hole dimension \((D)\). Therefore, the effect of freestream length
scales at high turbulence intensity levels is also studied. This was done by prescribing a Von-Karman spectrum in the freestream with maximum energy at a wavenumber corresponding to 4D. The horseshoe vortex in front of the jet is observed to be energized in the large length scale case. The interactions at the jet-freestream interface are also enhanced for the large length scale case, and the corresponding levels of $u'u'$ and $v'v'$ are higher in the jet-crossflow interaction region.

5. An inclined circular jet in a crossflow is studied at different blowing ratios ($M=0.5$ and 1.0). Simulations correspond to an experimental study at UTRC. The jet delivery tube was simulated using IBM. Heat transfer calculations are also performed. The coherent structures extraction using positive pressure Laplacian criterion revealed hairpin vortices in the wake. This analysis presents a unified explanation of the projected vorticity field on different observation planes. The heat transfer calculations showed that jet-penetration and spreading are accurately predicted.

A solution methodology for flow and heat transfer predictions in periodically varying geometries representative of internal coolant channels with turbulators in gas turbine blades.

Unsteady heat transfer calculations in periodic geometries are fairly common. An approach based on scaling arguments about the self-similar profiles at periodic sections for non-dimensional scalar field and surface phenomenon is presented. A specific case for internal coolant channel of gas turbine is studied. Low-dimensionality of this system was established using Proper Orthogonal Decomposition (POD) technique. The unsteady dynamics of coherent structures in this complex flow field is extracted using simple
pressure Laplacian criterion. Analysis of thermal fields and coherent structures suggested that a larger computational domain must be used for these calculations.

Understanding unsteady mixing phenomenon in the trapped-vortex combustor by rendering the geometry using Immersed Boundary Method on a regular Cartesian grid.

Mixing inside a trapped vortex combustor (TVC) is a complex phenomenon. The geometry of TVC was simulated using IBM. The doughnut shaped structure of the vortex is observed. The motion of this vortex inside the cavity leads to exchange of the fluid inside the cavity with fluid in the annulus. Details of the time-averaged flow-fields and turbulent stresses are also presented.

1. A Cartesian approximation of the TVC revealed essential dynamics of the trapped-vortex. However, there are some differences in the entrainment processes.

2. An immersed boundary implementation of true dumbbell shaped cylindrical flame-holder is performed. It clearly demonstrated the three-dimensional dynamics of trapped-vortex and the potential of IBM in simulating such complex flows.

Work will be continued to study the following issues:

1. The current code will be extended to a multi-block version. The development of a direct solver for pressure Poisson equation is the most challenging step in this direction. The approach being used is very similar to the influence matrix approach (Appendix V, Kleiser and Schumann, 1980 and Raspo et al, 1994).

2. To develop a simple turbulence model for economic calculations at high Reynolds numbers, we need to perform a highly accurate simulation with and without any models. The LES budgets will be tested against the DNS calculations to validate the predictive capability of LES at moderate Reynolds numbers. The LES runs at higher
Reynolds number will then provide the missing information about the turbulent stress budgets (Laurence, 2001). The calculation of second moment budgets and two-equation turbulence model budgets is currently under implementation. The relationship between LES stresses and the actual Reynolds stresses will be used rigorously to derive the budget equations that can be evaluated as run-time statistics (Appendix I, Zang, 1993 and Deshpande and Milton, 1998). These simulations will be performed using the multi-block version of the code for the simple Cartesian geometry.

3. The immersed boundary method will be used for more complex geometry situations. The resolution near complex geometries will be addressed using a zonal embedded (Nested) grid refinement approach (Appendix VI). Implementation of filters will be done independent of grid resolution to allow a) numerical studies with varying resolution with fixed filter width and b) numerical studies with varying filter width on fixed resolution. First set of studies can explore the issues of numerical errors due to discretization and grid resolution, while the latter set of studies can yield information on errors with SGS modeling (Geurts and Leonard, 2001).

4. The fundamental issues related to filtering techniques and SGS modeling will be investigated further. The issue of commutation errors and filtering in complex domain has been studied (Tyagi and Acharya, APS/DFD, 1999). The SGS model needs to be tested for more complex flows.

5. A parallel version of the code will be developed to cut-down on computational time using Message Passing Interface (MPI).
References


Adrian, R.J. (1999), On the physical space equation for large-eddy simulation of inhomogeneous turbulence, TAM Report No. 915.


