Steady Flow Past a Cylinder - Numerical Solution

Authors: John Singleton and Rajesh Bhaskaran, Cornell University

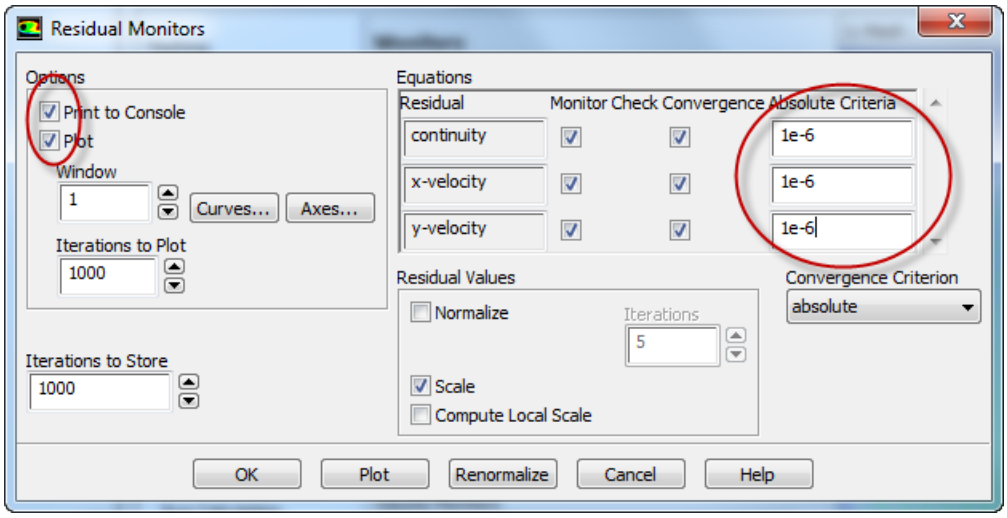
Numerical Solution

Second Order Upwind Momentum Scheme

Solution > Solution Methods > Spatial Discretization.
Set ***Momentum*** to ***Second Order Upwind***

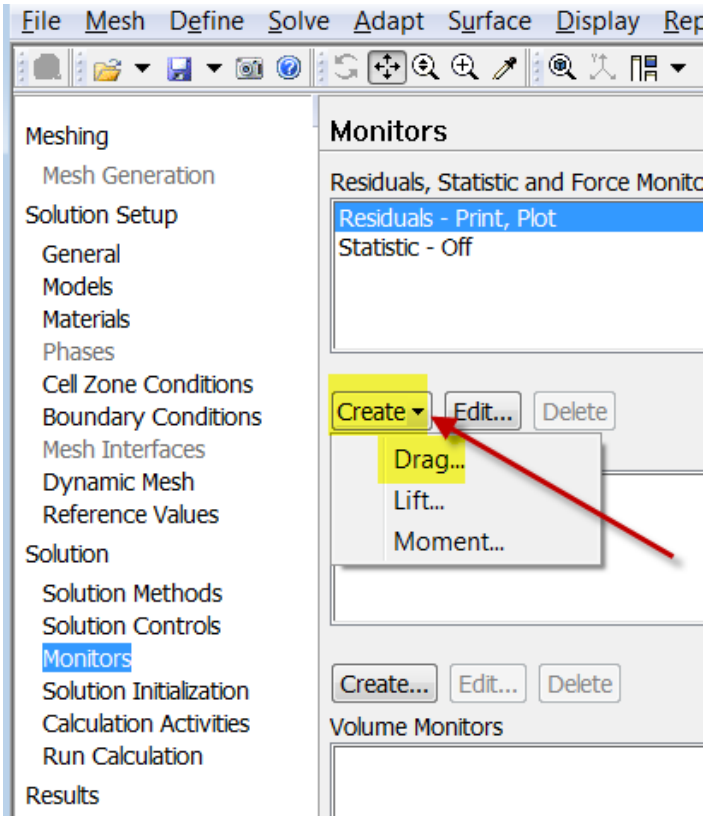
Convergence Criterion

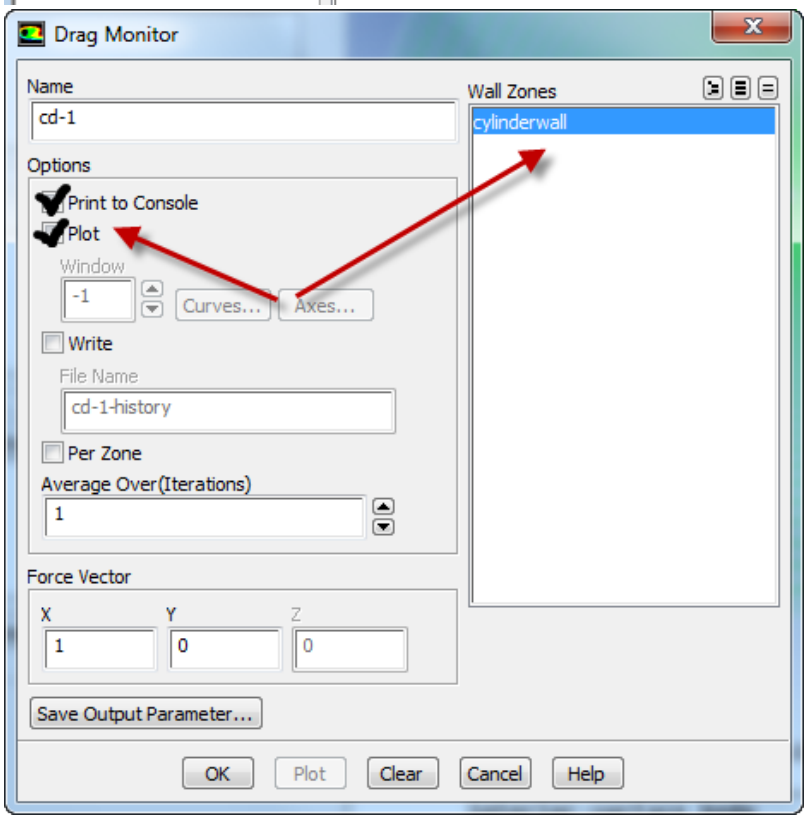
Solution > Monitors > Residuals > Edit....
Set the ***Absolute Criteria*** for ***continuity***, ***x-velocity*** and ***y-velocity*** to 1e-6. Click ***ok***



Solution > Monitors > Create > Drag

NOTE: In ANSYS 18.2, implementing the drag monitor is slightly different. The pictures below do not match exactly with version 18.2. See this tutorial video for version 18.2.





Then check ***Print to Console*** and ***Plot***. Next, click ***cylinderwall***, which is located under ***Wall Zones***. Lastly, click ***ok***

Initial Guess

Solution > Solution Initialization > Standard Initialization.
Set ***Compute From*** to ***farfield1.*** Alternately, you can simply set ***X Velocity*** to 1 m/s. Then, click ***Initialize***.

Iterate Until Convergence

Solution > Run Calculation.
Set the ***Number of Iterations*** to **2000**. Then, click ***Calculate***. (You may have to hit ***Calculate*** twice.) Now, have a cup of coffee. The solution should converge after approximately 1600 iterations. **NOTE**: In newer versions of ANSYS, solution may converge in lesser number of iterations.

Save Project

Go to Step 6: Numerical Results

Go to all FLUENT Learning Modules