Lecture 30

TYPES OF GRIDS AND GRID GENERATION PROCESS

30.1 INTRODUCTION

A mesh or grid is a set of points distributed over the problem domain for a numerical solution of a set of partial differential equations (PDEs). In CFD analysis, type of grid/mesh would depend on the discretization technique (FDM/FVM/FEM), geometry of the problem domain and underlying physics. 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.

The grid/mesh strongly affects the accuracy of the numerical solution, and hence, significant effort is usually invested in generation of a suitable grid. In industrial CFD analyses which involve numerical solution of problems in complex geometries, mesh generation process may consume nearly 50% of overall simulation time.

A lot of research effort has been invested in automatic (numerical) grid generation in past few decades, and it is still an active research area. In this introductory course on CFD, it is not possible to take a detailed look at different mesh generation techniques. Hence, this and next lecture would primarily provide a descriptive outline of automatic grid generation. Interested reader can refer to suggested books/references for complete mathematical details of different algorithms and their computer implementation.

30.2 TYPES OF GRIDS

Majority of grids can be broadly put into two categories:

1. Structured grids
2. Un-structured grids

Structured Grids

In structured grids, grid points follow a fixed structure. In particular,

- Grid points are located at the intersections of the grid lines. Hence, each grid points can represent the origin of a local coordinate system.
- Interior grid points have a fixed number of neighboring points.
- Grid points can be represented by a set of indices corresponding to the intersecting grid lines.

Structured grids consists of elements which can be mapped to a rectangle (in 2-D) or parallelepiped (in 3-D). Thus, all grid lines are oriented regularly in either two or three directions so that co-ordinate transformation of curvilinear lines results in a rectangle in 2D and parallelepiped in 3D.

Structured grids can be Cartesian or body-fitted grids. The former are used in simple rectangular geometries, whereas the latter are preferred for domains with curved boundaries. Further, the body-fitted grids can be

(a) orthogonal (in which curvilinear grid lines are perpendicular at the grid-points), or

(b) non-orthogonal (in which curvilinear grid lines intersect obliquely at the grid-points).

Block-structured Grids

For complex geometries, generation of a body-fitted structured grid may be very difficult. In this case, it is usually advisable to decompose the problem geometry in a set of blocks (sub-domains), and generate a structured grid in each in each block separately. The grid structure in each block can be different.

Block-structured grids allow different mesh densities in different regions (blocks), and thus fine grids can be easily put in regions likely to show steeper gradients of problem variables. The blocks can even be overlapping. Such block-structured grids (i.e. grids with overlapping subdomains) are called *composite* or *chimera* grids.

Un-structured Grids

Unlike structured grids, sides of a cell/element have no relation to the co-ordinate directions in unstructured grids. The un-structured grid consists of triangles or quadrilateral in 2D and tetrahedra, wedge, hexahedra or polyhedra in 3D.

Unstructured grids are much easier to generate in complex domains, and hence, majority of commercial CFD solvers have adopted unstructured grid based solvers. In finite volume applications, quadrilateral (hexahedral) elements are preferred for better accuracy in interpolation and integrations.

30.2 GRID GENERATION PROCESS

The process of grid-generation for complex geometries normally involves following steps:

- Decompose the problem domain into a set of sub-domains (blocks)
- In each block, generate the requisite grid. Typical sequence of operations would be
  - Generate edge-grid (i.e. divide the edges of a surface in desired number of one dimensional elements).
  - Using the edge grids, generate the grid on the block-surfaces.
  - Use surface grids as input to generate volume mesh.
- Check mesh quality, and modify the mesh as required.

REFERENCES/FURTHER READING

Chung, T. J. (2010). *Computational Fluid Dynamics*. 2nd Ed., Cambridge University Press.

Liseikin, V. D. (2006). *A Computational Differential Geometry Approach to Grid Generation*. Springer.

Taniguchi, T. (2006). *Automatic Mesh Generation for 3D FEM, Robust Delaunay Triangulation*. Morikita Publishing.

Thompson, J. F., Soni, B., Weatherill, N. P. (1999). *Handbook of Grid Generation*. CRC Press.

Thompson, J.F., Warsi, Z.U.A., Mastin, C. W. (1985). *Numerical Grid Generation, Foundations and Applications*. North Holland.