Lecture 36

TURBULENT FLOWS: FEATURES AND SIMULATION STRATEGIES

36.1 FEATURES OF TURBULENT FLOWS

Turbulent flows are encountered in wide range of natural and engineering systems. In fact, flow through all types of fluid machines (whether hydraulic or gas turbines, pumps, compressors or blowers) is invariably turbulent. Main features of turbulent flows are:

- Turbulent flows are highly unsteady, three-dimensional and diffusive.
- These contain great deal of vorticity and entail considerable dissipation of kinetic energy of the flow.
- Turbulent flows fluctuate on a broad range of time and length scales which makes direct numerical simulation of turbulent flows very difficult.

36.2 NUMERICAL SIMULATION OF TURBULENT FLOWS

Due to ubiquitous nature and importance in engineering applications, turbulent flows have received considerable attention from theoreticians, experimentalist and numerical analysts over past century. Difficult nature of the problem still fascinates scientists and engineers alike. In numerical simulation, three approaches have emerged for prediction of turbulent flows:

- Reynolds Averaged Navier-Stokes (RANS) simulation
- Large eddy simulation (LES) and
- Direct numerical simulation (DNS)

36.3 REYNOLDS AVERAGED NAVIER-STOKES SIMULATION

RANS simulations are based on time averaging of Navier-Stokes equations. New terms appear in governing equations which are *modelled* to ensure closure. The modelling reduces the requirements of very fine grids. RANS simulations are *work horse* of industrial CFD for design analysis. Their accuracy is dependent on the underlying turbulence models.

36.4 LARGE EDDY SIMULATION

In **LES**, one resolves the largest scales of motion of flow while modelling the small scales of motion. Main features of LES are:

- It represents a compromise between RANS and DNS in terms of accuracy and computational requirements.
- Mesh refinement in LES is more than RANS but smaller that DNS.
- It is feasible research tool for accurate simulation of large Reynolds number (high velocity) flows.
- It is slowly augmenting RANS simulation in industrial design analysis.

36.5 DIRECT NUMERICAL SIMULATION OF TURBULENT FLOWS

In **direct numerical simulation (DNS),** Navier-stokes equations are solved with extremely fine mesh and small time steps to resolve motions at all scales. DNS represents the simplest approach from conceptual point of view and any of the approaches suitable for accurate time integration discussed in earlier lectures can be used in DNS. The main features of DNS are

- Grid size is determined by the finest scale of turbulence – the viscosity determined length scale called the Kolmogorov scale η.
- Number of grid points are proportional to $\mathrm{Re}_l^{3/4}$, where Re_l is based on the magnitude of velocity fluctuations and integral length scale.
- Cost of DNS scales as Re_l^3
- Explicit time integration schemes are usually preferred.

Flow field computed from DNS is equivalent to a single snap-shot. Because of astronomical computing requirements, DNS is primarily used as a research tool to

- Understand the mechanism of turbulence production, energy transfer and dissipation.
- Understand the effect of compressibility on turbulence.
- Control and reduce drag on solid surfaces.
- Calibrate experimental techniques for near wall flows.
- Fine tune RANS and LES models.

Numerical discretization techniques for Navier-Stokes equations discussed earlier are equally applicable to turbulent flow simulations (taken as it is, those approaches represent DNS). However, we need to solve additional algebraic/differential equations in case of RANS and LES which require modelling of additional unknowns resulting from averaging (RANS) or filtering (LES). Consequently, we provide a brief outline RANS and LES models in the next three lectures.

FURTHER READING

Ferziger, J. H. And Perić, M. (2003). *Computational Methods for Fluid Dynamics*. Springer.

Lesieur, M. (2008). *Turbulence.* 4th Ed., Springer, Berlin.

Pope, S. B. (2000). *Turbulent Flows*. Cambridge University Press, Cambridge.

Versteeg, H. K. and Malalasekera, W. M. G. (2007). *Introduction to Computational Fluid Dynamics: The Finite Volume Method*. Second Edition (Indian Reprint) Pearson Education.