

معرفی مسئله

در این پروژه جریان خارجی آرام تراکم ناپذیر ناپایدار روی استوانه به عنوان یک نمونه از اجسام ضخیم به کمک نرم افزار فلونت شبیه سازی شده و پدیده های فیزیکی بحث شده در کلاس مشاهده خواهد شد.

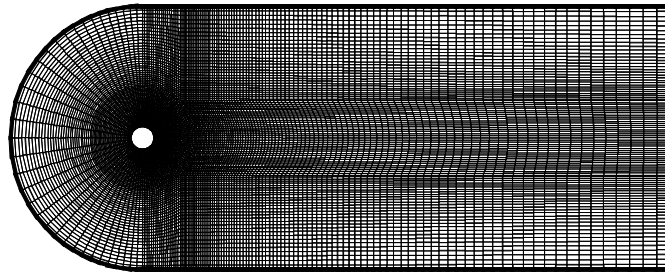
هندسه

جریان هوا روی یک استوانه با طول بینهایت با قطر 2 سانتی متر مد نظر است. سرعت وزش در ورودی و یا سرعت در بالا دست را به گونه ای در نظر بگیرید که جریان با اعداد رینولدز 25، 50، 100، 200، 300، 1000، ایجاد شود. شرط عدم لغزش روی استوانه برقرار است. سرعت در مرز ورودی و اطراف استوانه ثابت و در مرز انتهایی فشار برابر با فشار اتمسفر است. ابعاد میدان حل به شکل زیر است:

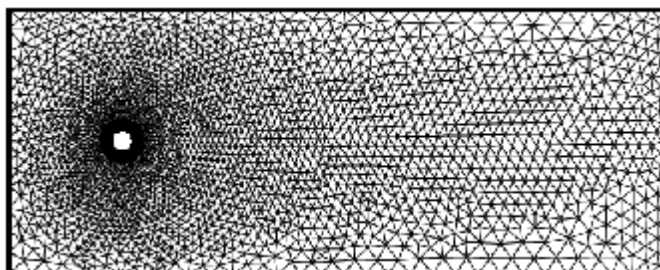
- 25 برابر قطر در ناحیه دنباله
- 5 برابر قطر در جلوی استوانه
- 15 برابر قطر ارتفاع میدان حل

برای مثال از شبکه های زیر می توان استفاده کرد:

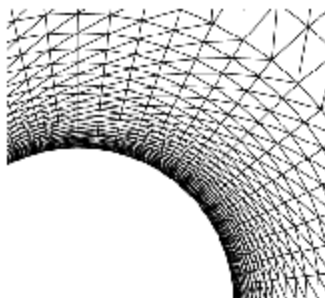
- شبکه ی با سازمان که در انجام پروژه به شما تحویل داده خواهد شد.



- شبکه ی بی سازمان



نکته: مشابه جریان روی صفحه تخت نزدیک سطح استوانه از المان‌های لایه مرزی مطابق شکل زیر استفاده میشود.



خواسته‌های فیزیکی

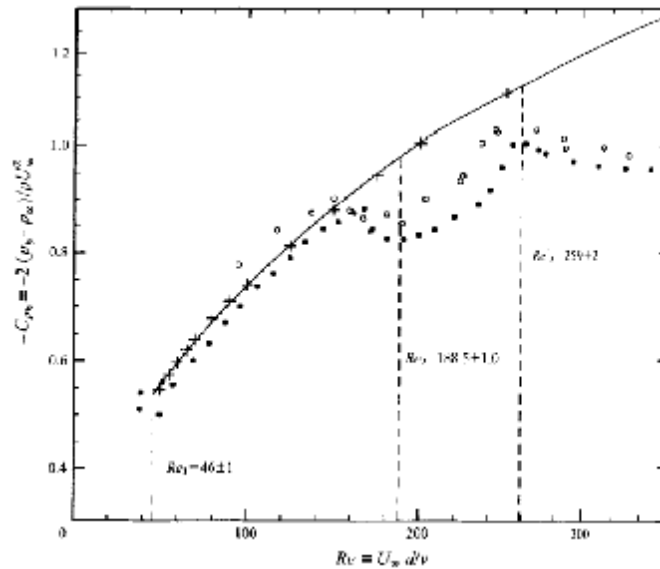
پس از شبیه‌سازی، موارد زیر برای اعداد رینولدز مختلف باید محاسبه شده و با مقادیر تحلیلی موجود مقایسه شود.

