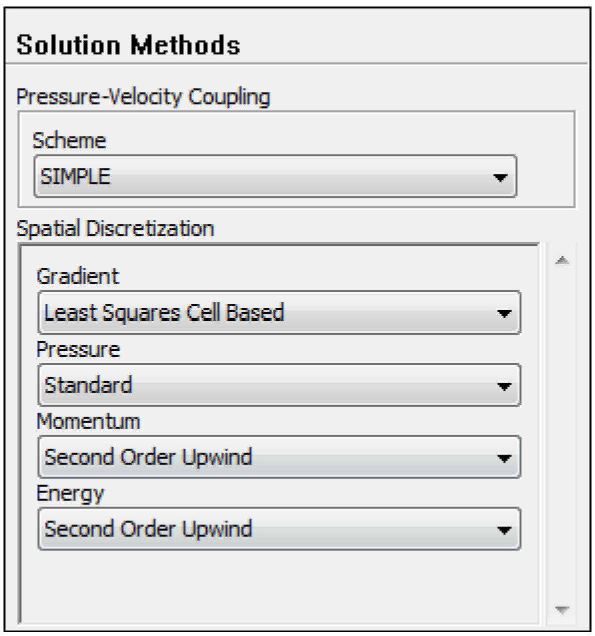
2D Steady Convection - Numerical Solution

Author: Benjamin Mullen, Cornell University

Numerical Solution

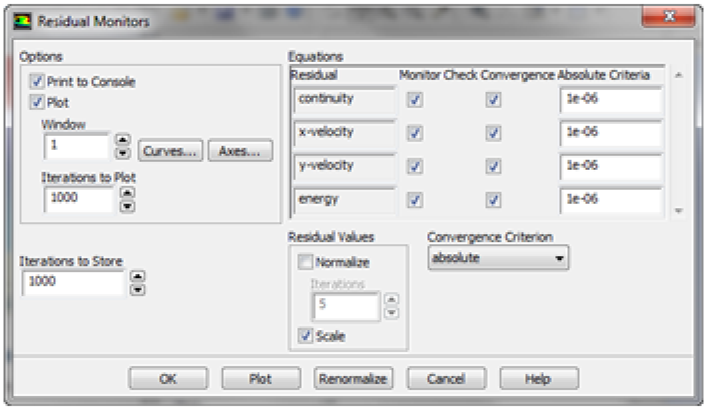
Solution Methods

In the *Outline* window, select ***Solution Methods*** to open the *Solution Methods* window. Under *Spatial Discretization*, change the option under *Momentum* from ***First Order Upwind*** to ***Second Order Upwind***. Under *Energy*, also change the option to ***Second Order Upwind***.



Monitors

In the *Outline* window, click ***Monitors*** to open the *Monitors* window. In the *Monitors* window, select ***Residuals - Print,Plot*** and press ***Edit...***. This will open the *Residual Monitors* window. We want to change the convergence criteria for our solution. Under *Equation* and to the right of ***Continuity***, change the ***Absolute Criteria*** to `1e-6`. Repeat for ***x-velocity***, ***y-velocity***, and ***energy***, then press ***OK***.



Reference Values

In the *Outline* window, select ***Reference Values***. Under *Compute From*, select ***Inlet***. Ensure that the values displayed are the values we specified.

Solution Initialization

In the *Outline* window, select ***Solution Initialization***. We need to make an "Initial Guess" to the solution so FLUENT can iterate to find the final solution. In the *Solution Initialization* window, select ***Standard Initialization*** then under ***Compute from***, select ***Inlet*** from the drop down box. Check to see that the values that generate match our inputted values, then press ***Initialize***

Data File Quantities

After we have initialized the solution, we can select additional quantities to be transferred over to CFD-Post. These quantities are not selected by default and so we must select them ourselves. Go to ***File>Data File Quantities***. Under ***Additional Quantities***, find and select ***Skin Friction Coefficient*** and click *OK*.

Run Calculation

In the *Outline* window, select ***Run Calculation***. Change the ***Number of Iterations*** to 1000. Double click ***Calculate*** to run the calculation. After the iterations have converged, save the project, and you may close the solver.

Go to Step 6: Numerical Results

Go to all FLUENT Learning Modules