Lab Exercice on Computational Fluid Dynamics

# Unsteady flow past circular cylinder with ANSYS FLUENT

Instructor : Olivier Botella olivier.botella@univ-lorraine.fr

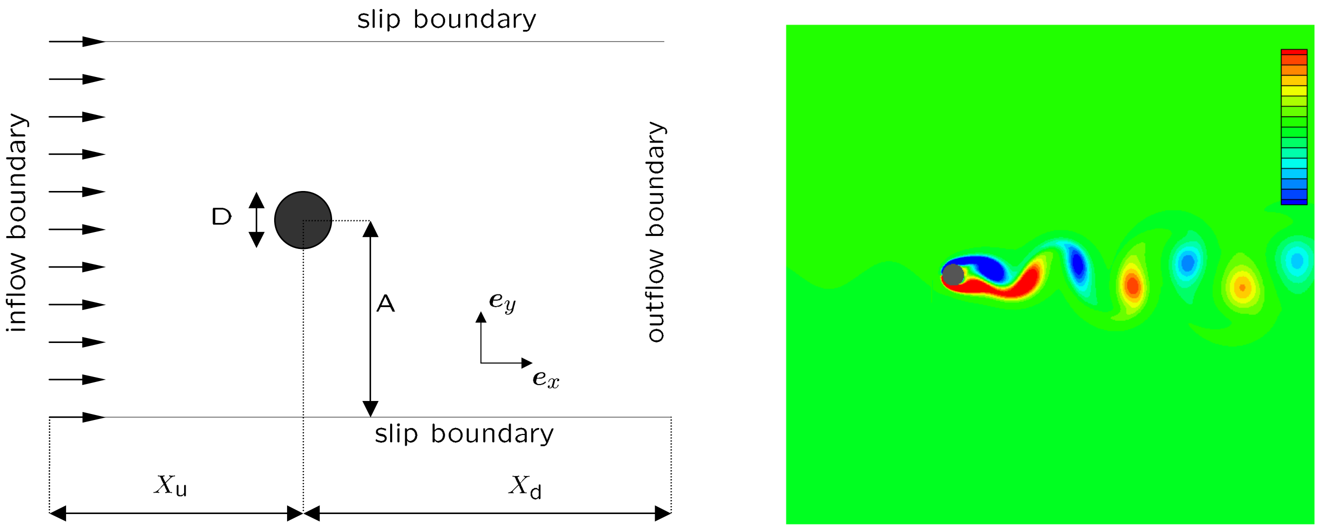


FIGURE 1 – *At left* : Computational domain and boundary conditions. The domain dimensions are : $D = 0.1$ m, $X_u = 0.8$ m, $X_d = 1.5$ m, $A = 0.8$ m. *At left* : instantaneous vorticity field $\omega = \partial v/\partial x - \partial u/\partial y$ at $Re = 100$.

## 1 Problem statement

This lab exercise concerns the CFD analysis of the laminar flow past a circular cylinder, see Fig. 1. The working fluid is *air*, and the *Reynolds* number of the flow is :

$$\mathrm{Re} = \frac{\rho U_\infty D}{\mu},$$

with the velocity at the inlet $U_\infty$ as reference velocity, and the cylinder diameter $D$ as reference length.

At low Reynolds number the flow is steady, and above the critical value $\mathrm{Re}_c \simeq 47$ the flow undergoes a Hopf bifurcation that causes unsteady oscillations of the wake : the well-known *von Kàrmàn vortex street* (*cf.* Fig. 1). In the periodic regime ($47 \lesssim \mathrm{Re} \lesssim 194$), the frequency $f$ of the vortex shedding is usually given as the dimensionless *Strouhal* number St :

$$\mathrm{St} = \frac{D}{U_\infty} f.$$

The Strouhal number is obtained by calculating the drag $F_x(t)$ and lift forces $F_y(t)$ exerted by the flow at the cylinder surface, and by performing a *FFT*[1] of the time-signal when the

