

# FLUENT - Laminar Pipe Flow Step 5

Problem Specification  
1. Pre-Analysis & Start-up  
2. Mesh  
3. Geometry  
4. Setup (Physics)  
**5. Solution**  
6. Analysis & Results  
7. Verification & Validation  
Problem 1  
Problem 2



## Useful Information

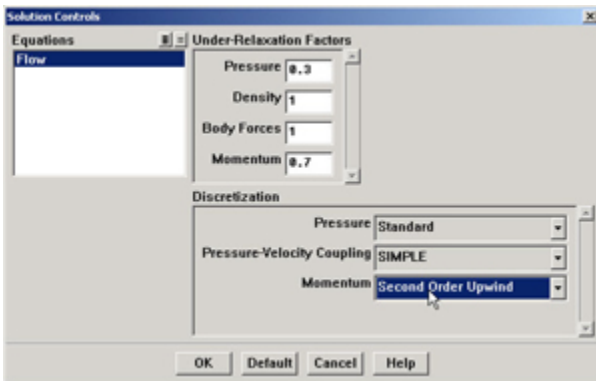
[Click here](#) for the FLUENT 12

## Step 5: Solve!

We'll use a second-order discretization scheme.

**Main Menu > Solve > Controls > Solution...**

Change *Momentum* to *Second Order Upwind*.



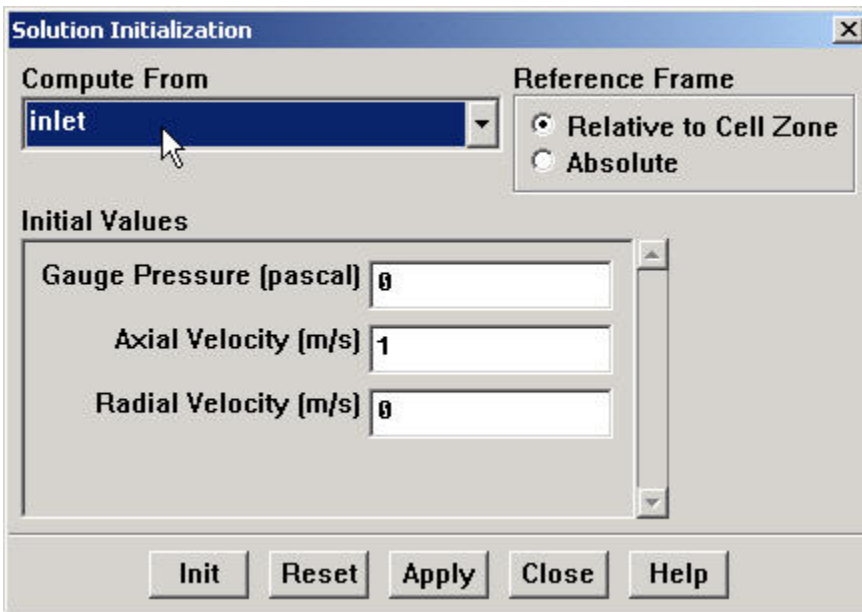
Click *OK*.

## 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 *Axial Velocity* for *all* cells will be set to 1 m/s, the *Radial Velocity* to 0 m/s and the *Gauge Pressure* to 0 Pa. These values have been taken from the inlet boundary condition.



**Solution Initialization**

Compute From: **inlet**

Reference Frame: ☒ Relative to Cell Zone ☐ Absolute

Initial Values:

Gauge Pressure (pascal):

Axial Velocity (m/s):

Radial Velocity (m/s):