- محل جدایش روی سطح استوانه در زمان نهایی،
- توزیع نیروی درگ و لیفت روی سطح استوانه در زمان نهایی،
- تغییرات نیروی درگ و لیفت کل استوانه با زمان،
- مشاهده لایه مرزی شکل گرفته روی استوانه در زمان نهایی،
- مشاهده سرعت و فشار در نقطه سکون در زمان نهایی،
- رسم خطوط جریان و بررسی رفتار جریان در اعداد رینولدز مختلف و مشاهده جدایش گردابه‌ها،
- رسم توزیع فشار و مشخص کردن مناطق کم‌فشار و پرفشار،
- محاسبه فرکانس جدایش گردابه‌ها و مقایسه با فرکانس بی‌بعد شده $0/2$ (عدد اشتروهال)

سوال امتیازی

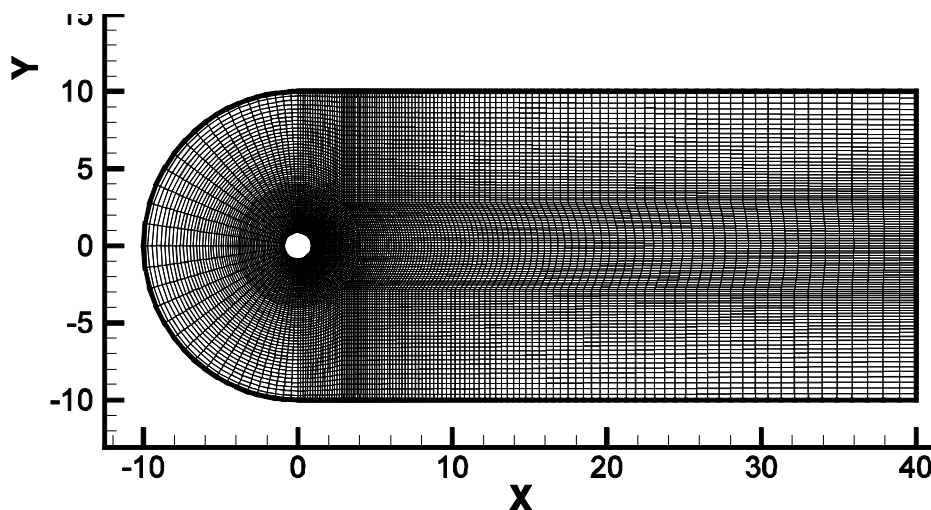
شکل زیر نمایانگر تغییرات ضریب فشار متوسط با عدد رینولدز می‌باشد. بر طبق نمودار فوق این شبیه‌سازی که به صورت دوبعدی انجام می‌شود (نقاط توپر و توخالی) با داده‌های تجربی (خط پیوسته) تا عدد رینولدز حدود 180 توافق خوبی دارند و پس از آن از همدیگر واگرا می‌شوند. مشابه همین پدیده در مورد ضریب درگ نیز مشاهده می‌شود. به نظر شما دلیل این پدیده چیست؟

این پدیده را برای نیروی درگ به کمک مقادیر تجربی بررسی کنید.



مشخصات هندسه

همانگونه که در شکل زیر پیداست، ابعاد میدان فیزیکی 40×20 و قطر استوانه برابر با 2 است.



Problem Specification

The purpose of this tutorial is to illustrate the setup and solution of an unsteady flow past a circular cylinder and to study the vortex shedding phenomenon. Flow past a circular cylinder is one of the classical problems of fluid mechanics. For this problem, we will be looking at Reynolds number of 150. We know $D = 2$ m. To obtain proper Reynolds number, we can arbitrarily set ρ , V and μ . For example to obtain $Re=150$, let's set $\rho = 75$ (kg/m^3), $V = 1$ (m/s) and $\mu = 1$ (kg/m.s). For other Reynolds number you should change these parameters to achieve a proper value.

Important: don't forget to save your work after each step!

Import Grid

Main Menu > File > Read > Case...

Navigate to the working directory and select the cylinder.msh file. This is the mesh file that was created using the preprocessor *GAMBIT* in the previous step. FLUENT reports the mesh statistics as it reads in the mesh.

Also, take a look under zones. We can see the five zones *farfield1*, *farfield2*, *farfield3*, *farfield4*, and *cylinder* that we defined in *GAMBIT*.

First, we check the grid to make sure that there are no errors.