1. FFT : Fast Fourier Transform.

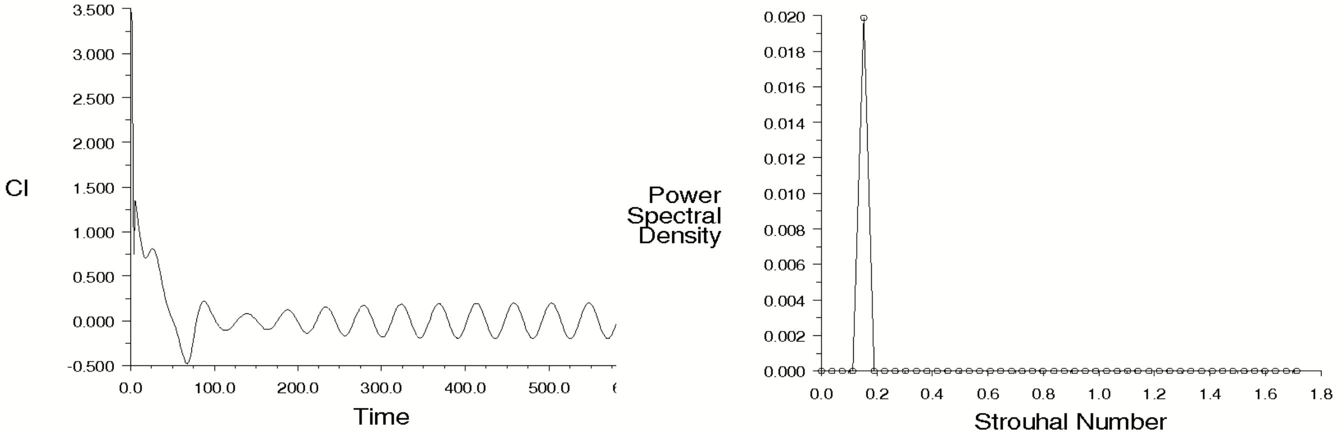


FIGURE 2 – *At left* : Time history of the lift coefficient $C_{\mathrm{L}}$ (the time is in seconds). *At right* : FFT of the lift signal in the time interval $t \in [400, 600]$, where the flow reached a periodic state.

flow has asymptotically reached its periodic regime, *cf.* Fig. 2. The hydrodynamic forces at the cylinder are usually given as the dimensionless *drag and lift coefficients* :

$$C_{\mathrm{D}}(t) = \frac{F_x(t)}{\frac{1}{2}\rho U_\infty^2 S}, \qquad C_{\mathrm{L}}(t) = \frac{F_y(t)}{\frac{1}{2}\rho U_\infty^2 S},$$

where $S$ ( $\mathrm{m}^2$ in SI units) is the projected area of the cylinder [2].

The purpose of this lab exercise is to compute the flow at Re = 100 with ANSYS Fluent , to study the vortex shedding and compare the values $C_{\mathrm{D}}$, $C_{\mathrm{L}}$ et St given by your computations with the reference values of Table 1.

| St | $\overline{C}_{\mathrm{D}}$ | $C_{\mathrm{L_{max}}}$ |
|---|---|---|
| 0.16 – 0.17 | 1.21 – 1.41 | 0.31 – 0.40 |

TABLE 1 – Experimental and computational reference data of the flow at Re = 100. $\overline{C}_{\mathrm{D}}$ denotes the mean value of the drag coefficient and $C_{\mathrm{L_{max}}}$ is the maximal value of the drag coefficient, calculated when the flow reached its periodic state.

## 2 Geometry creation with DesignModeler

The computational domain to create is shown in Fig. 1. It is important to center the domain with respect to its origin (the cylinder center). To create this domain, we consider that the fluid domain consist in a rectangle from which a circle is subtracted (with a boolean operation). For this reason, it is worthwhile to draw the rectangle and the circle in 2 different sketches.

**0)** Go to the Settings toolbox to apply the adequate parameters for sketching : Major Grid spacing : 0,1 m, Minor-steps per Major : 10. For ease of sketching, show the grid and toggle the Snap option so the drawings will be aligned to the grid lines.

2. The projected area $S$ is the frontal surface of the body projected on the spanwise direction. For 2D flow past circular cylinder of diameter $D$, we get $S = D$.

**1)** In a first sketch, create a rectangle with bottom-left corner of coordinates (-0.8 m, -0.8 m). It is then easy to center the rectangle with respect to the origin and give its correct dimensions. In a second sketch, create the circle [3].

**2)** Create the surface sketch from these 2 sketches with the Add Frozen option. Rename the first surface body as "fluid" and the second as "cylinder".

