ME 431A/538A/538B 23 October 2019

General Comments on Numerical Methods in Fluid Mechanics

In addressing a new problem in fluid mechanics, one has the choice of many approaches, depending on the types and quality of the information needed, the accuracy required, the costs, the time available, and other related factors. If rough estimates are needed, then 'back-of-the-envelop' estimates can sometimes be made, with such approaches as using Bernoulli's equation, control-volume analysis, potential flow analysis, and other related methods. Other problems, for example determining the dynamic stability of a certain flow, might require somewhat sophisticated mathematical analysis coupled with numerical solutions of the resulting equations. When more detailed information on a particular flow is required, then either laboratory experiments or computer (numerical) simulations are usually employed.

For most applications in fluid mechanics, the equations needed to be solved in a numerical solution, the Navier-Stokes equations plus possibly thermo-chemical equations, are highly nonlinear (except for very low Reynolds number flows), and the geometries can be very complex. For such problems analytical solutions are out of the question, and numerical methods are required. As numerical methods have improved, and computers have become faster, cheaper, and larger (in both processor memory and hard-drive or solid-state storage), numerical solution has become more feasible for complex problems, and are becoming quite common. Some examples of such problems are the following.

- Aircraft design, including both the aerodynamics and the ventilation systems, among other things.
- Auto and trucking industry, including drag forces (even the drag forces on the wheels), and engine design to decrease pollutants and increase performance.
- Power generation, e.g., flow in an industrial burner, or flow in a fuel cell.
- Biomedical, e.g., cardiovascular flows, the pulmonary system.
- Weather prediction.

In fact computational fluid mechanics is now used in most applications involving fluid mechanics.

There are a number of <u>advantages</u> to using computational fluid dynamics.

- Compared to laboratory experiments, there is not the need to build a new model for each test case. Generally the approach is then less expensive, and the design or research process is much faster.
- Compared to laboratory experiments, some issues can be studied experimentally that cannot be studied in the laboratory, e.g.,
  - fire on a spacecraft (at zero gravity so that there is no buoyancy);
  - hypersonic flows.
- The results from a numerical simulation are very comprehensive, e.g.,
  - results are obtained at each point of interest in space and time, which is generally not possible to do experimentally;

  - results are obtained for all the variables of interest, e.g., pressure and specie concentrations; some of these are impossible to measure experimentally.

However, there are also some disadvantages in using numerical simulations. These include the following.

- Models for the physics and chemistry used in the simulations may not be very accurate; this is especially true for turbulent flows, and can tremendously limit the quality of the results of a simulation.

- Numerical errors – there might not be enough resolution in the simulations, or the method of integration might not be sufficiently accurate, or both.

Because both numerical simulations and laboratory experiments have significant strengths and weaknesses, it is important to understand these strengths and weaknesses, and to consider which is most appropriate for a given problem. And sometimes, when considering a complex problem such as the design of airplanes, automobiles, trucks, etc., both methods can be employed synergistically, taking advantage of the strengths of each.