Main Menu > Grid > Check

Any errors in the grid would be reported at this time. Check the output and make sure that there are no errors reported. Check the grid size:

Main Menu > Grid > Info > Size

The following info should appear:

| Grid Size | | | | |
|-----------|-------|-------|-------|------------|
| Level | Cells | Faces | Nodes | Partitions |
| 0 | 14040 | 28308 | 14268 | 1 |

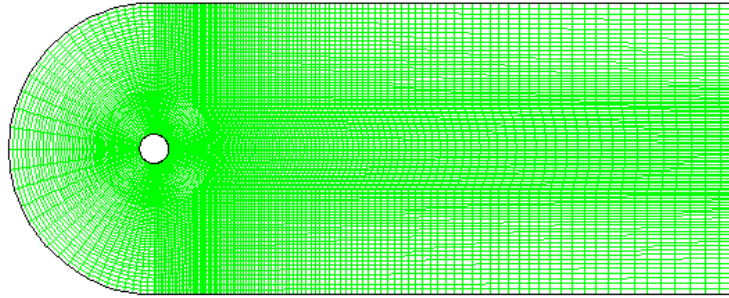
1 cell zone, 6 face zones.

Display the grid

Main Menu > Display > Grid...

Make sure all 6 items under *Surfaces* is selected. Then click *Display*. The graphics window opens and the grid is displayed in it. You can now click *Close* in the *Grid*

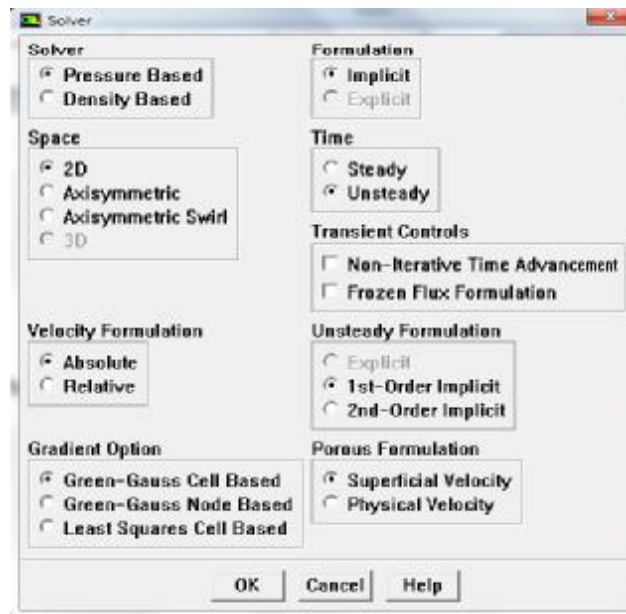
Display menu to get back some desktop space. The graphics window will remain.



Define Solver Properties

Main Menu > Define > Models > Solver

Under *Time*, select *Unsteady*. We will use the default *1st-Order Implicit Unsteady Formulation*. Click *OK*.



Main Menu > Define > Models > Viscous

Laminar flow is the default. So we don't need to change anything in this menu. Click *Cancel*.

Main Menu > Define > Models > Energy

For incompressible flow, the energy equation is decoupled from the continuity and momentum equations. We need to solve the energy equation only if we are interested in determining the temperature distribution. We will not deal with temperature in this example. So leave the *Energy Equation* unselected and click *Cancel* to exit the menu.

Define material properties

Change *Density* and *Viscosity* to proper value to approach the proper Reynolds number.

Click *Change/Create*. Close the window.

Define Operating Conditions

For all flows, FLUENT uses gauge pressure internally. Any time an absolute pressure is needed, it is generated by adding the operating pressure to the gauge pressure. We'll use the default value of 1 atm (101,325 Pa) as the *Operating Pressure*.

Click *Cancel* to leave the default in place.

Define Boundary Conditions

We'll now set the value of the velocity at the inlet and pressure at the outlet. Use the following table to set boundary type of each zone.

| Zone | Type |
|------------------|-------------------------------|
| <i>farfield1</i> | velocity-inlet, $V_x = U$ m/s |
| <i>farfield2</i> | velocity-inlet, $V_x = U$ m/s |
| <i>farfield3</i> | velocity-inlet, $V_x = U$ m/s |
| <i>farfield4</i> | pressure-outlet |
| <i>cylinder</i> | wall |

Main Menu > Define > Boundary Conditions...

