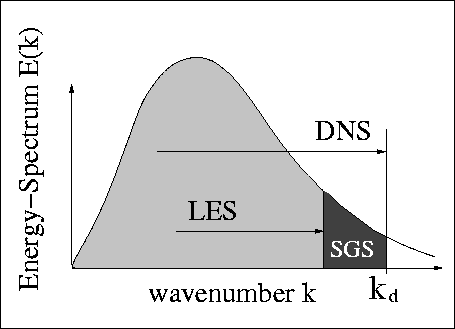
**Chapter 1**

**Introduction**

**1.1 Motivation**

Turbulent flows in the presence of one more solid surface, referred to as wall bounded turbulent flows, are prevalent in various practical situations. Wall bounded flows are important due to their extended use in engineering applications, such as the external flow around cars, ships, and buildings, the internal flows in engine pistons, pipes and channels. Many of these flows can also include the presence of a reacting region such as the combustor of jet engine or the flame spread along walls during accidental fires. The presence of reaction zones increases the complexity of the flow as well as the difficulty to understand and predict the flows using models. A further understanding of wall bounded turbulent flows where strong temperature gradients plays a major role is needed. The focus of this proposal is to present the numerical frame work necessary to study variable density wall bounded turbulent flows.

Turbulent flows in physical systems and in engineering applications are prevalent, such as the processing of fluids with pumps or flows around vehicles – e.g., airplanes, automobiles, and ships. Heat transfer in turbulent flows also plays an important role in many industrial engineering systems. An understanding of the behavior and characteristics of turbulent flows with heat transfer is of crucial interest for development of theoretical modeling and predictive computational tools. Often conducting wind tunnel experiments to study these flows can be expensive and numerical simulations may provide an attractive low cost option. The solution of turbulent flows is possible through computational simulations that vary by their level of resolution and accuracy. The most reliable computational strategy is Direct Numerical Simulation (DNS) in which all length-scales are fully resolved; due to its unparalleled accuracy and high computational cost it is only feasible for low Reynolds number flows. At the other end of the spectrum, Reynolds Averaged Navier Stokes Equation (RANS) resolve only the mean motion and rely on modeling the entire reynolds stress tensor, it has been most widely used in the engineering community for simulation practical high Reynolds number or complex flows. However, if steady state versions of RANS equations are used it will tend to suppress large scale instabilities that are fundamental to turbulent flow transport. Large Eddy Simulations (LES) represent a compromise between DNS and RANS since it does not intend to numerically resolve all turbulent length scales, but only a fraction of the larger energy-containing scales within the inertial sub range. Modeling in then applied to represent the smaller unresolved (SGS) scales, which contain only a small fraction of the turbulent kinetic energy.

****

The solution of turbulent flow problems is theoretically possible through the direct numerical simulation (DNS) of the Navier Stokes equation (with appropriate boundary conditions). This method constitutes the conceptually simplest approach to the problem of turbulence. Practically, however, the cost of DNS confines this approach to simple application in terms of Reynolds number and geometry complexities. In fact, in DNS all scales of motion must be resolved, from the integral to the Kolmogorov scales. The grid size must be on the order of the largest scale. This results in a number of grid points in each direction that is proportional to the ratio between the largest and the smallest scale (L/eta). Defining a Reynolds number based on the integral scale L, Re\_l, the number of grid points will be proportional to Re\_l(3/4) in each direction, so a total number of points proportional to Re\_l(9/4) is required by DNS. Moreover, the time step of the calculation dt is limited by CFL condition, that is dt < dx/U where U is the local velocity determined by the integral time and length scale, we have that the total number of steps in time is T/dt again proportional to (L/eta). In this way the total cost of a DNS calculation will be on the order of Re\_l^3. This mean that for high Re numbers, DNS takes an unrealistic time to be completed: assuming that computer power will increase by a factor of 5 every five years, Spalart [1] estimates that DNS will not be applicable for the study of the flow over an airliner or a car until 2080. In this context, large-eddy simulations (LES) presents a feasible alternative to DNS calculations since it strictly models the smallest scales of the flow. The large eddies containing the bulk of the energy, typically anisotropic and dependent on boundary conditions, are simulated directly. The fact the small, dissipative eddies are modeled helps reduce the cost of LES considerably compared to DNS.

Often flows of interest are found near solid surfaces such as the flow around a wing or around an engine piston. This configuration, wall bounded flows, introduces additional complexities that arise due to the interaction between the flow and the solid surface. In wall bounded flows, the presence of a strong shear layer is important and is responsible for introducing large velocity gradients producing strong skin-friction effects (i.e., drag). The presence of turbulence in wall-bounded flows presents an additional challenge since they are characterized by much less universal properties than free shear flows and are thus more challenging to compute. Within the viscous sublayer, the characteristic length is set by the friction velocity and the viscosity and outside from this the distance from the wall provides the appropriate scaling. As the Reynolds number increases and the thickness of the viscous sublayer decreases, the number of grid points required to resolve the near-wall structures increases. Thus, when LES have to be applied to wall-bounded flows at moderate to high Reynolds number it is still very demanding because of the inner layer resolution. In fact, as found in Chapman [2], considering a flat plate boundary layer, the outer part has energy-carry structures on the order of the boundary layer thickness, delta. Assuming that the grid spacing is fixed in the streamwise and spanwise direction and on the order of 0.1delta, Chapman estimate a total number of grid points proportional to Re\_l^0.4. In the inner part of the boundary layer, on the other hand, the number of points must be based on the inner layer scales; in fact, the dimension of quasi-streamwise vortices are constant in wall units (i.e., normalized by kinematic viscosity, wall stress and fluid density). In this way, the total cost of the calculation in the inner layer is estimated to be proportional to Re\_l^1.8 which makes this technique only suitable for moderate Reynolds number applications.