FLUENT tips

FLUENT tips

Two steps of getting started with FLUENT

FLUENT tutorial

The FLUENT tutorial developed by the department of MAE is a great resource to begin with. If you have never use FLUENT or learned about CFD before, it is a good idea to go through at least the first three sections: Introduction to CFD basics, Laminar pipe flow, Turbulent pipe flow.

This short course includes a concise overview of the fundamental ideas of CFD and steps-by-steps guides over the important features of FLUENT and Gambit. Practicing with the tutorial problems makes you get familiar with the software quickly.

Simulation Exercise

Now let's practice with a simple 2D model of a flocculation tank with one baffle, following the steps described below.

1. Download the case and data file and put them into the same directory.
2. Execute FLUETN 2ddp, click File>Read>Case & Data... and read the downloaded case and data file.
3. Plot the contour of energy dissipation rate: click Display>Contours, choose Contours of Turbulence and Turbulence Dissipation Rate(Epsilon) in the pull down menu and click display. A new window will appear with the contours of the Epsilon in the default color scale, normally too wide for the range of our interest. To change the color scale, in the Contours dialogue box, uncheck Auto Range under Options, set the Max to be 0.005 and click display. This should give a contour of energy dissipation rate with a reasonable color scale. Save the contour as a picture. (You can paste a screen shot in PowerPoint and save it as a picture.)
4. Now try solve the case you have read in again and see if you can get the same results:
    a. Click Solve>Initialize. Under the pull down menu under Compute from, choose inlet. The Initial Values should already be set-up correctly. Click Init and then Close at the bottom.
    b. Now click Solve>Iterate, set the Number of Iterations to be 5000 and click Iterate. The solution should be converged within several minutes. Plot the contour of energy dissipation again from your solution to compare with the downloaded data. Do you get the same results?

Other practical tips

1. Defining the material
   material should be defined in two steps:
    a. Add a material to the material list: on the main FLUENT window, click Define>Materials and add materials by manually inputting names and properties or by choosing from FLUENT database. This would create a **list** of material to choose from when you define the material for simulation in boundary conditions control panel.
    b. Boundary condition: on the main FLUENT window, click Define>Boundary Conditions>fluid>set, in the pull down box after Material Name, select from the list you created in step a the material you want to work with, and click OK.
    c. Note that you can also edit the material properties from the Boundary Condition control panel by clicking "Edit" after Material Name pull down box.
       It is important to keep these two steps in mind, because a lot of the previous simulations were carried out with air as the material when we mean to use water, since FLUENT uses air as the default material.
2. Summary report
   Click Report>Summary should give you a report summary of all of the parameters defined in FLUENT for a case/model. You can print it to the FLUENT window or save it as a text file. This is a great way of keeping track of the parameters of your model or compare between different models.

Important Resources

Whenever running into a how-to question, you can always find answers in FLUENT/Gambit documentation (both in PDF and HTML form), which includes complete guides on FLUENT, UDF , text interface commands(some features are not included in the graphical interface), etc.

http://www.fluentusers.com is another useful resource, which has tutorials, case studies, etc. Any student can get a free account using http://www.fluent.com/software/university/usc_student_form.htm and with it access the documentation at the USC as well as the CFD Learning Center. It generally takes a day or two for the password to be sent out after the request.