Select *farfield1* under *Zone*. Change the *Type* of boundary as *velocity-inlet*. A new window will pop up. Change *Magnitude, Normal to Boundary* to *Components* under *Velocity Specification Method*. Input value U next to *XVelocity*. (U is the inlet velocity based on calculated Reynolds number). Click *OK*. Do the same for *farfield2* and *farfield3*.

The (absolute) pressure at the farfield downstream (*farfield4*) is 1 atm. Since the operating pressure is set to 1 atm,

the outlet gauge pressure = outlet absolute pressure – operating pressure = 0.

Choose *farfield4* under *Zone*. The *Type* of this boundary is *pressureoutlet*.

Click on *Set...*. The default value of the *Gauge Pressure* is 0. Click *Cancel* to leave the default in place.

Lastly, click on *cylinder* under *Zones* and make sure *Type* is set as *wall*.

Click *Close* to close the *Boundary Conditions* menu.

Now we are ready to solve! But before it:

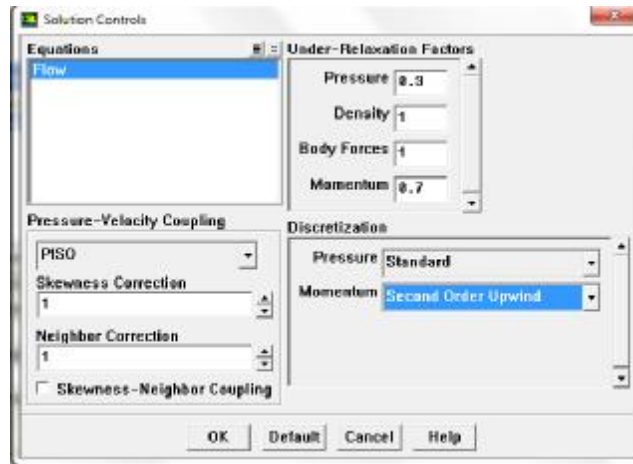
Set Solution Control

Main Menu > Solve > Controls > Solution...

Select *PISO* from the *Pressure-Velocity Coupling* drop-down list.

Uncheck *Skewness-Neighbor Coupling*.

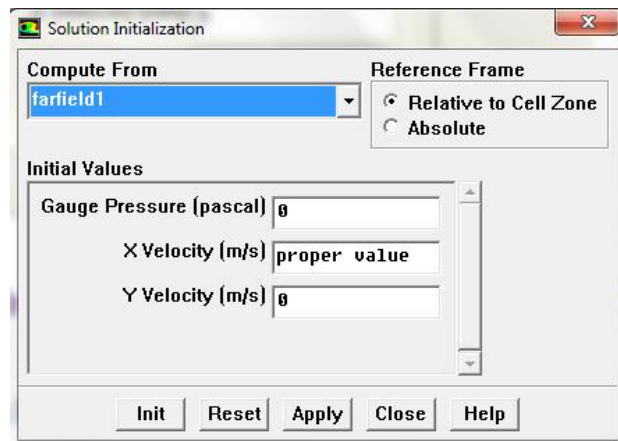
Select *Second Order Upwind* from the Momentum drop-down list in the *Discretization* group box. Click *OK* to close the *Solution Controls* panel.



Initializing

Main Menu > Solve > Initialize > Initialize...

In the *Solution Initialization* menu that comes up, choose *farfield1* under *Compute From*. The *X Velocity* for all cells will be set to U m/s, the *Y Velocity* to 0 m/s and the *Gauge Pressure* to 0 Pa. These values have been taken from the *farfield1* boundary condition.



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

Patch Region

We will patch the upper region downstream of the flow to create asymmetry so that we can obtain stable oscillation of vortex shedding faster.

To do this, we will create a register to patch the Y velocity in downstream of cylinder.

Main Menu > Adapt > Region...



Enter 1 and 40 for *X Min* and *X Max*. Enter 0 and 10 for *Y Min* and *Y Max*. Click *Mark*. FLUENT will print the following message in the console window:

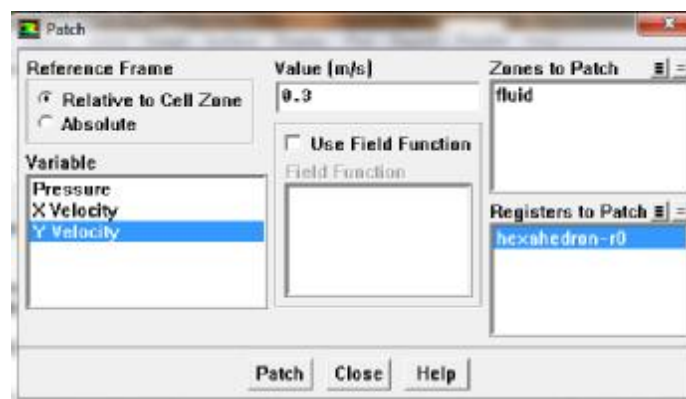
5416 cells marked for refinement, 0 cells marked for coarsening

Close the *Region Adaption* panel.