Bredberg, J. (1997), Turbine blade internal cooling, Chalmers University, Goteborg.


Chen, H.-C., Wei, G. and Han, J.-C. (1999), Computation of discrete-hole film cooling by a near-wall second-moment turbulence closure, IMECE, Nashville,TN.


Durbin, P. A. and Pettersson Reif B. A. (2001), Statistical theory and modeling for turbulent flows, John Wiley & Sons Ltd.


262


Han, J.C. and Ekkad, S.V. (1998), Recent development in turbine blade film cooling, ISTP-11.


Kellogg, S. (2000), Immersed boundary methods with applications to flow control, M.S. Thesis, Rice University, TX.


Lilly, D.K., (1966), On the application of the eddy viscosity concept in the inertial subrange of turbulence, NCAR Manuscript 123, NCAR, Boulder, CO.


McComb, W. D. (1990), The physics of fluid turbulence, Oxford University Press.


Ou, S., Mehendale, A. B., and Han, J. C. (1990), Influence of high mainstream turbulence on leading edge film cooling heat transfer: effect of film hole row location, ASME90-WA-HT-5, ASME Winter Annual meeting, Dallas.


Rajaratnam, N. (1976), Turbulent jets, Elsevier scientific publishing company.


Rivir, R.B., Gogineni, S.P., Goss, L.P. and Pestian, D.J. (1997), The unsteady structure of simulated turbine film cooling flows from PIV, AGARD propulsion and energetics panel, paper no. 47.


Appendix I: Relationship Between Reynolds Stress Tensor and SGS Stress Tensor

The relationship between Reynolds stress tensor and SGS tensor can be established using Germano’s identity. Let us denote the ensemble average or long time average by \(<\cdot>\) and the corresponding fluctuation field with a prime. Again, the overbar represents the filtered field and the corresponding subfilter (or subgrid as it is commonly referred to) is represented by double prime. Therefore, we have a relation for the representation of any instantaneous field amongst the ensemble in terms of either RANS field or the LES field.

\[ u = \langle u \rangle + u' = \bar{u} + \tilde{u} \]

Define the Reynolds stress tensor \( R_{ij} \) as follows

\[ R_{ij} = \langle u_i u_j \rangle - \langle u_i \rangle \langle u_j \rangle \]
\[ = \langle u_i u_j \rangle + u'_i \langle u_j \rangle + u'_j \langle u_i \rangle + u'_i u'_j - \langle u_i \rangle \langle u_j \rangle \]
\[ = \langle u_i u'_j \rangle \quad \text{\( \because \langle u_i \rangle \rangle = \langle u_i \rangle \langle u_i \} = 0 \) \}

Define the SGS tensor \( T_{ij} \) as follows

\[ T_{ij} = \frac{u_i u_j - u_i u_j}{u_i u_j + u'_i u_j + u'_j u_i + u'_i u'_j - \left( \frac{u_i u_j + u'_i u_j + u'_j u_i + u'_i u'_j}{u_i u_j} \right)} \]

Rewriting in terms of resolvable part and the modeled eddy viscosity part,

\[ T_{ij} = L_{ij} + M_{ij} \]
\[ L_{ij} = \frac{u_i u_j - u_i u_j}{u_i u_j + u'_i u_j + u'_j u_i + u'_i u'_j} \]

Now, apply ensemble averaging or long time averaging operation on SGS tensor

\[ \langle T_{ij} \rangle = \langle L_{ij} \rangle + \langle M_{ij} \rangle \]
Again, apply filtering operation on Reynolds stress tensor and using the fact that filtering and ensemble averaging operations commute, we get

\[ R_{ij} = \langle u_i u_j \rangle - \langle u_i \rangle \langle u_j \rangle - \langle u_i^* \rangle \langle u_j \rangle - \langle u_i \rangle \langle u_j^* \rangle \]

For an ensemble averaged filtered field, the unclosed stress terms would be

\[ R_{ij} = \langle u_i u_j \rangle - \langle u_i^* \rangle \langle u_j \rangle \]
\[ R_{ij} - \bar{R}_{ij} = \langle u_i \rangle \langle u_j \rangle - \langle u_i \rangle \langle u_j \rangle \]
\[ R_{ij} - \langle T_{ij} \rangle = \langle u_i u_j \rangle - \langle u_i \rangle \langle u_j \rangle \]
\[ \therefore \bar{R}_{ij} - \langle T_{ij} \rangle = \langle u_i u_j \rangle - \langle u_i \rangle \langle u_j \rangle \]