**3)** Now, you have to subtract the cylinder body to the fluid body by a boolean subtraction (Create → Boolean from the menu bar). Use the following details : Operation : Subtract, Target body : Fluid, Tool body : Cylinder, Preserve Tool Body : No.

After application of the boolean operation, you should have created 1 part, with 1 "fluid" body. Set its Fluid/solid detail to Fluid, save the project and exit DesignModeler .

## 3 Mesh creation with Ansys Meshing

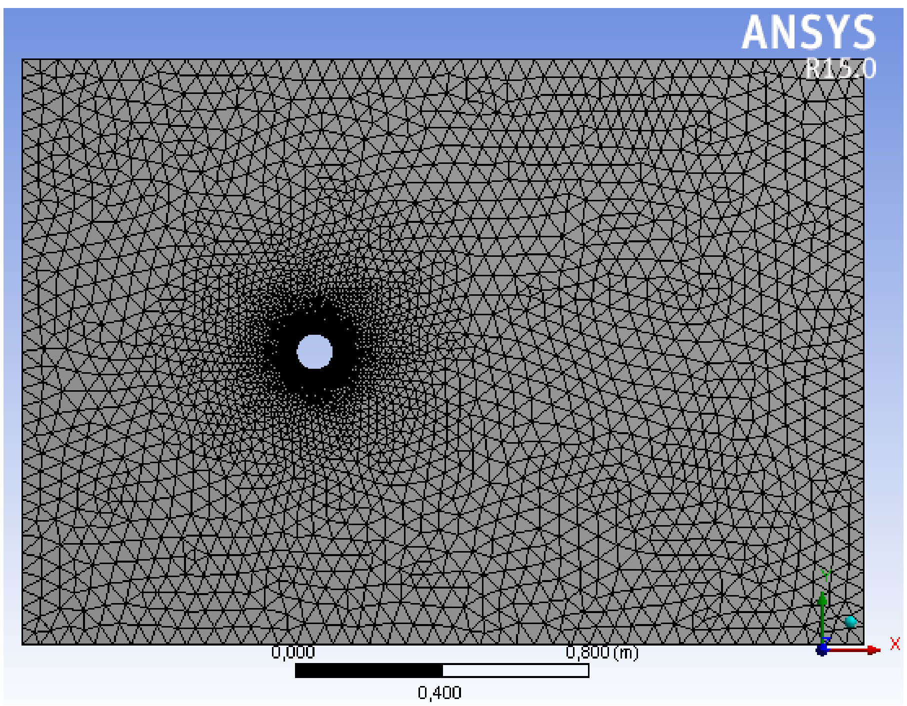


FIGURE 3 – Triangle mesh generated with the curvature size function and edge sizing of the cylinder surface. This mesh has 9715 cells.

The purpose of this part is to generate the unstructured triangle mesh shown in Fig. 3, which is refined around the cylinder surface in order to resolve the fine vortical structures in the boundary layers and the wake. To do so, Ansys Meshing use the following strategies :

– First, you define a Global Meshing Method in order to define the type of elements used (quad, tri, . . .) and the average size of the elements. In addition, DesignModeler use a "size function" to automatically control the mesh density in the vicinity of highly curved boundaries by using various strategies (proximity, curvature, . . ., see the CFD lectures). These global mesh parameters are set in the Details View of the mesh. By default, the size function is enabled (Use Advanced Size Function : On :Curvature).

3. If needed, use the Coincident constraint to put its center to the origin of the domain.

– Secondly, you may ameliorate the mesh in specific areas by inserting Local Meshing Controls such as edge or face sizing, inflation, *etc.*..(see CFD lectures). This step is optional if you only need a crude mesh, but is required to obtained quality meshes.

Following this strategy, the mesh is generated by the following steps :

**1)** Global Meshing controls : In the Tree Outline, select the mesh and insert a meshing method [4] for the fluid domain. In its Details View, choose Method : Triangles. Now, define the global meshing controls : in the Sizing Details of the mesh, choose Max Face Size : 0,05 m and Growth Rate : 1,10 for the size function.

**2)** Generate the mesh. You may observe that the size function has detected the cylinder surface as a highly curved boundary and has refined the mesh in its vicinity [5]. However, the mesh is not as fine near the boundary as the one in Fig. 3.