We will now patch Y velocity in the registered region.

Main Menu > Solve > Initialize > Patch...

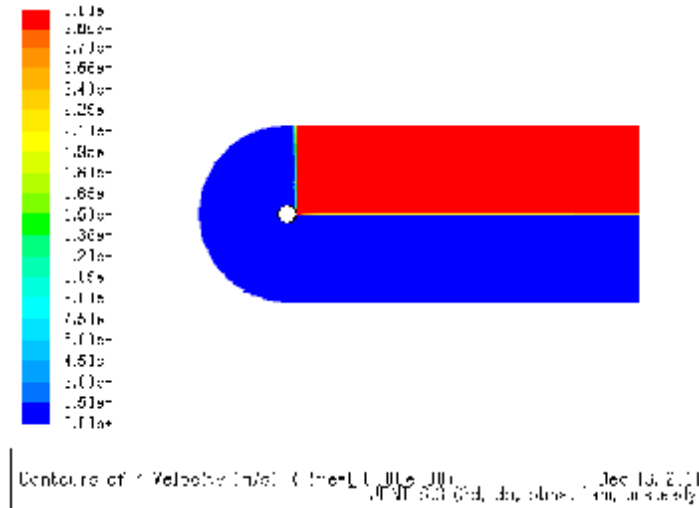
Select *hexahedron-r0* from the *Registers to Patch*. Select *Y Velocity* from the *Variable* selection list. Enter 0.3 for *Value*. Click *Patch*.



To check whether you have patch the region, plot contour of velocity in the y direction.

Main Menu > Display > Contours...

Select *Velocity...* and *Y Velocity* under *Contours of* drop-down list. Make sure to check the *Filled* under *Options*. Click *Display*.



As can be seen, the Y Velocity is zero everywhere except for the patched region, we have Y Velocity of 0.3 m/s.

Set Convergence Criteria

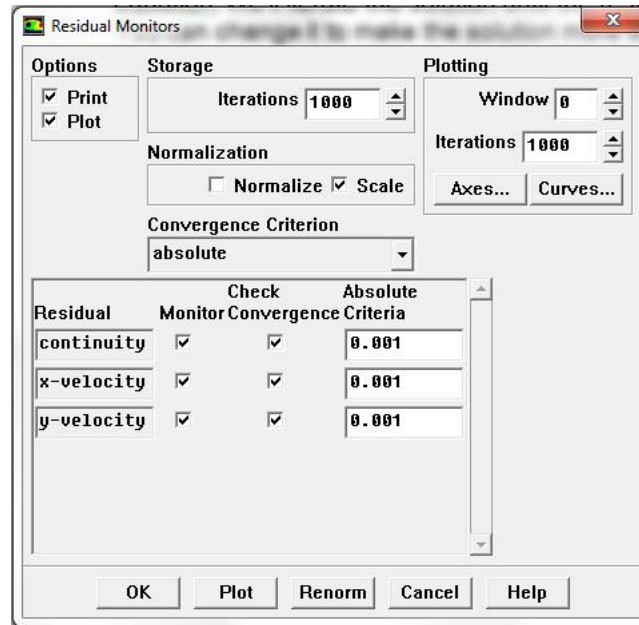
FLUENT reports a residual for each governing equation being solved. The residual is a measure of how well the current solution satisfies the discrete form of each governing equation. We'll iterate the solution until the residual for each equation falls below $1e-3$. You can change it to make the solution more exact.

Main Menu > Solve > Monitors > Residual...

Default value for *Convergence Criterion* for *continuity*, *x-velocity*, and *yvelocity* is $1e-3$.

