

# Steady Flow Past a Cylinder - Numerical Solution

Authors: John Singleton and Rajesh Bhaskaran, Cornell University

[Problem Specification](#)

[1. Pre-Analysis & Start-Up](#)

[2. Geometry](#)

[3. Mesh](#)

[4. Physics Setup](#)

[5. Numerical Solution](#)

[6. Numerical Results](#)

[7. Verification & Validation](#)

[Exercises](#)

[Comments](#)

## Numerical Solution

### Second Order Upwind Momentum Scheme

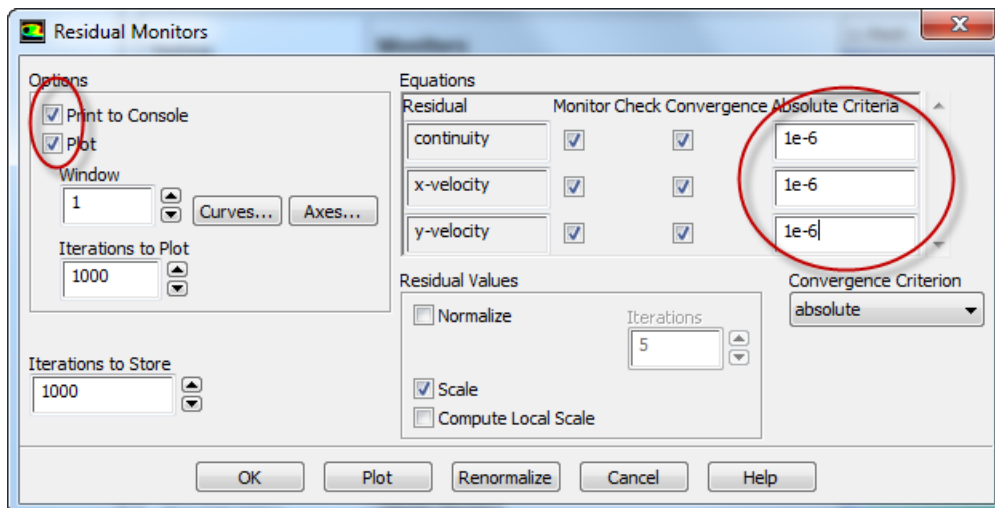
*Solution > Solution Methods > Spatial Discretization.*

Set *Momentum* to *Second Order Upwind*

### Convergence Criterion

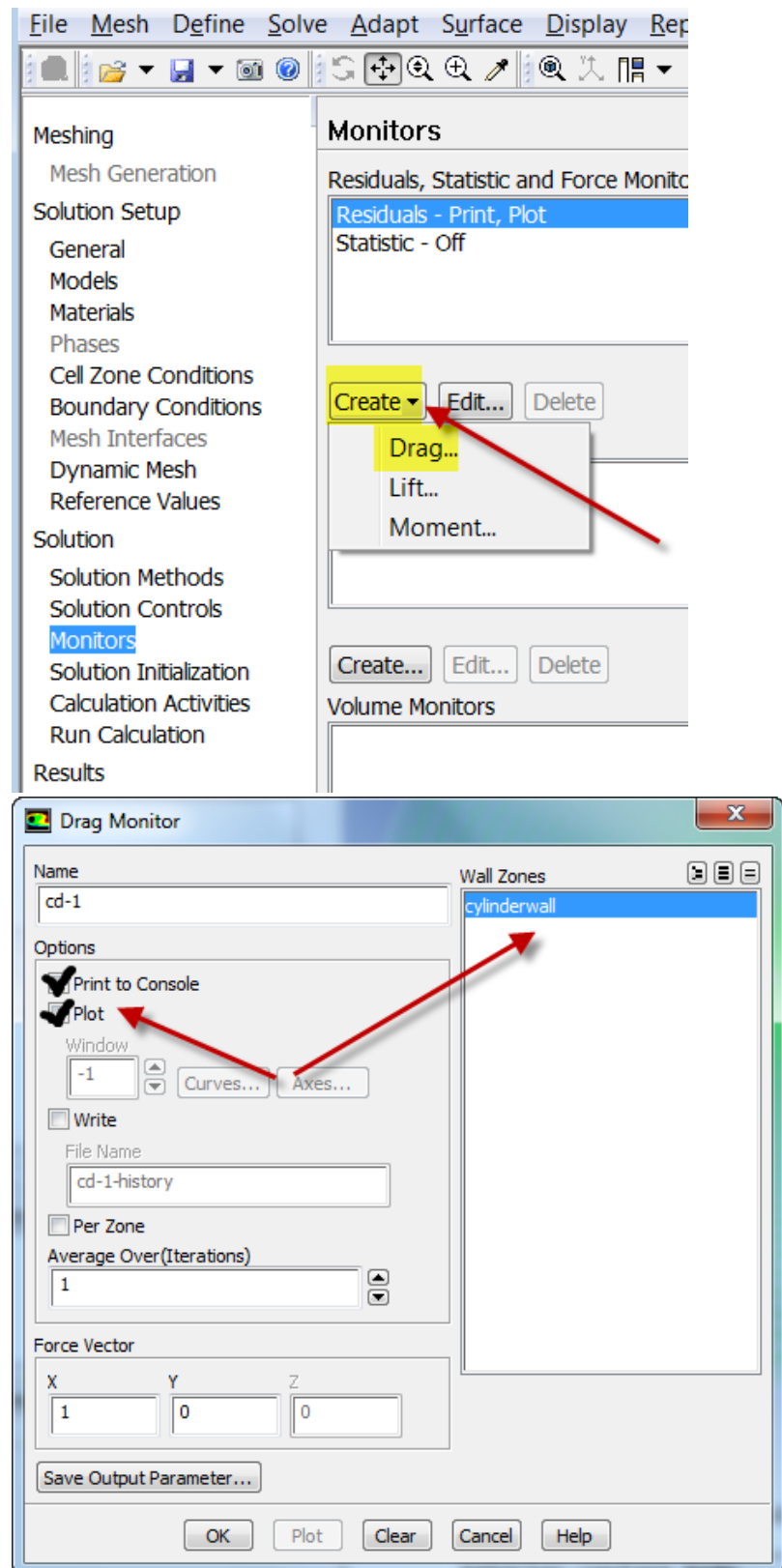
*Solution > Monitors > Residuals > Edit...*

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](#)