Therefore, the above stated relation acts as a bridge between any RANS field and an ensemble of LES fields. Making an approximation that the subfilter or subgrid fields are stochastic in nature and hence the ensemble average or long time average of these fields can be neglected (Zang, 1993). However, the correlations of such fields do not vanish during such averaging operation.

\[ \bar{R}_{ij} \approx \bar{T}_{ij} + \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j \]
\[ \bar{R}_{ij} - \langle T_{ij} \rangle \approx \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j \]
\[ \bar{R}_{ij} - \langle M_{ij} \rangle \approx \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j + \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j \]

Note that the right hand side expression can be evaluated as runtime averages from LES for any given flow and since it involves only the filtered fields, it is not expected to be sensitive to the SGS model for the LES. For anisotropic and inhomogeneous flow situation, the calculable part captures most of the turbulence production. Hence, in this paper, we will use the difference of filtered Reynolds stress and ensemble averaged SGS
tensor as a surrogate to Reynolds stress tensor to describe the dynamics and evolution of flow structures.
Appendix II: The Eddy Viscosity Concept and Smagorinsky’s Model

Using the arguments for energy spectrum in the equilibrium range, one can relate, energy decay, \( \varepsilon \), to the magnitude of wave number, \( k \), by

\[
\varepsilon = \left[ v + \alpha \int_0^\infty k^{-\frac{3}{2}} \left[ E(k') \right]^\frac{1}{2} \, dk' \right] \left[ \int_0^k 2k''^2 \, E(k'') \, dk'' \right] \]

\[
H(k) = \int_0^k k''^2 \, E(k'') \, dk''
\]

\[
\therefore \varepsilon = 2 \left[ v + \alpha \int_0^k k^{-\frac{3}{2}} \left[ \frac{dH(k)}{dk'} \right]^{\frac{1}{2}} \, dk' \right] H(k)
\]

On dividing by \( H(k) \) and differentiating, we get

\[
\frac{dH(k)}{dk} = \frac{4\alpha^2}{\varepsilon^2} \frac{[H(k)]^4}{k^5}
\]

\[
\therefore [H(k)]^{-3} = 3\alpha^2 \frac{\varepsilon}{\varepsilon^2} k^{-4} + \left( \frac{\varepsilon}{2\nu} \right)^{-3}
\]

Since \( H(k) \to \varepsilon/2\nu \) as \( k \to \infty \), hence

\[
E(k) = \frac{1}{k^2} \frac{dH}{dk} = \left( \frac{8\varepsilon}{9\alpha} \right)^{\frac{3}{2}} k^{-\frac{5}{3}} \left[ 1 + \frac{8\nu^3}{3\alpha^2 \varepsilon} k^{-4} \right]^{\frac{4}{5}}
\]

Therefore, the eddy viscosity will be \( \nu_T \sim \varepsilon^{1/3} k^{4/3} \) in the inertial subrange and \( \nu_T \sim 0 \) in viscous subrange. The expression of \( \nu_T \) can be related to \( \varepsilon \) now. Smagorinsky employed the traditional analogy between the turbulence effects and molecular properties. The subgrid stress tensor is supposed to be expressible in terms of explicit scales by relationship

\[
\tau_{ij} = 2\nu_s \bar{S}_{ij} = \frac{(u_{i,j} + u_{j,i})}{2}
\]

\[
\bar{S}_{ij} = \frac{(u_{i,j} + u_{j,i})}{2}
\]

286
using a velocity scale $S\Delta$, where $S$ is the magnitude of the resolved strain rate tensor and $\Delta$ is the filter width.
Appendix III: The Dynamic One-Equation Subgrid Scale Model

