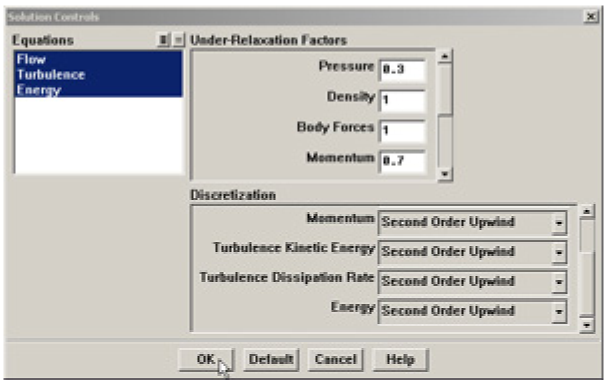
FLUENT - Forced Convection over a Flat Plate step 5

Step 5: Solve!

We'll use a second-order discretization scheme.

Main Menu > Solve > Controls > Solution...

Change ***Density, Momentum, Turbulence Kinetic Energy, Turbulence Dissipation Rate,*** and ***Energy*** all to ***Second Order Upwind***. Leave *Pressure* and *Pressure-Velocity Coupling* set to the default methods (***Standard*** and ***SIMPLE***, respectively). The other *Pressure* and *Pressure-Velocity Coupling* methods are useful for flows with particular characteristics not present in our problem.



Higher Resolution Image

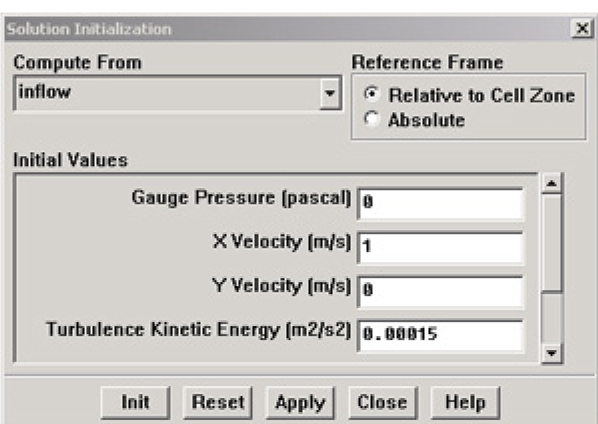
Click ***OK***.

Set Initial Guess

Initialize the flow field to the values at the inflow:

Main Menu > Solve > Initialize > Initialize...

In the *Solution Initialization* window that comes up, choose ***inflow*** under *Compute From*. The ***X Velocity*** for *all* cells will automatically be set to 1 m/s, the ***Y Velocity*** to 0 m/s and the ***Gauge Pressure*** to 0 Pa. These values have been taken from the inflow boundary condition.



Click ***Init***. This completes the initialization. Then click ***Close***.

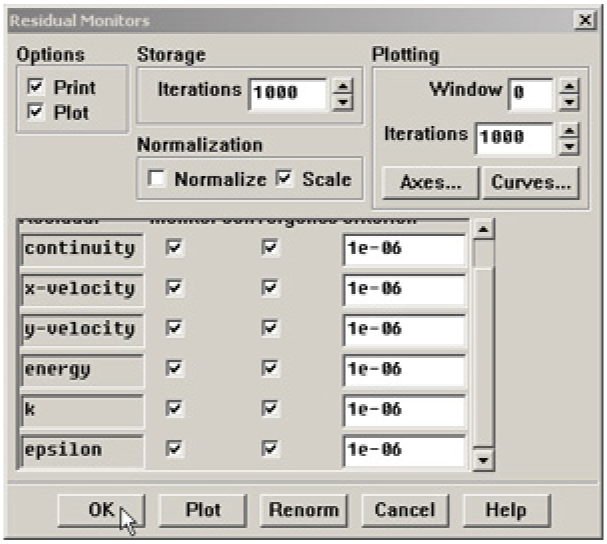
Set Convergence Criteria

FLUENT reports a residual for each governing equation being solved. The residual is a measure of how well the current solution satisfies the discrete form of each governing equation. We will iterate until the residual for each equation falls below 1e-6.

