New ANSYS CFD Curriculum Materials Released

In fluid mechanics and transport phenomena courses, students are often expected to visualize fluid flows and analyze pressure/temperature drop and transport of scalars within the flow domain — a task easier said than done. Technology that helps students visualize flow phenomena and relate them to physics and equations describing the same can be highly beneficial in the classroom. Some professors may take a step further and introduce computational fluid dynamics (CFD) in their fluid mechanics or transport phenomena courses. Professors might need tools to teach CFD at the undergraduate or graduate level. To meet these concerns, ANSYS recently introduced a variety of teaching materials .

Teaching materials include tutorials with embedded documentation for students that explain the models used in the modeling, teaching guides for professors that explain the physics, learning outcomes, as well as validation with analytical results. This makes it easy for them to introduce the concepts. General guidelines explaining steps to solve the tutorials, and installation videos are also available.

In the past, professors used Flowlab software from ANSYS to introduce CFD and flow visualization in fluid mechanics classes. As Flowlab was retired in December 2013, ANSYS customized the ANSYS® Workbench™ interface to provide the same functionality and user environment. The company also recreated some of the Flowlab tutorials. Professors familiar with the Flowlab interface can easily transition their courses to new Workbench-based materials; ANSYS periodically offers on-line orientation to help professors get started with these new materials.

The customized interface (shown below) has minimum input options, thus reducing efforts required to set up the problem. This allows any student who has a basic knowledge of fluid mechanics to obtain and visualize the results.

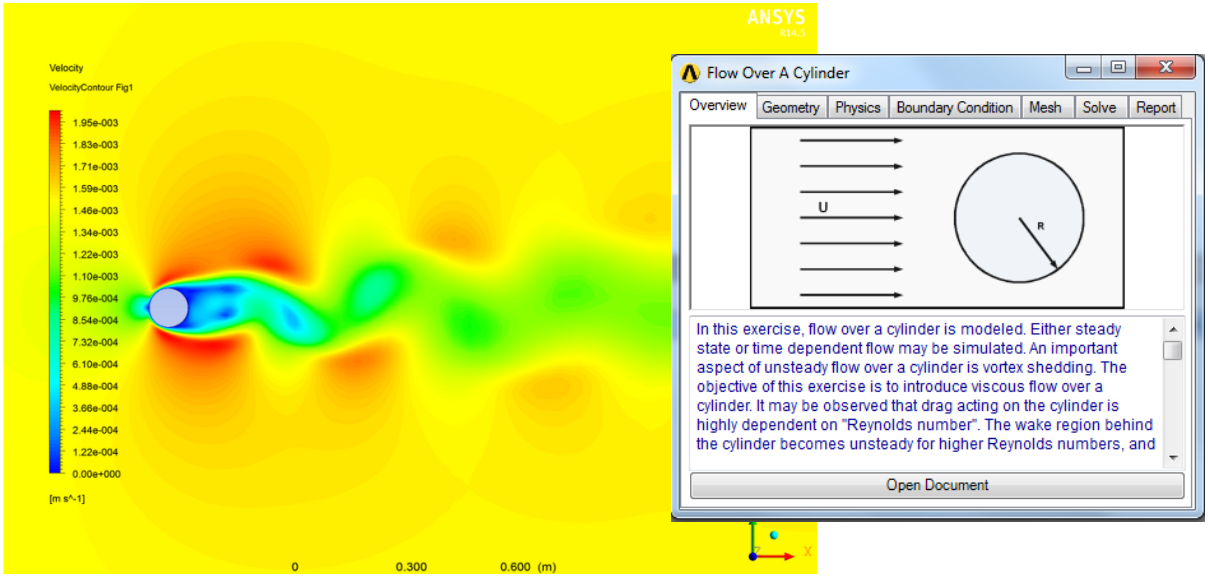

Figure 1: Flow over cylinder tutorial interface and result

Leading fluid mechanics textbooks such as Cengel and Cimbala have transitioned to the new Workbench-based interface. A separate body of CFD teaching material accompanies the Cengel and Cimbala text. These materials, based on customization of the Workbench interface, include documentation, installation instructions, etc.

However, not all professors wish to rely on the Flowlab-type interface. Some prefer to introduce their students to the Workbench interface directly, as it is intuitive and easy to use. ANSYS has collaborated with Professor Fred Stern of the University of Iowa to use the Workbench interface in the following courses: 57:020: Mechanics of Fluids and Transport Processes and 58:160: Intermediate Mechanics of Fluids.

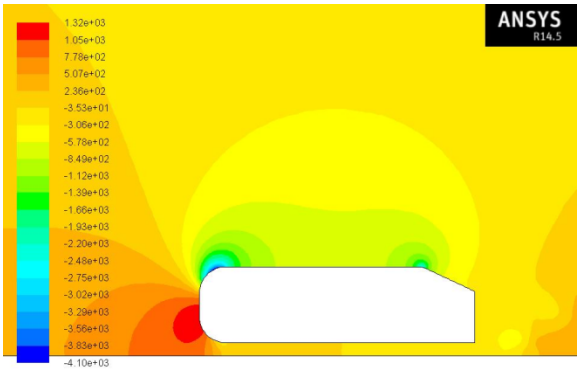

Figure 2: Flow over Ahmed body tutorial (Courtesy Prof Fred Stern, University of Iowa)

Professors wishing to use the direct Workbench approach can take advantage of these materials from Professor Stern or ANSYS. Professor Stern's material can be found at http://www.engineering.uiowa.edu/~me_160/. Today, most students prefer video-based tutorials over reviewing a PDF document that accompanies a project file. Therefore, ANSYS has converted Professor Sterns' tutorials to videos. These are useful for the fluids and introductory CFD courses.

For CFD classes at undergrad and grad levels, ANSYS has developed a set of video tutorials together with teaching materials that professors can use. These materials incorporate illustrative problems in the lab and help students learn about the use and application of CFD together with physics and concepts.

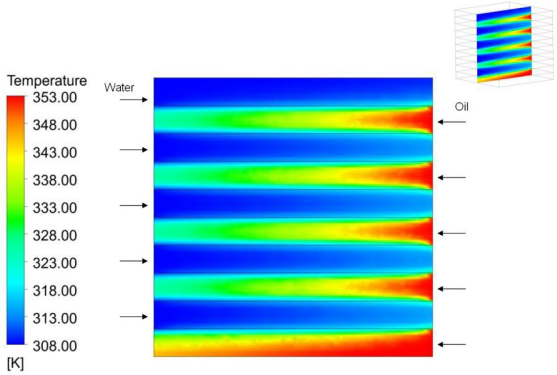

Figure 3: Heat transfer in a plate heat exchanger tutorial

The video tutorials guide students to generate the geometry and mesh, apply boundary conditions, and view the results. Teaching materials include: documentation for students explaining tutorial modeling details and teaching materials for professors that explain physics, visual observation, learning outcomes, and validation with analytical results.

Most of these materials are available on the ANSYS Customer and Student Portals. Here is the link to access the materials:
https://support.ansys.com/AnsysCustomerPortal/en_us/Knowledge+Resources/Academic+Training+Resources

To learn more about the teaching materials, or to enroll in an online orientation session, please contact vishal.ganore@ansys.com.