The transport equation of subgrid kinetic energy can be written as follows using the generalized moments concept (Germano, (1992), Ghosal et al (1995) and Davidson, (1997))

\[
\frac{\partial k_{SGS}}{\partial t} + \left( u_j k_{SGS} \right)_j = -\Phi_f \left( u_j, u_j \right) u_{i,j,i,j} - \left\{ \frac{1}{2} \Phi_f \left( u_j, u_i, u_i \right) + \Phi_f \left( u_j, p \right) \right\}_j + \nu k_{SGS,jj} - \nu \Phi_f \left( u_{i,j}, u_{i,j} \right)
\]

\[
k_{SGS} = \frac{1}{2} (u_i u_j - \bar{u}_i \bar{u}_j)
\]

\[
\Phi_f \left( u_i, u_j \right) \equiv u_i u_j - \bar{u}_i \bar{u}_j = \tau_{ij}
\]

\[
\frac{1}{2} \Phi_f \left( u_i, u_i, u_j \right) = \frac{1}{2} u_i u_j u_i - \bar{u}_i \tau_{ij} - \frac{1}{2} u_j \tau_{ii} - \frac{1}{2} \bar{u}_i \bar{u}_j
\]

\[
\Phi_f \left( u_i, p \right) = \frac{1}{\rho} \left( u_i p - \bar{u}_i \bar{p} \right)
\]

\[
\Phi_f \left( u_{i,j}, u_{i,j} \right) = u_{i,j} u_{i,j} - \bar{u}_{i,j} \bar{u}_{i,j}
\]

here, the subscript \( f \) represents the grid filter level.

The transport equation for test filter-level kinetic energy, \( K \) can be obtained in the similar fashion. Now, using the following expression for the SGS tensor, we get

\[
\tau_{ij} = -2C \Delta k_{SGS}^{0.5} \bar{S}_{ij}
\]

\[
\hat{P}_{k_{SGS}} = \frac{1}{\Delta} C_k k_{SGS}^{3/2} = \frac{k_{SGS}}{K} \left( P_k - \frac{1}{\Delta} C_k K^{3/2} \right)
\]

\[
C_{n+1} = \left( P_k - \hat{P}_{k_{SGS}} - \frac{1}{\Delta} C_n k_{SGS}^{3/2} \right) \frac{\hat{\Delta}}{K^{1/2} k_{SGS}}
\]

In deriving this expression, the transport of SGS kinetic energy is assumed proportional to that of \( K \) with the proportionality constant equal to the ratio of test-filtered SGS energy and \( K \). The dynamic coefficient is not taken out of the test-filter, instead the localized form of Ghosal et al (1995) is generally used (Piomelli and Liu, 1995).
However, there must be bounds on the value of $C$ for numerical stability. This model seems to incorporate more flow physics than standard eddy-viscosity model, but it is computationally more expensive too.
Appendix IV: The Differential Stress Model

Since the second moment closure approximations are less severe than single scale formulation of eddy viscosity, the details of the differential stress models are presented here. The full model is explained with assumptions and approximations generally made for the closure. The aim is not to use this model but to expose the reader to the kind of approximations that are made and need to be evaluated rigorously.

\[ L = \text{grad}(v), D = \frac{1}{2}(L + L^T), S = 2\nu D \]

\[ B = v \otimes v - \bar{v} \otimes \bar{v}, \text{sym}(F) = \frac{1}{2}(F + F^T) \]

\[ \partial_t(B) + \text{div}(B \otimes \bar{v}) = -(\bar{L}B^T + BL^T) + M + \Pi + E + U \]

\[ M = \text{div}(\bar{v} \otimes \bar{v} - \bar{v} \otimes v \otimes v + B \otimes \bar{v}) + 2\text{sym}\left[ \text{div}(\bar{v} \otimes (S - \text{grad}(pI)) - \bar{v} \otimes (\bar{S} - \text{grad}(\bar{p}I)) + \bar{v} \otimes B \right] \]

\[ \Pi = 2(pD - \bar{p}D), E = \left( \bar{S} \bar{L} - SL + L' \bar{S} - \bar{L}S \right) - U = \left( v \otimes f + f \otimes v - \bar{v} \otimes \bar{f} - \bar{f} \otimes \bar{v} \right) \]

The closure of various tensors are approximated as follows

\[ k = \frac{1}{2} \text{tr}(B), \nu_k = c_k \Delta^{1/2}, \epsilon = c_\epsilon \frac{k^{3/2}}{\Delta} \]

\[ c_k = 0.07, c_M = 4.15, c_\epsilon = 1.05 \]

\[ M \approx \text{div}(\nu_k \text{grad}(B)) \]

\[ \Pi = \Pi^1 + \Pi^2, \Pi^1 \approx -c_M \frac{k^{1/2}}{\Delta} B_D, \Pi^2 \approx \frac{2}{5} k \bar{D}_D \]

\[ E \approx -\frac{2}{3} \epsilon I \]

\[ \partial_t(B) + \text{div}(B \otimes \bar{v}) = -(\bar{L}B^T + BL^T) + \text{div}(\nu_k \text{grad}(B)) - c_M \frac{k^{1/2}}{\Delta} B_D + \frac{2}{5} k \bar{D}_D = \frac{2}{3} \epsilon I \]

