FLUENT - Turbulent Flow Past a Sphere - Step 5

UNDER CONSTRUCTION

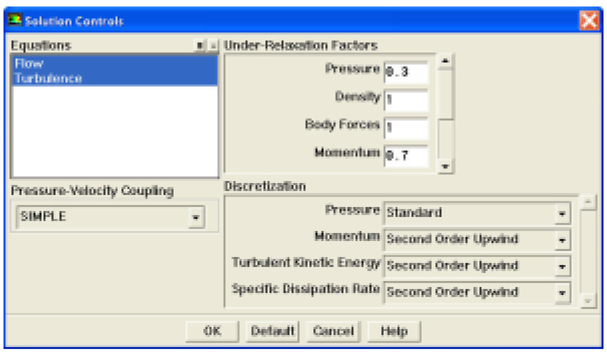
Author: Daniel Kantor and Andrew Einstein, Cornell University

Step 5: Solve!

We'll use a second-order discretization scheme.

Main Menu > Solve > Controls > Solution...

Take a look at the options available. Under ***Discretization***, set ***Pressure*** to ***Standard***, set ***Momentum*** to ***Second Order Upwind***, set ***Turbulent Kinetic Energy*** to ***Second Order Upwind***, and set ***Specific Dissipation Rate*** to ***Second Order Upwind***. All other values should remain at their default.



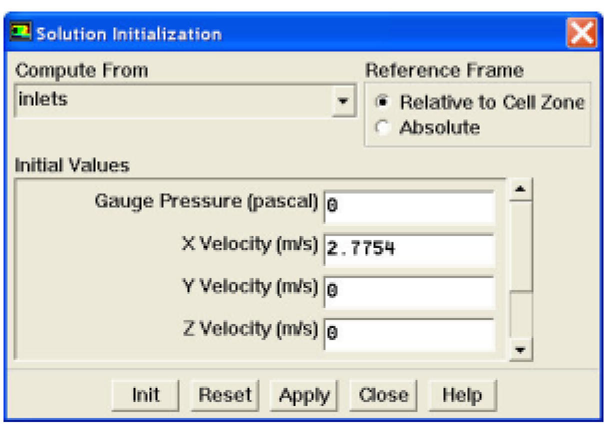
Click ***OK***.

Set Initial Guess

Initialize the flow field to the values at the inlet:

Main Menu > Solve > Initialize > Initialize...

In the *Solution Initialization* menu that comes up, choose ***inlet*** under ***Compute From***. The ***X Velocity*** for *all* cells will be set to 2.7754 m/s, the ***Y Velocity*** to 0 m/s and the ***Gauge Pressure*** to 0 Pa. These values have been taken from the inlet boundary condition.



Click ***Init***. This completes the initialization. ***Close*** the window.

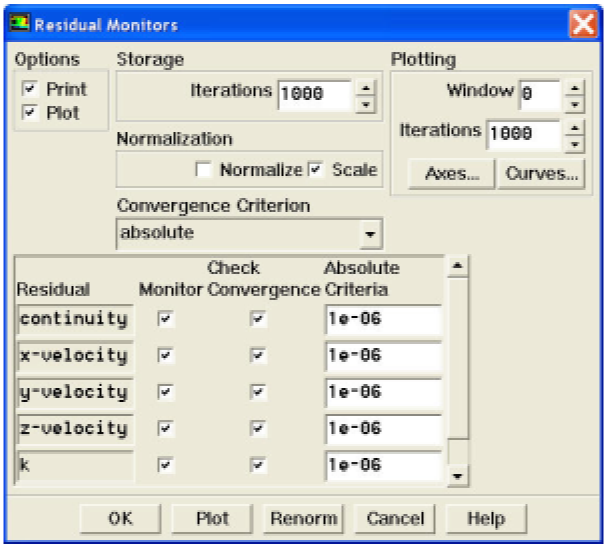
Set Convergence Criteria

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'll iterate the solution 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***, all to 1e-6.

