Lecture 2

NUMERICAL SIMULATION PROCESS

2.1 NUMERICAL SIMULATION PROCESS

Numerical simulation of a physical problem involves approximation of the problem geometry, choice of appropriate mathematical model and numerical solution techniques, computer implementation of the numerical algorithm and analysis of the data generated by the simulation. Thus, this process involves the following steps:

- Model the geometry of the problem domain.
- Choose appropriate mathematical model of the physical problem.
- Choose a suitable discretization method.
- Generate a grid based on the problem geometry and the discretization method.
- Use a suitable solution technique to solve the system of discrete equations.
- Set suitable convergence criteria for iterative solution methods.
- Prepare the numerical solution for further analysis

Let us have a bit more detailed look at the preceding steps. Each of these steps clearly reminds us of the approximations involved at each step of numerical simulation: from approximations/idealizations used in geometry modelling to solution process and post-processing.

2.2GEOMETRY MODELLING

The numerical simulation requires a computer representation of the problem domain. For most of the engineering problems, it may not be possible or even desirable to include all the geometric details of the system in its geometric model. The analyst has to make a careful choice regarding the level of intricate details to be chosen. For example, in the numerical simulation of flow field around an automobile, finer details of the front air-intake grills would be avoided. Incorporation of these finer features would make the grid generation process very difficult, but would hardly contribute to the accuracy of the velocity and pressure fields.

2.3 MATHEMATICAL MODELLING

An appropriate mathematical model for the problem has to be selected keeping in view the objective of the simulation, and physics of the flow problem. For example, one can opt for incompressible Navier-Stokes equations for low speed aerodynamics (Mach number < 0.3, e.g. flow over a car or train). Similarly, for high speed compressible flow over a whole aircraft, one may choose inviscid model (Euler's equation). The choice of the model also depends on the available computing resources and level of accuracy desired.

2.4 DISCRETIZATION METHOD

For computer simulation, the continuum mathematical model must be converted into a discrete system of algebraic equation using a suitable discretization procedure. There are many discretization approaches. The most popular are the finite difference method (FDM), the finite element method (FEM) and the finite volume method (FVM). Choice of the discretization method depends on the problem geometry, preference of the analyst and pre-dominant trend in a particular application area. For instance, FEM is very popular for stress analysis applications, whereas FDM has traditionally been more popular for simulation of turbulent flows. Similarly, commercial CFD codes have shown a distinct preference for the finite volume method.

2.5 GRID GENERATION

The problem domain is discretized into a mesh/grid appropriate to the chosen discretization method. The type of the grid also depends on the geometry of the problem domain. Structured grid is required for the finite difference method, whereas FEM and FVM can work with either structured or unstructured grids. In case of unstructured grids, care must be taken to ensure proper grading and quality of the mesh.

2.6 NUMERICAL SOLUTION

The discretization method applied to the mathematical model of the problem leads to a system of discrete equations: (a) a system of ordinary differential equations in time for unsteady problems, and (b) a system of algebraic equations for steady state model. For unsteady problems, time integration methods for initial value problems are employed, some of which transform the differential system to a system of algebraic equations at each time step. Iterative methods are usually employed to solve the system of algebraic equations, choice of methods being dependent on the type of the grid and size of the system.

The convergence criterion for the iterative solvers depends on the accuracy as well as efficacy requirements. The tightness of the specified error tolerance would also depend on the precision chosen for numerical computations.

2.7 POST-PROCESSING

Numerical simulation provides values of field variables at discrete set of computational nodes. For analysis of the problem, the analyst would like to know the variation of different variables in space-time. Further, for design analysis, secondary variables such a stresses and fluxes must be computed. Most of the commercial CFD codes provide their own post-processor which compute the secondary variables and provide variety of plots (contour as well as line diagrams) based on the nodal data obtained from simulation. These computations involve use of further approximations for interpolation of nodal data required in integration and differentiation to obtain secondary variables or spatial distributions.

2.8 VALIDATION

Numerical solution of a physical problem must be validated with available experimental data to ensure that it gives a reasonably accurate description of the physical reality. In general, numerical solution is sought for a problem for which no experimental results are available. For example, it is not feasible to perform experiments on a full scale prototype of an airplane or high-speed train. In such situations, validation of the simulation process is carried out with the scale model for which experimental data are available. Thereafter, the simulation process can be extended for numerical solution of the full-scale problem.

REFERENCES/SUGGESTED READING

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

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