**3)** Add Local Meshing controls : insert an Edge Sizing method in order to control the cell distribution on the cylinder surface with Number of Divisions : 200. The mesh generated will be similar to the one in Fig. 3.

**Question 1.** *Display a figure of the various meshes you created in your* Word *report. The final mesh, to be used for the CFD analysis, should have the boundary conditions of Fig. 1 labeled with* Named Selections *before exiting* Ansys Meshing *.*

# 4 Flow computation with Fluent

The unstructured mesh generated with DesignModeler is now used to compute the flow $\mathrm{Re} = 100$ (*reminder* : the working fluid is **air**).

## 4.1 Step 1 : Solution setup

In this preliminary stage, you will define the flow physics (flow equations, boundary conditions, ...), set up the solver and the discretization methods. Follow the guidelines given in the first lab exercise. Only new steps will be detailed below.

### Guidelines for solution setup

- Since the $\mathrm{Re} = 100$ flow is time-periodic, use the Unsteady solver. The PISO algorithm for pressure-velocity coupling is recommended.
- For setting the Inflow BC, reply to the following question :

**Question 2.** *Calculate the inflow velocity $U_\infty$ corresponding the the $Re = 100$ regime.*

- Choose the most accurate and robust discretization methods, according to your experience with the first lab exercise.
- Select 2nd-order accurate time-stepping [6] otherwise your computation may blow up.

---

4. Alternatively, toggle the volume selection icon , right-click Insert → Method on the fluid domain.
5. You may also generate a mesh with Use Advanced Size Function : Off and observe the differences.
6. In the Solution Method Task Page, choose Second Order Implicit in the Transient Formulation drop-down list.

### Set up the time history of the *drag* $C_{\mathrm{D}}$ and *lift* $C_{\mathrm{L}}$

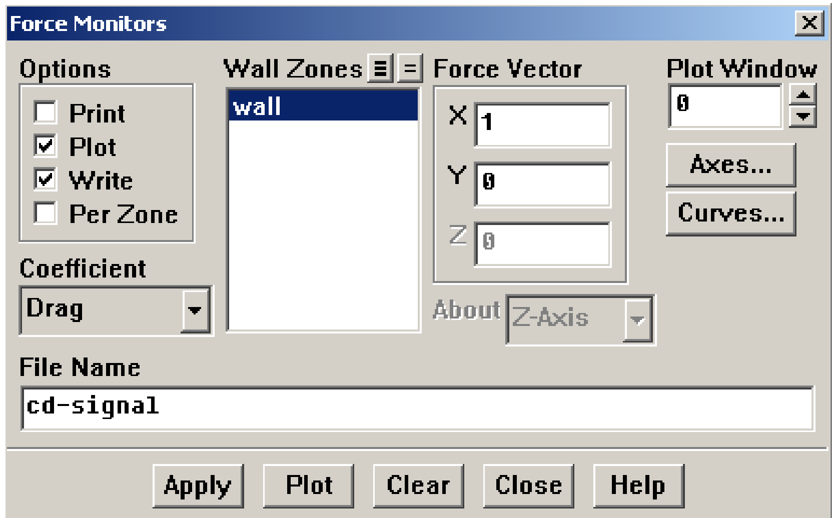


FIGURE 4 – Configuring the Drag Monitor dialog box. The drag is the horizontal component (Force Vector : $\mathbf{e}_x$) of the hydrodynamic force acting on the cylinder surface (Wall Zones). The Plot option will plot the signal in a graphical window during the computation, while the Write option will write the data in the text file cd-signal.

During the unsteady computation, Fluent is able to calculate on-the-fly the forces coefficients $C_{\mathrm{L}}(t)$ et $C_{\mathrm{D}}(t)$ in order to postprocess these results as in Fig. 2. Prepare these calculations with the following steps :

- Define Reference Values of the flow (Solution Setup panel). The projected area $S$ is denoted Area. These references values will be used by Fluent to calculate dimensionless flow quantities such as force coefficients.
- Set up the drag calculation with the Monitors task panel. Click on [Create] → Drag... and follow the instructions of the dialog box of Fig. 4. Do the same for the lift coefficient.