Main Menu > Solve > Monitors > Residual...

Change the residual under ***Convergence Criterion*** for ***continuity, x-velocity,*** and ***y-velocity, energy, k,*** and ***epsilon*** all to 1e-6.

Also, under ***Options***, select ***Print*** and ***Plot***. This will print the residuals in the main window and plot the residuals in the graphics window as they are calculated.



Click ***OK***.

This completes the problem specification. Save your work:

Main Menu > File > Write > Case...

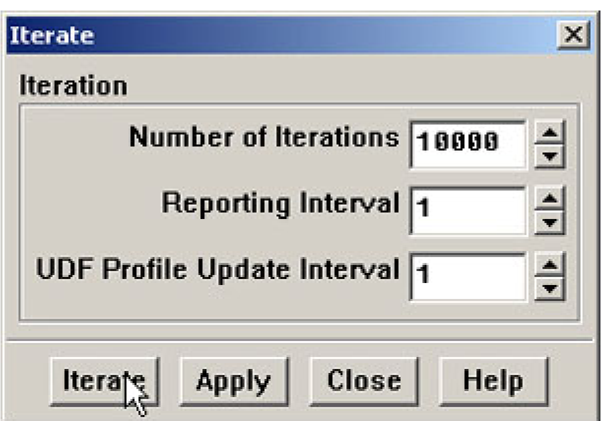
Type in `plate.cas` for ***Case File***. Click ***OK***. Check that the file has been created in your working directory. If you exit FLUENT now, you can retrieve all your work at any time by reading in this case file.

Iterate Until Convergence

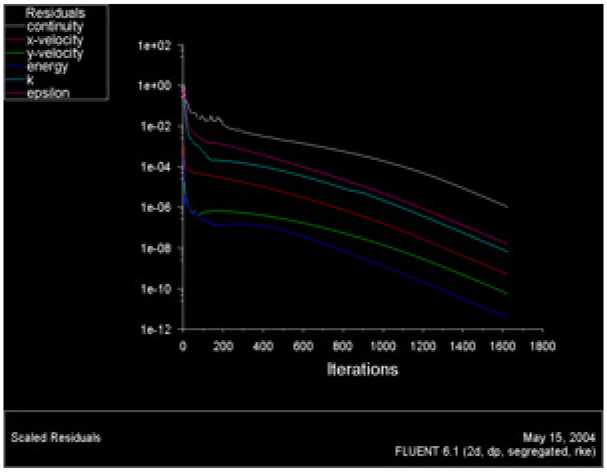
Start the calculation by running 10,000 iterations. The solution will converge before 10,000 iterations are performed, which will stop the iteration process.

Main Menu > Solve > Iterate...

In the *Iterate* Window, change the ***Number of Iterations*** to `10000`. Click ***Iterate***.

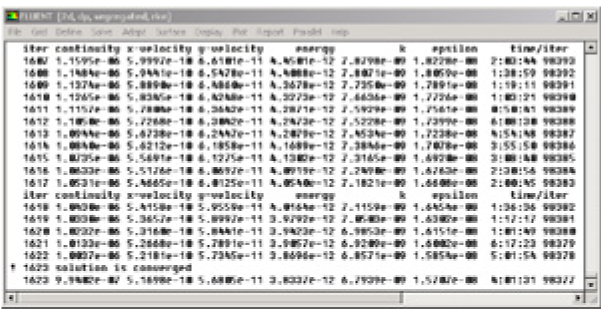


The residuals for each iteration are printed out as well as plotted in the graphics window as they are calculated.



Higher Resolution Image

The residuals fall below the specified convergence criterion of 1e-6 in approximately 1623 iterations.



Higher Resolution Image

Save the solution to a data file:

Main Menu > File > Write > Data...

Enter `plate.dat` for ***Data File*** and click ***OK***. Check that the file has been created in your working directory. You can retrieve the current solution from this data file at any time.

Go to Step 6: Analyze Results

See and rate the complete Learning Module

Go to all FLUENT Learning Modules