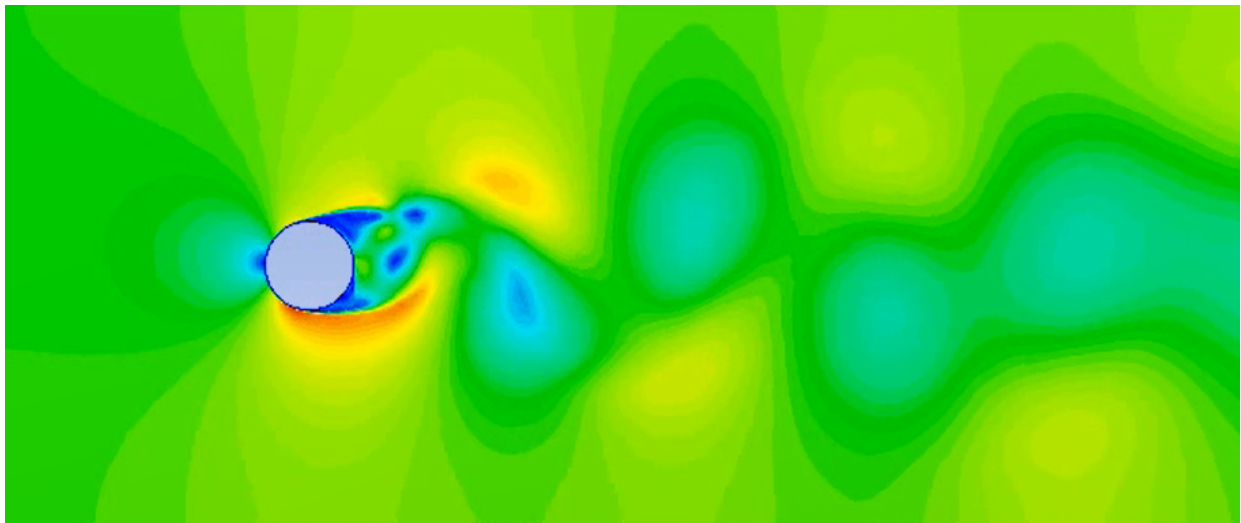


Numerical study of a flow around a cylinder



Students :

Cyrielle DESOLLE

Zoé CORLAY

Amélie PRUDHOMME

Chenghan YANG

Project's coordinating teacher :

Muhittin CELEP

Date of report's submission : *11/06/2022*

Reference of the project : *STPI/P6/2022 – 40*

Title of the project : *Numerical study of a flow around a cylinder*

Type of project : *Simulation*

Goals of the project :

- Discover and deepen our knowledge of fluid mechanics*
- Discover and become familiar with the different methods of numerical simulation and the simulation software ANSYS Fluent*
- Carry out numerical simulations of a flow around a cylinder*

Keywords of the project : *Fluid flow, numerical simulation, CFD software.*

TABLE OF CONTENTS

Acknowledgments	3
Notations, Acronyms	3
Introduction	4
Description of our Work	5
Work and Results	6
General and mathematical description	6
The physical phenomenon	6
The boundary layer	6
The Reynolds number	6
The Navier-Stokes equations	7
Simplification of the equations	7
The RANS (Reynolds Averaged Navier-Stokes) equations	7
The LES (Large Eddy Simulation)	8
The DNS (Direct Numerical Simulation)	9
The modelisation of the turbulences	10
The k- ϵ model	10
The k- ω model	11
The laminar model	11
Fluent resolution (Pre-processing)	11
Fluent	11
Generation of the geometry	11
Generation of the mesh	12
Choice of boundary conditions and initial conditions	13
Specification of numerical parameters	14
Results and discussion	15
Re=1 (Laminar steady flow):	16
Re=40 (Pair of vortices, laminar steady flow):	16
Re=140 (Laminar vortex street, unsteady flow):	17
Re=20000 (Transitional unsteady flow):	18
Conclusion and perspectives	20
Bibliography	21
Table of illustrations	22
Annexes	23
Annexe 1: Drag and lift coefficient for Re=1	23
Annexe 2: Drag and lift coefficient for Re=40	24
Annexe 3: Drag and lift coefficient for Re=140	25
Annexe 4: Drag and lift coefficient for Re=20000	26

ACKNOWLEDGMENTS

First of all, we would like to thank Mr. Muhittin CELEP, our project's coordinating teacher who allowed us to carry out our project. He knew how to introduce fluid mechanics at our level, he guided us in our research and knew how to answer our expectations. He listened to us and was available for our various questions. We thank him for his commitment and the time he gave us. We would also like to thank the INSA, for having offered us the possibility to realize this project. Finally, we would like to thank each member of this group for their investment, without which the project would not have been possible.