### Setup the initial condition

In order to break the flow symmetry and trigger the Hopf instability, you will use the initial condition :

$$\boldsymbol{v}(\boldsymbol{x}, t=0) = \boldsymbol{v}_0 + \boldsymbol{v}_{\text{pert}}(\boldsymbol{x}), \tag{1}$$

where $\boldsymbol{v}_0 = \mathbf{0}$ (fluid is initially at rest) and the perturbation $\boldsymbol{v}_{\text{pert}} = u_{\text{pert}}(y)\boldsymbol{e}_x$ such as :

$$u_{\text{pert}}(y) = U_\infty \frac{y + |y|}{2y}. \tag{2}$$

In this fashion, the initial condition correspond the sheared velocity :

$$u(y, t=0) = \begin{cases} U_\infty & \text{if } y > 0 \text{ (upper part of the domain),} \\ 0 & \text{otherwise.} \end{cases} \tag{3}$$

This initial condition is defined in Fluent with the following steps :

- Define analytically $u_{\text{pert}}(y)$ with the Custom Field Functions calculator [7].

7. Note that the $y$ coordinate is given in the Field Functions drop-down list [Mesh...]

- The initial condition is now defined with the Solution Initialization task panel. Define $\boldsymbol{v}_0 = \mathbf{0}$ in the fluid domain as in the first lab exercise. Then, add the perturbation $u_{\text{pert}}(y)$ by "patching" it [8] to $\boldsymbol{v}_0$.

**Question 3.** *Define this initial condition with* Fluent *. Plot the velocity field in the whole computational domain and on a vertical section. Displays these plots in your* Word *report.*

## 4.2 Step 2 : Flow computation

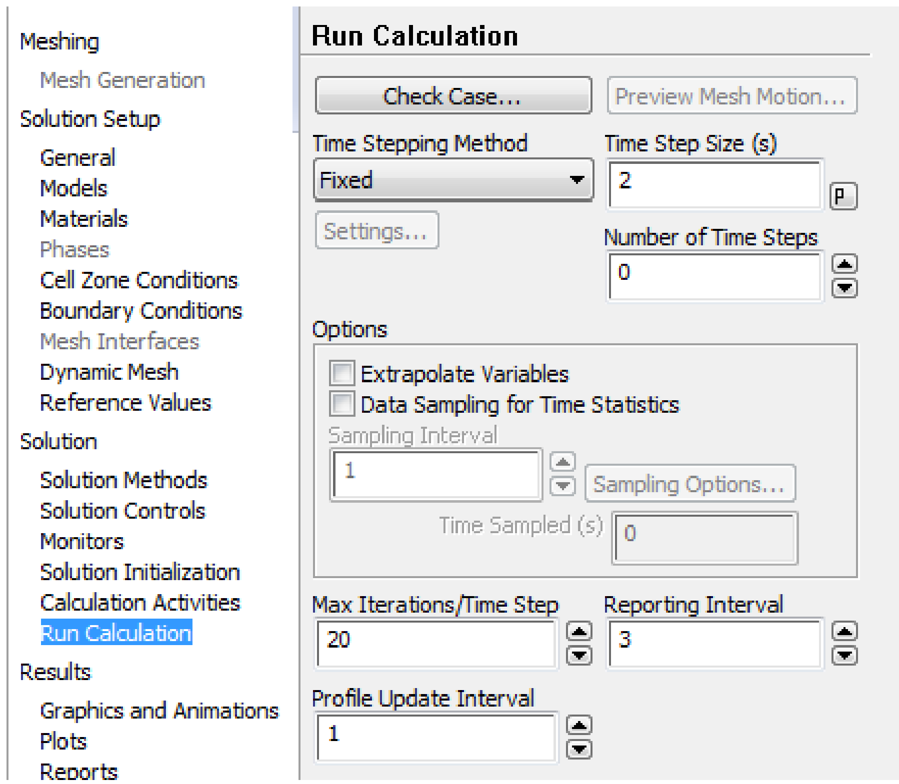


FIGURE 5 – Run calculation task page for the unsteady solver. The Fixed option uses a non-varying value of $\Delta t$ (Time Step Size), necessary to perform a FFT of the time-signals. Max Iterations/Time Step defines the number of inner iterations of the velocity-pressure algorithm algorithm. Reporting Interval is the monitoring frequency of the algorithm.

The task page for the unsteady solver is shown in Fig. 5. You need to define the size of the time-step $\Delta t$ (in seconds) and the number of time-steps. These values should be chosen from numerical considerations (such as the CFL condition be respected for stable computations), and also from the timescales of the flow at $\text{Re} = 100$ , as done below.