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

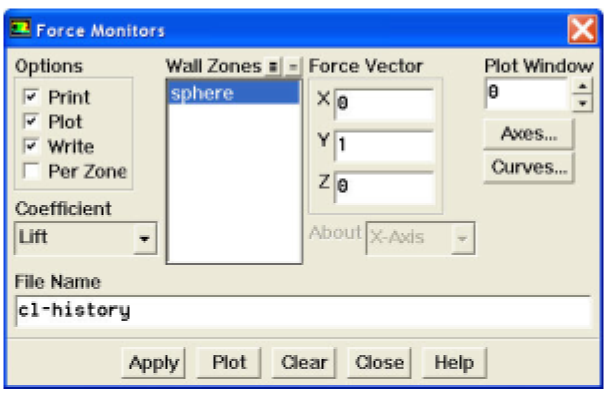


Click ***OK***.
Monitor also the drag and lift coefficient on the sphere.

Main Menu > Solve > Monitors > Force...

Select ***Sphere*** under ***Wall Zones***. Under ***Options***, select ***Print***, ***Plot*** and ***Write***. Under ***Coefficient***, choose ***Drag***. Click ***Apply***.

Choose ***Sphere*** under ***Wall Zones***. Under ***Options***, select ***Print***, ***Plot*** and ***Write***. Under ***Coefficient***, choose ***Lift***. Click ***Apply***.

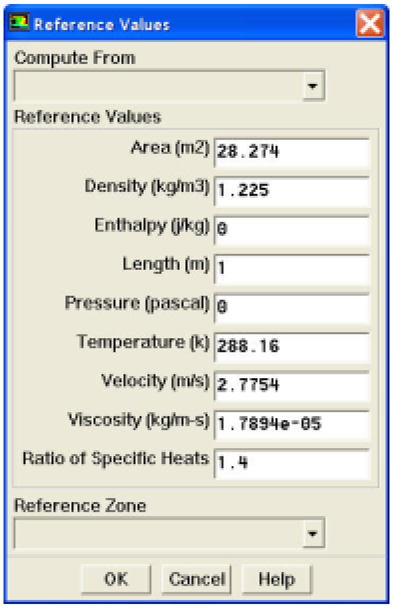


Setting Reference Values

To plot C_d and C_l we need to set the reference value.

Main Menu > Report > Reference Values...

Under ***Reference Values***, change ***Area*** to *28.274*, ***Density*** to *1.225*, ***Velocity*** to *2.7754* and ***Viscosity*** to *1.7894E-05*.



Click ***OK***.

This completes the problem specification. Save your work:

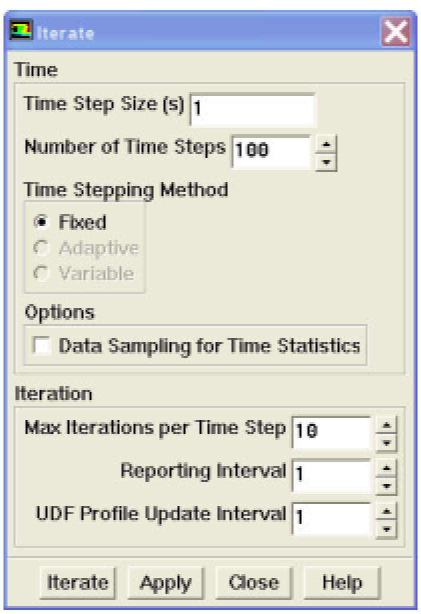
Main Menu > File > Write > Case...

Type in `SingleSphere.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

Main Menu > Solve > Iterate...

In the *Iterate Window* that comes up, change the ***Time Step Size*** to `1`, change the ***Number of Time Steps*** to `100`, and change ***Max Iterations per Time Step*** to `10`. Click ***Iterate***.



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

Save the solution to a data file:

Main Menu > File > Write > Data...

Enter `SingleSphere.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