Computational Fluid Dynamics

Jennifer Sinclair Curtis – Purdue University

The Computational Fluid Dynamics (CFD) Task Force is working to develop methods and tools for incorporating CFD in undergraduate education. CFD is the numerical solution of the differential momentum balance and continuity equation and is sometimes accompanied by the solution of turbulent transport equations, the differential energy balance, or species continuity equations.

In the academic setting, CFD has historically been used as a research tool. However, with advancements in the capabilities, user-friendliness, and speed of simulations, CFD can now also be used as an educational tool. Understanding of concepts in transport and reaction engineering classes can be enhanced through visualization. Students can see developing flow, temperature, or concentration profiles. The CFD graphics bring "life" to the lecture or homework assignment. CFD can be used as a computational laboratory where students can explore the effects of changes, for example, fluid properties, geometry of system, and operating conditions. Students can also investigate 2-D and 3-D problems that can not be solved analytically. In 1-D situations, they can compare their simulation results to analytical solutions obtained in class, results from empirical correlations, or their own laboratory data.

When using CFD students need to think through the modeling of the process and answer questions such as:

Is 2-D acceptable?
What boundary conditions should be applied?
What inputs and parameters need to be specified?
Where are large gradients anticipated and where should a finer grid be used?

As a supplement to CFD, students also can be introduced to numerical solution techniques for non-linear coupled differential equations or more advanced turbulence models.

Through the introduction of CFD in ChE education, students become familiar with a tool for design, scale-up and optimization of flow processes. More companies are using CFD for a range of single-phase and multiphase applications. Without learning CFD in their undergraduate education, it is very easy for students to complete their transport course sequence with the impression that if the flow problem can not be solved analytically or there are no empirical correlations, there are no methods to address the problem.

Recent efforts with the Fluent program FLOWLAB have focused on developing a series of CFD problems that can be applied seamlessly into an undergraduate fluid mechanics course. These problems were distributed at the recent 2002 ASEE Summer School. If you would like a copy of these problems, please contact CFD Task Force leader, Jennifer Sinclair Curtis at jlds@ecn.purdue.edu.