Buttons: Init, Reset, Apply, Close, Help

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

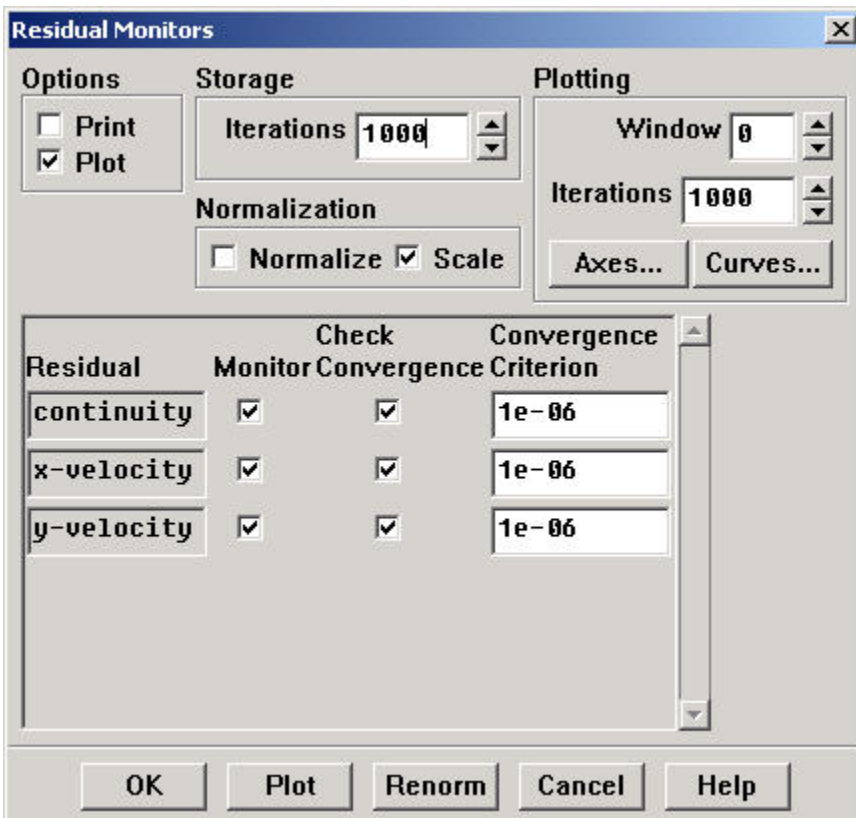
### Set Convergence Criteria

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.



**Residual Monitors**

Options: ☐ Print ☒ Plot

Storage: Iterations

Normalization: ☐ Normalize ☒ Scale

Plotting: Window  Iterations  Axes... Curves...

Residual	Check Monitor	Convergence Criterion
continuity	<input checked="" type="checkbox"/>	<input type="text" value="1e-06"/>
x-velocity	<input checked="" type="checkbox"/>	<input type="text" value="1e-06"/>
y-velocity	<input checked="" type="checkbox"/>	<input type="text" value="1e-06"/>

Buttons: OK, Plot, Renorm, Cancel, Help

Click **OK**.

This completes the problem specification. Save your work:

**Main Menu > File > Write > Case...**

Type in `pipe.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.

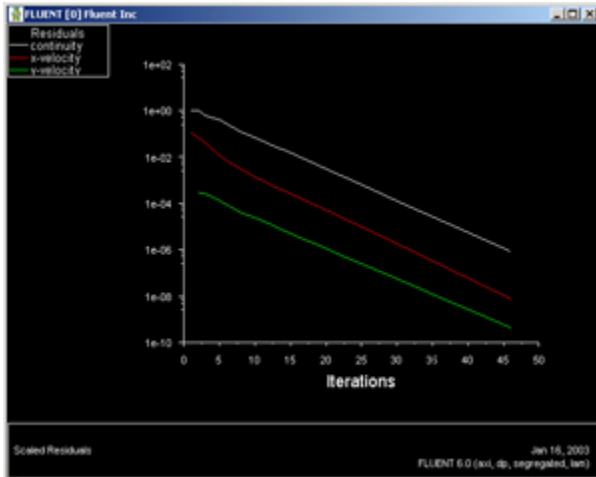
## Iterate Until Convergence

Start the calculation by running 100 iterations:

**Main Menu > Solve > Iterate...**

In the *Iterate Window* that comes up, change the **Number of Iterations** to 100. Click **Iterate**.

The residuals for each iteration is 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 about 46 iterations. Actual number of convergence steps may vary slightly.

```
iter      time/iter
?    46 solution is converged
```

Save the solution to a data file:

**Main Menu > File > Write > Data...**

Enter `pipe.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](#)