It can be shown that effective dissipation is tensorial and linearly dependent on the SGS anisotropy tensor, thereby providing improved treatment of flow and grid anisotropies.
Appendix V: Direct Solver for Possion Equation in Complex Geometries

Martin (1973) generalized the classical capacity matrix technique to incorporate large class of arbitrary and unusual internal boundary conditions. Schumann (1980) obtained direct solution to fluid-structure interaction problems using influence matrix technique (IMT). Orszag (1980) extended the spectral methods to complex geometries using mapping and patching. Kopriva (1986) presented a multidomain spectral method for hyperbolic systems. The division of computational domain allowed local refinement and flexibility in distribution of mesh points. Gunzburger and Nicolaides (1986) presented an extension of substructuring algorithm that carries out the block Gauss elimination procedure without the need for interchanges even when a pivot matrix is singular. Macaraeg and Strett (1986) enforced a global flux balance that preserves high-order continuity of the solution at the interfaces. Shen (1995) developed a fast Poisson solver based on the Legendre-Galerkin approximations with the complexity $O(N^2 \log N)$ in two-dimensional rectangular domain. McKenney et al. (1995) presented a fast Poisson solver based on potential theory. Greengard and Yee (1996) presented a direct, adaptive solver for Poisson equation based on domain decomposition approach using local spectral approximation, as well as potential theory and the fast multipole method. In 2-D, the algorithm requires $O(NK)$ work, where $N$ is the number of discretization points and $K$ is the desired order of accuracy. Tufo and Fischer (1997) presented a fast direct solver for parallel solution based on the (quasi-) sparse factorization of the inverse of $A$ for linear systems of the form $Ax = B$. Averbuch et al. (1998) presented a direct method for the solution of Poisson equation based on a pseudospectral Fourier approximation and a polynomial subtraction technique. The solution can be evaluated at $N^2$ interior points.
requiring $O(N^2 \log N)$ operations. Braverman et al. (1998) presented a 3-D version of the method of Averbuch et al. Golub et al. (1998) developed a fast direct Poisson solver for the projection method of the incompressible Navier-Stokes equation with finite difference schemes on the half-staggered grid. Gustafsson and Hemmingsson (1998) presented a fast domain decomposition high order Poisson solver for parallel computations. The method used deferred correction by solving a sequence of systems with narrower stencil and domain decomposition and it remains direct in the sense that for any given order of accuracy, the number of arithmetic operations is fixed. Plagne and Berthou (2000) presented a tensorial basis collocation method for Poisson’s equation and showed that maximum number of iterations for a given $N$ (number of grid points in each direction) leading to a competitive iterative scheme is $12N/128$.

Kim and Moin (1985) presented a fractional step method in conjunction with approximate factorization technique. They derived boundary conditions for the intermediate velocity field to achieve higher temporal accuracy. Tuckerman (1989) generalized the influence-matrix method of enforcing incompressibility and showed it to be an application of the classic Sherman-Morrison-Woodbury formula of numerical linear algebra. Perot (1993) analyzed the fractional step method as a block LU decomposition. Armfield and Street (1999) showed that pressure correction method could achieve second-order accuracy while projection method is only first-order in time, and it requires considerably less CPU time as compared to iterative methods. Strikwerda and Lee (1999) analyzed the accuracy of the fractional step method and showed that the pressure in any projection method can be at best first-order accurate. Brown et al. () showed the coupling of approximation of pressure gradient in momentum equation, the
formula used for global pressure update during time step and the boundary conditions can be combined to yield a fully second-order accurate projection method.