NOTATIONS, ACRONYMS

CFD : Computational fluid dynamics

D : Diameter of the cylinder (m)

E : Rate of deformation

\vec{g} : Gravitational vector

p : Pressure (Pa)

\bar{p} : Filtered pressure field

Re : Reynolds number (*no unit*)

\bar{S}_{ij} : Deformations tensor

\bar{T}_{ij} : Filtered Reynolds tensor

t : Time (s)

u : Flow velocity ($m \cdot s^{-1}$)

\vec{u} : Velocity field

Δ : Size of the filter (m)

μ : Dynamic viscosity of the fluid ($Pa \cdot s$)

μ_t : Eddy viscosity ($Pa \cdot s$)

ν : Kinematic viscosity of the fluid ($m^2 \cdot s^{-1}$)

ρ : Density of the fluid (*no unit*)

∇ : Divergence

∇^2 : Laplacian

1. INTRODUCTION

What is CFD?

CFD can be defined as the approximation of the governing partial differential equations of fluid flow with the discretization method.

How is CFD useful?

There are three different approaches to getting the results we want:

The numerical approach can solve difficult problems such as non-linear problems, and precise information of the flow but sometimes high computing time. A theoretical approach is limited in particular for industry because there are a lot of hypotheses. The experimental approach is realistic, and can solve difficult problems but is very expensive (20 000€/day) and has technical difficulties. With the advent of the high-speed digital computer, CFD will remain the third dimension in fluid dynamics, of equal stature and importance to experiment and theory. There is a rapid decrease in the cost of computations compared to the rapid increase in the cost of wind tunnel tests. So in most cases, CFD is cheaper. CFD allows us to make easy movements that are difficult in real life (move the floor and the air). This allows us to obtain the characteristics of the fluid throughout the simulation (temperature, density...).

Limitations of CFD

CFD has several limitations:

- It is difficult to have accurate solutions to Navier-Stokes equations for high Reynolds numbers.
- It is complicated to judge the accuracy due to the approximations of the calculations.
- A compromise between time and cost/accuracy must be done. Indeed, if we use very fine meshes to have a more precise result, in this case there will be more calculations to do and therefore a very long simulation time.
- It can't reproduce physics that is not properly included in the formulation of the problem (turbulence).

However, we can visualize our result which allows us to see if it is coherent and to critically analyze it before believing it.

Hypothesis

The hypothesis that we have on our fluid for the simulation is that it is incompressible, monophasic and the temperature is constant.

Aim

Our objective is to carry out simulations of a flow field around a cylinder using the ANSYS Fluent software by understanding each step and what is obtained.

2. DESCRIPTION OF OUR WORK

Description of the organization adopted for the conduct of the work:

When we started the project, we noticed that it was important to combine individual research work with teamwork. We, therefore, shared the tasks to be carried out in order to work efficiently. In addition to weekly meetings dedicated to the project, we regularly discussed the progress of the project in face-to-face or remote sessions. We also created a shared folder where we shared our work every week. During the P6 meetings, we all presented the previous week's work and research together.

Flow chart of tasks performed and students involved:

Member of the group	Superficial research	Books	Extensive research	Simulations	Report	Poster	Presentation
Amélie	Numerical simulation	2	CFD		Everyone participated in every part	X	Everyone made his part
Chenghan	Computation of fluid dynamics	1	Physical phenomenon				
Cyrielle		1	Fluent	X			
Zoé	Numerical simulation	2	RANS-LES-DNS				
Weeks	1-3	4-6	7-10	9-13	9-13	13-14	14

Table 1: Distribution of work by week (1,2: cf [bibliography](#))

3. WORK AND RESULTS

3.1. General and mathematical description

3.1.1. The physical phenomenon

3.1.1.1. The boundary layer

The boundary layer is the region in which flow adjusts from zero velocity at the wall to a maximum in the mainstream of the flow. Outside the boundary layer, the speed of the fluid is closer to a fixed value.