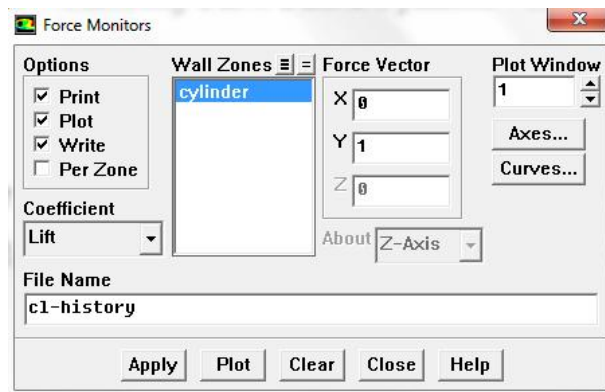
Also, under *Options*, select *Plot* and *Print*. This will plot the residuals in the graphics window as they are calculated.

Click *OK*.



Monitor also the lift coefficient on the cylinder:

Main Menu > Solve > Monitors > Force...

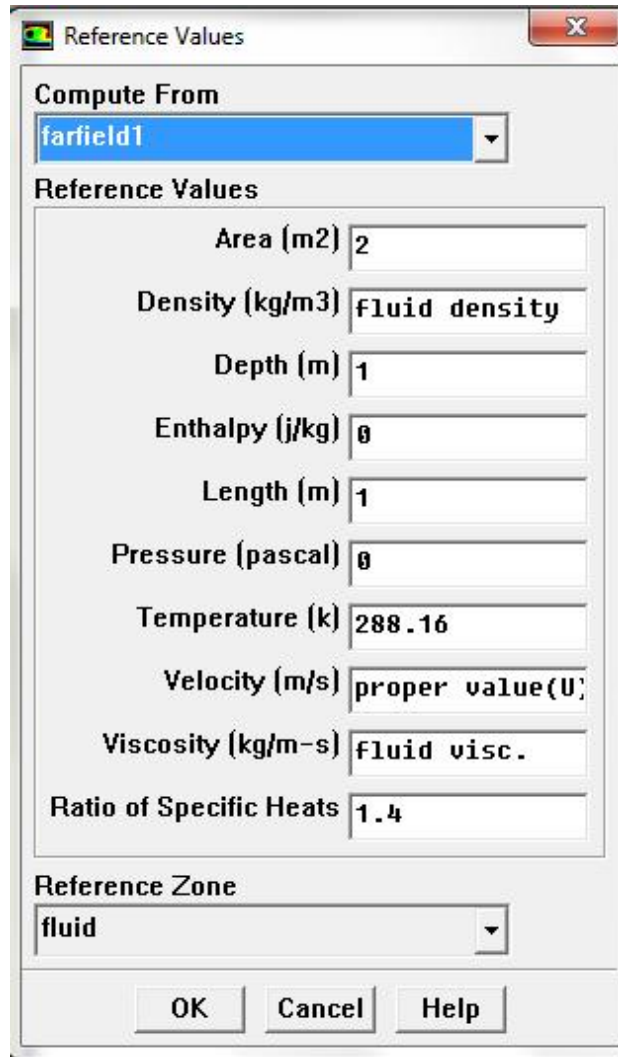


Under *Coefficient*, select *Lift*. Select *cylinder* under *Wall Zones*. Under *Options*, select *Print*, *Plot* and *Write*. Note that *Plot Window* is 1. Click *Apply*.

Set Reference Values

The reference values are used to non-dimensionalize the forces and moments action on the wall surface.

Main Menu > Report > Reference Values...

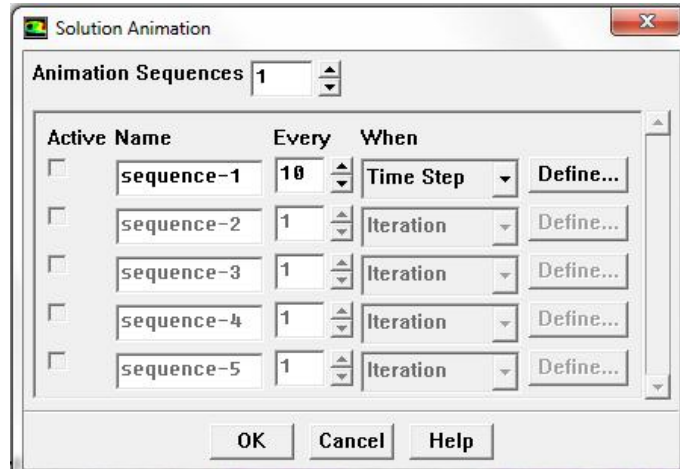


Select *farfield1* from the *Compute From* drop-down list. FLUENT will update the reference values based on the boundary conditions at *farfield1*. Change *Area* to 2. Click *OK*. Density and viscosity depend on material properties that define earlier.