The extension of the direct solver for pressure Poisson equation in multi-domain geometry will be done in the future work using the influence matrix approach. However, the solution will be solved twice in each sub-domain to avoid huge storage requirements and construction of final solution involving enormous matrix-vector multiplications. The formulation of the direct-solver in multi-patched domains will be as follows. For simplicity, consider a rectangular domain be divided into two sub-domains. Let interior of the sub-domains be $\Omega_i (i = 1,2)$, boundary of the sub-domains be $\partial \Omega_i (i = 1,2)$ and the interface between the sub-domains be $\Gamma_{12} = \partial \Omega_1 \cap \partial \Omega_2$. Also, let the outward normal to the domain boundary be $\eta$. Consider the following sub problems with respective boundary conditions

**Non-homogeneous problem 1:**

$$\nabla^2 P_1^* = \nabla \cdot u / \Delta t \text{ in } \Omega_1$$

$$\frac{\partial P_1^*}{\partial \eta} = 0 \text{ on } \partial \Omega_1 \setminus \Gamma_{12} \text{ and } P_1^* = 0 \text{ on } \Gamma_{12}$$

**Non-homogeneous problem 2:**

$$\nabla^2 P_2^* = \nabla \cdot u / \Delta t \text{ in } \Omega_2$$

$$\frac{\partial P_2^*}{\partial \eta} = 0 \text{ on } \partial \Omega_2 \setminus \Gamma_{12} \text{ and } P_2^* = 0 \text{ on } \Gamma_{12}$$

**Homogeneous problem 1:**

$$\nabla^2 P_1^\prime = 0 \text{ in } \Omega_1$$

$$P_1^\prime = 0 \text{ on } \partial \Omega_1 \setminus \Gamma_{12} \text{ and } \frac{\partial P_1^\prime}{\partial \eta} = \left( \frac{\partial P_1^*}{\partial \eta} - \frac{\partial P_2^*}{\partial \eta} \right) \text{ on } \Gamma_{12}$$

**Homogeneous problem 2:**

$$\nabla^2 P_2^\prime = 0 \text{ in } \Omega_2$$
\[ P_2' = 0 \text{ on } \partial\Omega_2 \backslash \Gamma_{12} \text{ and } \partial P_2'/\partial \eta = -(\partial P_1^* / \partial \eta - \partial P_2^* / \partial \eta) \text{ on } \Gamma_{12} \]

Then, it can be easily shown that the final solution is \( P = (P_1^* + P_1') \cup (P_2^* + P_2') \)

where \( P \) satisfies the following equation \( \nabla^2 P = \nabla \cdot u / \Delta t \) in \( \Omega_1 \cup \Omega_2 \) and the boundary conditions \( \partial P / \partial \eta = 0 \) on \( (\partial \Omega_1 \cup \partial \Omega_2) \backslash \Gamma_{12} \).
Appendix VI: Zonal/Local Refinement of Cartesian Mesh

The issue of very fine grids in the vicinity of walls is a big challenge for the large eddy simulations to be a practical tool for engineering flows. The smaller scales of motion need finer resolution in all three directions. The non-uniform grid stretching is incapable to resolve the smaller scales and hence the energy contained in them, though they might be sufficient for the better predictions of mean flow field. Moreover, the non-uniform grids introduce commutation errors in the LES formulation. There have been several suggestions to avoid or minimize commutation errors in such complex situations. The accurate simulations of turbulence need high-order accurate discretization of the governing equations. The zonal refinement approach can address the issue of resolution and accurate modeling of SGS stresses as well as commutation errors satisfactorily while retaining high-order of accuracy of discretization schemes.

