Flow over an Airfoil - Numerical Solution

Author: Benjamin Mullen, Cornell University

Numerical Solution

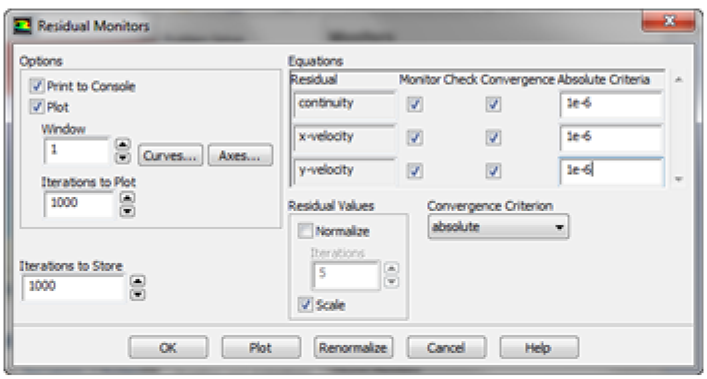
Methods

First, go to ***Solution > Solution Methods***. Everything in this section should have defaulted to what we want, but let's make sure that under ***Flow*** the selection is ***Second Order Upwind***. If this is the selection, we may move on.

Monitors

Now we are ready to begin solving the simulation. Before we hit solve though, we need to set up some parameters for how Fluent will solve the simulation.

Let's begin by going to ***Solution > Monitors***. In the *Monitors* Window, look under ***Residuals, Statistic, and Force Monitors***. Select ***Residuals - Print, Plot*** and press ***Edit***. In the *Residual Monitors* Window, we want to change all of the ***Absolute Criteria*** to `1e-6`. This will give us some further trust in our solution.



Initial Guess

Now, we need to initialize the solution. Go to ***Solution > Solution Initialization***. In the *Solution Initialization* Window, select ***Compute From > Inlet***. Ensure the values that appear are the same values we inputted in Step 5. If the are, initialize the solution by clicking ***Initialize***.

Solve

Once the solution has been initialized, we are ready to solve the simulation. Go to ***Solution > Run Calculation***. Change ***Number of Iterations*** to `3000`, then double click ***Calculate***. Sit back and twiddle your thumbs until Fluent spits out a converged solution.

Go to Step 6: Numerical Results

Go to all FLUENT Learning Modules