8. Click on Patch... to open the patch dialog box. Choose Variable : X-Velocity and Zones to Patch : fluid, then click on Patch.

### Determine the time-step size

**Question 4.** *Use the value of the Strouhal number (Table 1) to calculate the period $T_{shed}$ (in seconds) of the vortex shedding. Deduce the value of $\Delta t$ that allows about 20 time-steps during one vortex-shedding period.*

### Determine the number of time-steps

**Question 5.** *From the value of $\Delta t$ and the relation $t_{max} = n_{max}\Delta t$ (in seconds), calculate the number of time-steps $n_{max}$ for computing 600 seconds of the flow.*

### Compute the flow

**Question 6.** *Click on Calculate to compute the flow. Check that the instability is triggered by monitoring the time evolution of the drag and lift coefficients. Export these plots to your* Word *reports.*

Most Fluent solvers for incompressible flows [9] are fully implicit, so that stable computations are obtained even with a CFL value greater than 1. Nonetheless, unstable computation may occur due to poor mesh quality, inadequate boundary conditions or stiff initial conditions. If solution blow-up happens in this lab exercise, review first your solution setup. If the computation is still unstable, restart the computation with a lower value of $\Delta t$ for the first few time-steps, then increase $\Delta t$ to its original value for the rest of the computation.

It is advised to perform an autosave of the solution during time-integration, see first lab exercise.

## 4.3 Step 3 : Postprocessing

Now that the computation has been completed, this part is devoted to study the flow physics.

9. With the notable exception of Non Iterative Time Advancement and Projection Method.

## Solution visualization

> **Question 7.** *Export to your* Word *report the drag and lift history plots. Calculate* $\overline{C}_D$ *and* $C_{L_{max}}$ *when the flow reached the periodic regime, and compare your results with those Table1.*

> **Question 8.** *Display the iso-vorticity lines (see Fig. 1) at a selected flow-time. In order to clearly visualize the cylinder wake, carefully choose an odd number of levels equally distributed around the zero isolevel (*$\mathrm{Min} = -0.5$ *and* $\mathrm{Max} = -0.5$*, for example).*

Unfortunately, only the norm of the vorticity, named Vorticity Magnitude[10], is defined in Fluent . You need to define the vorticity $\omega = \partial v/\partial x - \partial u/\partial y$ as a Custom Field Function in order to plot it.

## Calculation of the Strouhal number

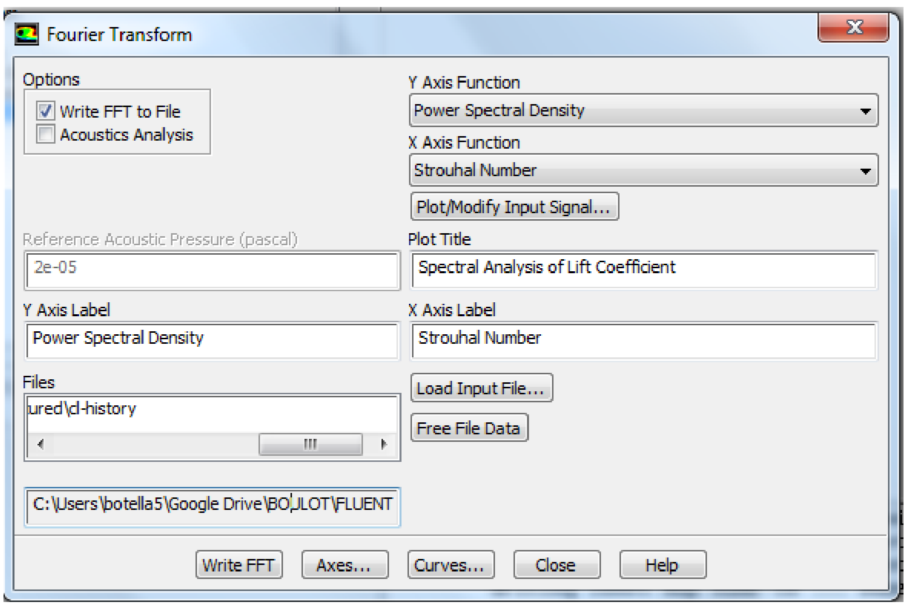


FIGURE 6 – Fourier Transform dialog box for FFT analysis of the lift coefficient.