There is a large body of literature on zonal embedded grids. The relevant issues that need to be addressed while using such an approach for LES should be conservation, accuracy, stability, consistency and the implied modifications to the underlying solution algorithm. Although the use of zonal refinement is fairly recent in the area of LES, the idea of local mesh refinement on Cartesian meshes is pretty old. Rai (1986) presented a zonal approach, wherein the given region is subdivided into zones and the grid for each zone is generated independently. This procedure introduces zonal boundaries at the interfaces of various zones that are accounted for in an accurate, conservative and stable fashion. Berger (1987) presented a procedure to derive conservative difference approximations at the grid interfaces for two-dimensional grids that overlap in an arbitrary configuration. The interface formulas were computed for grids that are refined
in space and/or time, and for continuous grids where a switch in scheme causes the discontinuity. Kallinderis (1992) proposed interface treatment schemes that have certain accuracy and conservation properties. An interface treatment that avoids interface grid stretching error and that is non-conservative was found to be more accurate over a conservative treatment for viscous flows that do not include moving shocks. Schmidt (1995) presented the construction of multigrid method on locally refined grids and the constraints on the prolongation and restriction operators. Coirier and Powell (1995) performed critical assessment of the accuracy of the Cartesian mesh approaches as compared to the body fitted meshes. The global order of accuracy was second-order while the local error was between first- and second-order accurate for their simulations. Edwards (1996) presented a flux continuous locally conservative approximation that removes the interface error and has a symmetric positive definite matrix for general discrete anisotropic coefficients. Minion (1996) presented a projection method for locally refined grids. The adjointness relation between gradient and divergence operators for refined grid MAC projection and a refined grid approximate projection was developed. Kravchenko et al. (1996, 1999) developed a B-spline based numerical method on zonal embedded grids for the computations of turbulent flows using LES. Sullivan et al. (1996) presented a grid nesting methodology in the framework of large eddy simulations (LES). They used a conservation rule for averaging fine grid resolved and SGS turbulent fluxes and kinetic energy to the coarse grid that is equivalent to Germano’s identity used to develop dynamic SGS models. Boersma et al. (1997) performed nested-grid calculations with LES. They used different grid-communication strategies to show that a local increase of resolution can be achieved through grid-nesting procedures. Colella et al.
(1998) observed that the dependent variables in a finite difference method are represented as arrays defined on subsets of an index space and the transformations on arrays can be expressed as combinations of pointwise operations on the arrays, and of sums over nearby points of arrays i.e. stencil operations. They proposed to use the stencil locations and the locations where the stencil operations are applied and computed using a set calculus on the index space. This provides a convenient and consistent infrastructure for a general-purpose algorithm. Day et al. (1998) presented a graph-based strategy for representing the computational domain for embedded boundary discretizations. This representation allowed recursive generation of coarse-grid geometry representations suitable for multigrid and adaptive mesh refinement calculations. Shariff and Moser (1998) developed a two-dimensional mesh embedding procedure with B-spline as basis functions. Bennett and Smooke (1998, 1999) showed that local rectangular refinement multiple-scale discretization produced a smaller overall error than the single scale discretizations commonly used on unstructured grids, and the layering technique also reduces errors while increasing grid robustness. These results were comparable in accuracy to those obtained on larger equivalent tensor product grids. Roma et al. (1999) used adaptive version of immersed boundary method on locally refined meshes to simulate complex geometries on the Cartesian grids. Cook (1999) discussed the issue of commutation errors in LES using the adaptive mesh refinement. Multiple uniform grids in a nested hierarchy using a constant-width filter for each grid was used as a means to mostly avoid commutation errors where increased grid resolution is required to capture key flow features. Moore (1999) derived finite difference approximations for grids with irregular nodes to ensure consistency and accuracy. Washio and Oosterlee (2000)
proposed an interpolation technique on interior boundaries of the composite grid based on conservative discretization and presented a rigorous error analysis on the locally refined grids. Teigland and Eliassen (2001) described a patched-based local refinement that is essentially block-unstructured. Gullbrand et al. (2001) proposed a high-order wall treatment procedure using Lagrangian interpolations and extrapolations in locally refined Cartesian grids for LES.

In the future development, a nested grid approach will be used. The interpolation between grid interfaces will be achieved using cubic splines. Note that such an interpolation will be conserving both mass and momentum fluxes. Moreover, the nesting of finer meshes in the coarse cells provides a rational basis for SGS modeling without any commutation errors on the coarse mesh. Also, the mesh refinement will be performed in the multiple of odd-integers. This approach has advantages in the staggered mesh arrangement for the coarse node will always coincide with the node of central refined mesh and hence only injection from fine mesh to coarse mesh is needed, avoiding interpolation of finer mesh data. Furthermore, the interfaces are also positioned consistently in such a refinement strategy. Thus, the little loss in the flexibility to perform arbitrary refinement is well compensated by these added advantages.
Mayank Tyagi was born in Ghaziabad, Uttar Pradesh, India, in December 1974. He completed his high school studies at the top 0.1% in the nation and joined the undergraduate studies program in mechanical engineering at the Indian Institute of Technology, Kanpur, India, in 1991. He finished his undergraduate studies in 1995. In the same year, he joined Louisiana State University, Baton Rouge, to pursue his doctoral studies. He expects to receive the degree of Doctor of Philosophy at the May 2003 commencement.