Errors and Uncertainty in CFD

Sources of Errors

1) Iterative Convergence Errors:

Consider the discretized equation

$$A_P\phi_P = A_E\phi_E + A_W\phi_W + A_N\phi_N + A_S\phi_S$$

By dividing through by A_P we formulate an equation which we can iterate over. Upon "iterative convergence" the left side of the above equation would exactly equal the right side after finishing a sweep through the domain. However, before convergence, there is a cell-by-cell difference between the two sides which is known as a residual. If one sums the magnitude of the residuals after each complete iteration then a measure of iterative convergence is obtained. The goal is to drive that number toward zero.

Commercial solvers often "normalize" the total residuals by the total residual obtained after the first few iterations. The guideline (or rule of thumb) is that one should iterate until at least a 3 order of magnitude reduction in residuals is obtained; that is, continue until the residual for each equation drops to 10^-3 or below. However, this is not always possible to achieve, particularly for complex geometries with complex physics.

2) Discretization or Truncation Errors

We discussed these errors in a previous lecture. In essence, the numerical integration for the finite volume method (say a midpoint or Simpson's rule) has a truncation error associated with it. This carries over into the CFD calculation. Similarly for any finite-difference approximation or interpolation to cell faces that are implemented in the code.

Recall that if one uses a 1st order method and decreases the grid size by a factor of 2, the truncation error should decrease by a factor of 2; for a 2nd order method the decrease would be a factor of 4, a third order method a factor of 8, and so on.

When developing a code one must balance truncation error with complexity. This becomes an issue particularly near boundaries.

3) Roundoff errors

Due to finite precision arithmetic on the computer. For CFD, not as large a problem as the previous two. Recommend running codes in double precision.

The procedure used by most CFD users to assess error is to run a code for a given problem using 3 levels of mesh refinement and compare the results. The difficulty arises when considering the appropriate means of comparing results. Although not particularly rigorous, comparing line plots, contour plots, etc is still perhaps the most common approach.

V&V or Verification and Validation

These terms are well established in the CFD community.

The term Verification involves quantifying errors such as iterative convergence, discretization (truncation), and roundoff.

Validation involves an assessment of the physical model uncertainty (for instance a combustion or turbulence model) and input uncertainty (for instance, boundary conditions, fluid properties, etc.).

As a guideline for best practices, I refer you to the American Society of Mechanical Engineers Journal of Fluids Engineering link which provides a discussion of numerical accuracy for CFD results:

https://www.asme.org/wwwasmeorg/media/ResourceFiles/Shop/Journals/JFENumAccuracy.pdf