Step 5: Solve!

We'll use a second-order discretization scheme.

Main Menu > Solve > Controls > Solution...

Change Momentum to Second Order Upwind.

quations	Under-Relaxation Factors
Flow	Pressure 0.3
	Density 1
	Body Forces 1
	Momentum 0.7
ressure-Velocity Co	upling Discretization
SIMPLE	Pressure Standard
	Momentum Concernment of the second se

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 1 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.

Compute From	Reference Frame
farfield1	Relative to Cell Zone Absolute
Gauge Pressure (pascal) X Velocity (m/s) Y Velocity (m/s) 0	*
	-

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-5.

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

Options	St	orage				Plottin	ng		
□ Print □ Plot		Ite	erations	1000	•		Wind	low 0	-
	N	ormalizat	lion			Itera	tions	1000	-
		□ Normalize 🖓 Scale		Axes		Curves			
	C	onverger	nce Crite	rion		-			
	a	bsolute							
Residual		Monitor	Check Converg	jence	Absolu Criteria	te 1			
continui	ty	2			1e-5				
x-veloci	ty				1e-5				
y-veloci	ty	1			1e-5				
							-		

Click OK.

Monitor also the drag coefficient on the cylinder.

Main Menu > Solve > Monitors > Force...

Select cylinder under Wall Zones. Under Options, select Plot and Write. Note that Plot Window is 1.

Setting Reference Values

To plot C $_{\rm d}\text{,}$ we need to set the reference value.

Unable to find DVI conversion log file.

Note that cross sectional area for a 2D cylinder is the diameter of the cylinder.}

Main Menu > Report > Reference Values...

Under Reference Values, change Area to 2, Density to 1, Velocity to 1 and Viscosity to 0.1.

Reference Values	×
Compute From	
farfield1	•
Reference Values	
Area (m2)	2
Density (kg/m3)	1
Depth (m)	1
Enthalpy (j/kg)	9
Length (m)	1
Pressure (pascal)	9
Temperature (k)	288.16
Velocity (m/s)	1
Viscosity (kg/m-s)	9.1
Ratio of Specific Heats	1.4
Reference Zone	•
OK Cance	Help

This completes the problem specification. Save your work:

Main Menu > File > Write > Case...

Type in cylinder.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 1000 iterations:

Main Menu > Solve > Iterate...

In the Iterate Window that comes up, change the Number of Iterations to 1000. Click Iterate.

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



Higher Resolution Image

🖉 FLUTNT (1) Flann inc							CIDX
Cđ	9.000 0.0000 0.000 0.000 0.000 0.000 0.000 0.000 0.000 0.000 0.000 0.000 0.000		le				
	0.000	100	200	xii Iterations	400	608	805
Drug Consequence History				2019/22112		FURN	Feb 38, 2009 5.3 (Dir, dp. plote, tard)

Higher Resolution Image

Save the solution to a data file:

Main Menu > File > Write > Data...

Enter cylinder.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