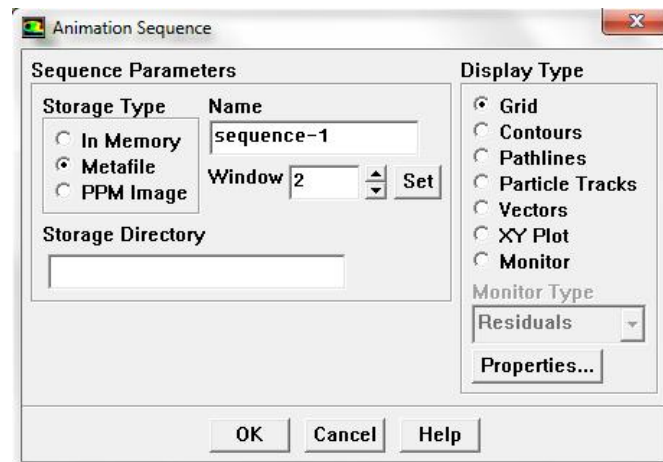
Set Animation Control (optional)

Let's set the animation to observe the vorticity magnitude.

Main Menu > Solve > Animate > Define...



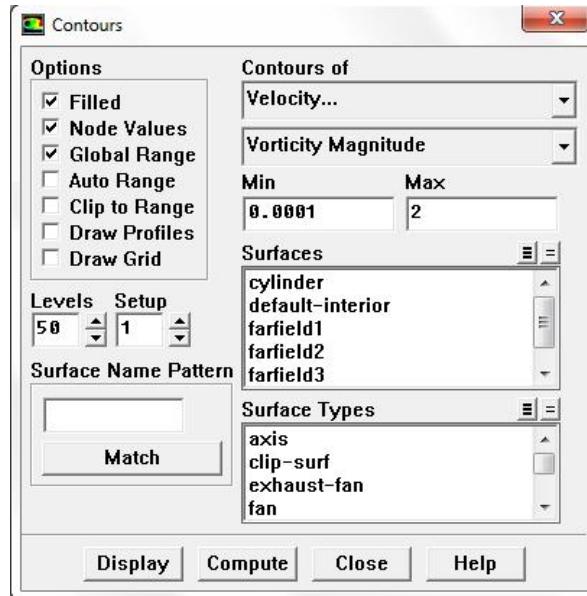
Increase the *Animation Sequences* to 1. Enter 10 for *Every*. Select *Time Step* from *When* drop-down list. Click *Define...* for *sequence-1* to open the Animation Sequence panel.



Increase *Window* to 2 and click the *Set* button to open a graphics window. Select *Contours* from the *Display Type* list to open the Contours panel. Select *Velocity...* and *Vorticity Magnitude* from the *Contours of* drop-down lists.

Disable *Auto Range* and *Clip to Range* from the *Options* group box. Enter 0.0001 and 2 for *Min* and *Max*, respectively. Select *Levels* to 50. Click *Display*. Click *OK* to close the *Animation Sequence* panel. Click *OK* to close the *Solution Animation* panel. This will save .hmf file after every 10 time steps. We can later create an animation in the form of movie clip using these files. Save the case and data files.

Main Menu > File > Write > Case & Data...



Iterate Solution

Main Menu > Solve > Iterate...

You will have to input the time step size for iteration. Smaller time step means more accurate result but more computational time. We need to find the balance between accuracy and computational time

Calculating Time Step Size

The Strouhal number for flow past cylinder is roughly 0.183 as reported by Williamson .

$$St = \frac{fD}{U} = 0.183 \rightarrow f = \frac{0.183U}{D}$$

In order to capture the shedding correctly, we should have at least 20 to 25 time steps in one shedding cycle. Let's use 25 for our case.

For our case, **D = 2**.

