ENE 206 – Fluid Mechanics

WEEK 13

- ## Introduction

➢ CFD is "solving fluid flow problems numerically" . Computational Fluid Dynamics is a tool that allows us to solve flow problems that do not have known analytical solutions and cannot be solved in any other way. CFD is becoming increasingly important in research and design, in consultancy and industry, and in all domains of engineering, continues to grow. This is caused by:

i) the increasing awareness of the power of this approach;

ii) the increase in computational resources; and

iii) the availability of increasingly user-friendly and powerful CFD software.

At the same time, these reasons are also the reasons why CFD is often criticized.

- ## Methods of prediction

Prediction of fluid flow process can be achieved by three different methods. These are the experimental investigation, theoretical calculation and numerical computation.

➢ **Experimental investigation:** The most reliable information about a fluid flow process can be obtained by actual measurement. If a full-scale prototype is investigated experimentally, it is possible to predict the performance of identical copies of this prototype under similar conditions. However, performing experiments with full-scale equipment are usually very expensive and sometimes even impossible. For this reason, experiments are usually performed on small scale models and the resulting information is extrapolated to the full scale prototype by using the rules of similarity. However, as was discussed in Chapter 7, it is very difficult to establish complete similarity between the prototype and model. Besides, the small scale models do not always simulate the features of the full scale equipment and there are serious difficulties in measurements since the measuring instruments are not error free.

➢ **Theoretical prediction:** In order to perform a theoretical calculation, a fluid flow phenomenon must be mathematically modeled by a set of differential equation. If the methods of classical mathematics are used for the solution of these differential equations, only a few fluid flow problems of practical engineering importance cen be solved in closed form. Besides, these closed form solutions involve infinite series, transcendental equations for eigenvalues, etc., which can be evaluated numerically.

➢ **Numerical calculation:** The advantages that a numerical computation offers over a corresponding experimental investigation are as follows:

i) Low cost

ii) Speed

iii) Complete and detailed information

iv) Ability to simulate realistic conditions

v) Ability to simulate ideal conditions.

- ## Mathematical description of fluid flow phenomenon

The science of fluid dynamics is based on three fundamental physical principles which are the

i) conservation of mass,

(ii) conservation of momentum or the Newton's second law of motion and

(iii) conservation of energy

The fundamental governing equations of fluid dynamics are based on these three physical principles and they are the

(i) continuity equation

(ii) momentum equation and

(iii) energy equation

When the fundamental physical principles are applied to an infinitesimal fluid element, the governing partial differential equations of fluid dynamics are directly obtained. When the fluid element is moving with the flow, nonconservative forms of the governing equations are obtained. However, conservative form of the governing equations is obtained if the fluid element is stationary.

- ## Mathematical behavior of partial differential equations on CFD

➢ The mathematical behavior of partial differential equations can be quite different from one circumstance to another. For this reason, the same governing flow equations can yield completely different solutions in different regions of the flow field.

To understand the mathematical behavior of partial differential equations, consider the following second order quasi-linear partial differential equation:

$$a\frac{\partial^2\varphi}{\partial x^2}+b\frac{\partial^2\varphi}{\partial x\partial y}+c\frac{\partial^2\varphi}{\partial y^2}+d\frac{\partial\varphi}{\partial x}+e\frac{\partial\varphi}{\partial y}+f\varphi+g=0 \qquad (10.1)$$

where *A*, *B*, *C*, *D*, *E*, *F* and *G* are general functions of the dependent variable φ and the independent variables *x* and *y*.

Characterization depends on the roots of the higher order (here second order) terms:

i) (b^2 - 4*ac*) > 0 then the equation is called hyperbolic.

ii) (b^2 - 4*ac*) = 0 then the equation is called parabolic.

iii) (b^2 - 4*ac*) < 0 then the equation is called elliptic.

If *a*, *b*, and *c* themselves depend on *x* and *y*, the equations may be of different type, depending on the location in *x*-*y* space. In that case the equations are of mixed type.

- The origin of the terms “elliptic,” “parabolic,” or “hyperbolic” used to label these equations is simply a direct analogy with the case for conic sections. The general equation for a conic section from analytic geometry is:

  where if.

  i) (b^2 - 4ac) > 0 the conic is a hyperbola.

  ii) (b^2 - 4ac) = 0 the conic is a parabola.

  iii) (b^2 - 4ac) < 0 the conic is an ellipse.

Numerical discretization techniques

- Analytical solutions of partial differential equations are closed form expressions which give the variation of dependent variables continuously in the solution domain.

- However, numerical solutions of partial differential equations give values only at discrete points of the solution domain, which are referred to as the **grid points.** If the grid points are regularly arranged, the grid is referred to as the structured grid. However, recent research in CFD uses unstructured grids where the grid points are placed in the flow field in an irregular fashion.

- There are three discretization techniques, which are widely used in computational fluid dynamics. These are the

  (i) finite difference method,

  (ii) finite element method and

  (iii) finite volume method..

References

1. Aksel, M.H., 2016, “Notes on Fluids Mechanics”, Vol. 1, METU Publications
2. Anderson, J.D.A., 1995, “Computational Fluid Dynamics”, McGraw Hill Book Company, New York
3. Hirsch C., 1989, “Numerical Computation of Internal and External Flows”, Vol. 1 & 2, John Wiley and Sons, Inc., New York
4. Hoffman, K.A., 1989, “Computational Fluid Dynamics for Engineers”, The university of Texas at Austin
5. Versteeg, H.K. and Malalasekera, W., 1989, “Introduction to Computational Fluid Dynamics: The Finite Volume Method”, Addison Wesley Longman Limited, Great Britain.