The frequency of the vortex shedding can be estimated from the time history of the lift signal, *cf.* Question 7. However, it is customary to use signal processing tools such as the Fourier transform to precisely determine the flow regime (periodic, quasiperiodic, chaotic, ...). From the Plots task page, click on FFT to open the diaplog box of Fig. 6, and follow these steps :

- Load the lift history file cl-signal.
- Use Plot/Modify Input Signal... to choose a time interval[11] where the periodic regime is reached, for example $t \in [400\,\mathrm{s},\ 600\,\mathrm{s}]$.

10. In the Contours dialog box of the Graphics and Animation panel, Vorticity Magnitude is available in the Contours of Velocity... drop-down list.

11. We recall that the accuracy of frequency calculation with FFT is inversely proportional to the length of

- Click on Plot FFT

**Question 9.** *Export the plot of the FFT of the lift signal in your* Word *report. Calculate the frequency of the vortex shedding and give the incertitude of your calculation. Compare your results with the reference (Table 1).*

### Graphical animation of the vortex shedding

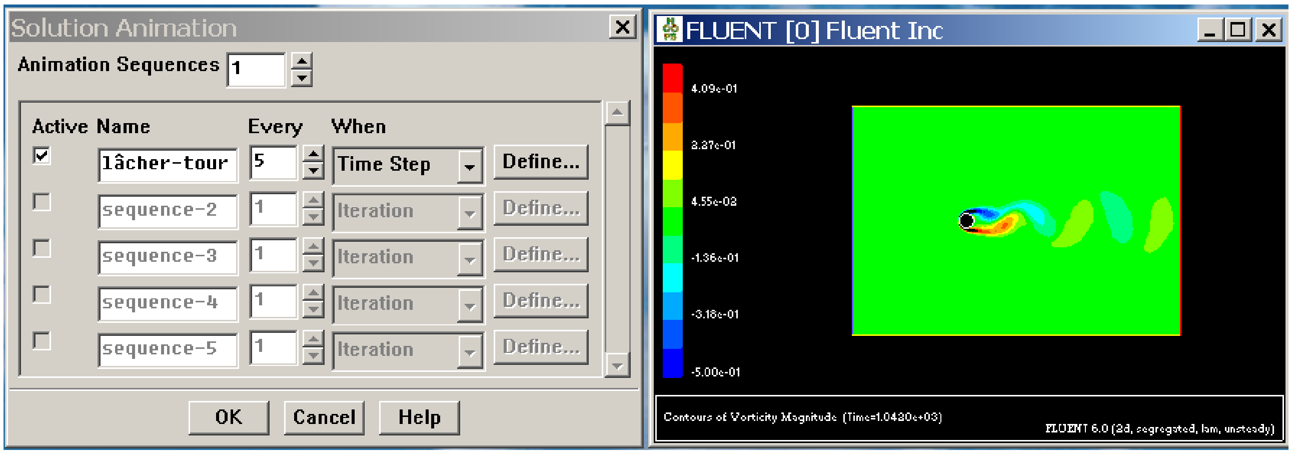


FIGURE 7 – *At left* : Solution Animation dialog box. *At right* : Iso-vorticity contours to be animated.

This part is devoted to the animation of the computed solution for visualization of the vortex shedding. This is done by performing a simulation on a few periods (for example $5\,T_{\text{shedd}}$ seconds) and periodically saving snapshots of the iso-vorticity lines on the disk (for example every 5 time-steps). The few dozen of snapshots can then be animated in a video-clip.

In the Calculation Activities task panel, open the Solution Animation dialog box of Fig. 7, and follow these steps :

- Activate an animation and label it. With the option Define... enter the iso-vorticity lines of Question 8.
- Resume the flow computation for $5\,T_{\text{shedd}}$ seconds to generate the snapshots.
- In the Graphics and Animation task page, use Solution Animation Playback to animate the iso-vorticity lines, and save the results as a video-clip.

**Question 10.** *Generate a video-clip of the vortex shedding, and insert it in your* Word *report.*

the time interval (the incertitude is $\pm\frac{1}{2}\frac{1}{\Delta t_{FFT}}$ s$^{-1}$ for an interval of length $\Delta t_{\text{FFT}}$ s). So choose a large time interval for your FFT to have accurate results !