For example if **U=1 (m/s)** then

$$f = \frac{1}{t}$$

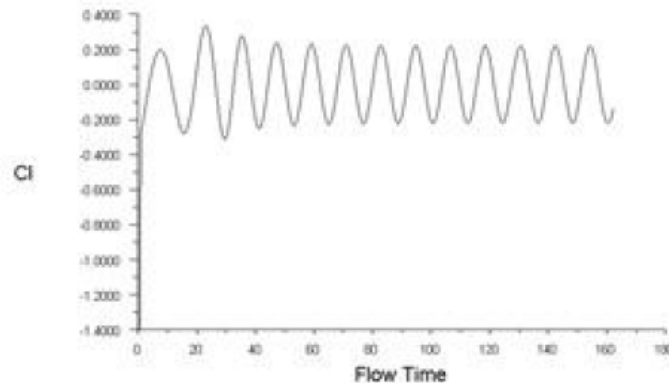
Therefore, shedding frequency is **f = 0.0915**

Cycle time:

$$t = \frac{1}{f} = 10.9$$

Therefore, **Time Step Size = 10.9/25 = 0.436 sec ~ 0.4 sec**

Enter 0.4 for *Time Step Size (s)*. Enter 30 for *Max. Iterations per Time Step*. Enter 800 for *Number of Time Steps*. Click *Apply*. Click *Iterate* to start the iterations.



We can see a clear sinusoidal pattern, a sign of sustained vortex shedding process after 40s. Stop the iteration after about 350s.

Save the case and solution.

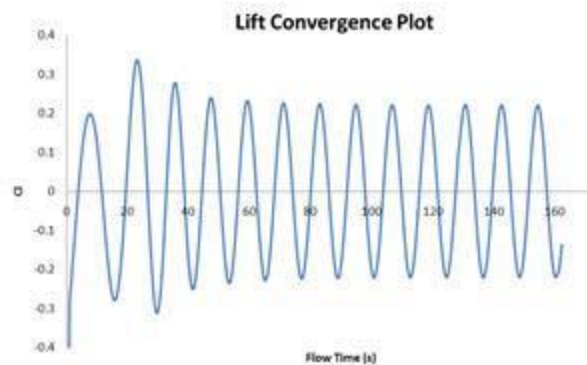
Main Menu > File > Write > Case & Data...

Use the default name (Mesh's file name "*cylinder*") and click *OK*.

Analyze Results

Calculate Shedding Frequency And Strouhal number

To accurately calculate the shedding frequency, open the cl-history file (saved previously in the same location where the original mesh was read) and plot the data using excel for better data representation and graphing option. Take an average of 10 shedding cycles (e.g 10 CL peak). An example of Lift Convergence Plot plotted using excel is shown below:



$$t = \frac{t_2 - t_1}{10}, f = \frac{1}{t}, St = \frac{fD}{U}$$

Reviewing Animation

We can review the animation created after we are done with iterations.

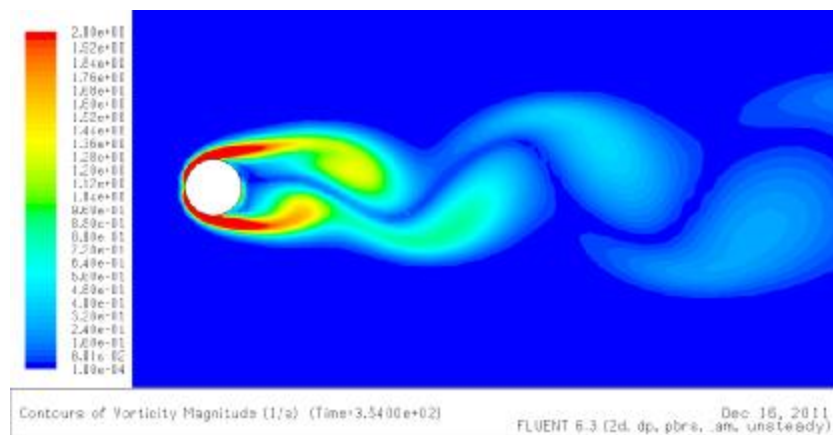
Solve > Animate > Playback...

To write in MPEG format, go to *Write/Record Format*, select *MPEG*. Click *Write*.

Note that you don't necessarily get a good format when exporting to MPEG. It is advisable to use the available playback option.

Display Contours Of Vorticity Magnitude

Under *Contours of*, choose *Velocity..* and *Vorticity Magnitude*. Disable *Auto Range* and *Clip to Range* from the *Options* group box. Enter 0.0001 and 2 for *Min* and *Max*, respectively. Select *Levels* to 50. Click *Display*.



This figure shows clear vortex shedding process. Zoom in the view around cylinder.

White Background on Graphics Window

To get white background go to:

Main Menu > File > Hardcopy

Make sure that *Reverse Foreground/Background* is checked and select *Color* in *Coloring* section. Click *Preview*. Click *No* when prompted "Reset graphics window?"