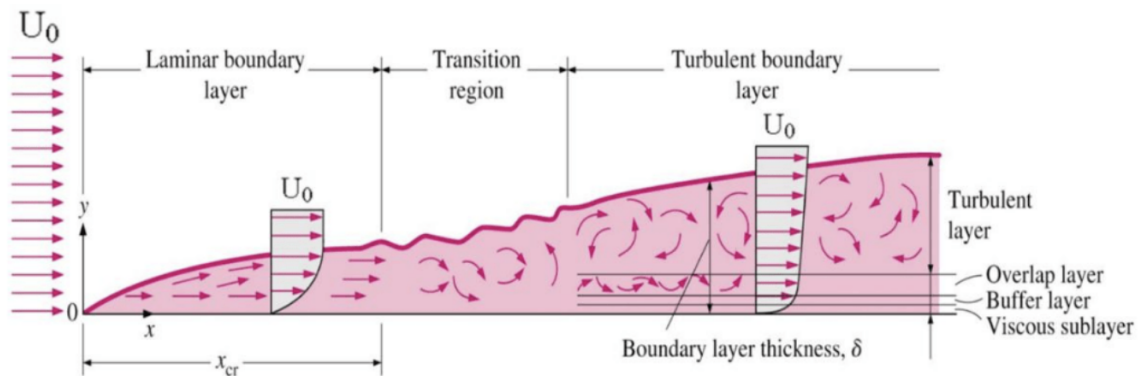


Figure 1: Boundary layer pattern

3.1.1.2. The Reynolds number

The Reynolds number is a dimensionless number used in fluid mechanics. It characterizes a flow, and helps predict flow patterns in different fluid flow situations. It is the ratio of inertia forces to viscous forces and is a convenient parameter for predicting if a flow condition will be laminar or turbulent. At low Reynolds numbers, viscosity dominates the inertia forces so flows tend to be laminar around the body of examination, whereas at high Reynolds numbers inertia forces dominate the viscosity and flows tend to be turbulent.

eq 1. $Re = \frac{uD}{\nu} = \frac{\rho uD}{\mu}$

	$Re < 5$: No separation, laminar steady flow		$150 < Re < 3.10^5$: Transitional unsteady flow
	$5 < Re < 45$: Pair of vortices, laminar steady flow		$3.10^5 < Re < 3.10^6$: Turbulent unsteady flow
	$45 < Re < 150$: Laminar vortex street, unsteady flow		$Re > 3.10^6$: Turbulent vortex street, unsteady flow

Table 2 : Flow profiles depending on Reynolds numbers

3.1.1.3. The Navier-Stokes equations

The Navier-Stokes equations are a set of partial differential equations used to describe the flow of fluids. In those equations, velocity, pressure, temperature, density, and viscosity are the main properties that should be considered simultaneously when conducting the fluid flow examination.

The expression consists in the continuity equation : $\vec{u} \cdot \nabla = 0$ **eq 2.** and the momentum equations :

$$\rho \frac{d\vec{u}}{dt} = - \nabla p + \rho \vec{g} + \mu \nabla^2 \vec{u} \quad \text{eq 3.}$$

with the acceleration $\frac{d\vec{u}}{dt} = \frac{\partial \vec{u}}{\partial t} + (\vec{u} \cdot \nabla) \vec{u}$ **eq 4.** in which : $\frac{\partial \vec{u}}{\partial t}$ is the change in velocity over time $(\vec{u} \cdot \nabla) \vec{u}$ is the speed and the direction of the fluid and with ∇p the pressure gradient, $\rho \vec{g}$ the body force term which expresses external forces, $\mu \nabla^2 \vec{u}$ the diffusion term which expresses internal stress forces.

All the governing equations are based on the laws of conservation. The Navier- Stokes equations are applied to a mathematical model to examine changes in those properties during dynamic or thermal interactions. The equations are adjustable regarding the content of the problem and are expressed based on the principles of conservation of mass, momentum, and energy.

3.1.2. Simplification of the equations

In order to solve the Navier-Stokes equations for our flow, we will need to find the best simplifications of its terms. There are numerous possibilities, but it is important to choose approximations depending on the case that we study, in order to keep the most precise ones : we would like to be able to make the calculations and have the best results.

3.1.2.1. The RANS (Reynolds Averaged Navier-Stokes) equations

It's one of our most powerful tools for modeling complex fluid flows. In the following, we're going to start with writing down the Navier Stokes equations. We're going to decompose the flow field into an average and a fluctuating component. There are two conclusions about Reynolds averaged Navier Stokes equations.

mass conservation : $\vec{u} \cdot \nabla = 0$ **eq 2.**

momentum conservation : $\vec{u}_t + (\vec{u} \cdot \nabla) \cdot \vec{u} = - p + \frac{1}{Re} \nabla^2 \vec{u}$ **eq 5.**

Essentially what we're going to do is we're going to take our flow field variable u which is about space and time, then we're going to decompose it into an average component and a fluctuating component : $u(x, t) = \bar{U}(x) + u'(x, t)$ And it is an important point that \vec{u} is a vector variable, but u is not. We use this type of decomposition because it turns out that this is a really good idea for modeling this partial differential equation with fewer degrees of freedom than we would normally need to model on a computer.

And the following is the average velocity field at each point in space. $\bar{U}(x) = \lim_{T \rightarrow \infty} \frac{1}{T} \int_0^T u(x, t) dt$

Thus, from the mass conservation, we can get : $\frac{\partial \bar{U}}{\partial x} + \frac{\partial \bar{V}}{\partial y} + \frac{\partial \bar{W}}{\partial z} + \frac{\partial u'}{\partial x} + \frac{\partial v'}{\partial y} + \frac{\partial w'}{\partial z} = 0$

$$\nabla \cdot \vec{U} = 0 \text{ eq 6.}$$

$$\nabla \cdot \vec{u}' = 0 \text{ eq 7.}$$

From the mass conservation, we can get : $u'_t + \left(u' \frac{\partial}{\partial x} + v' \frac{\partial}{\partial y} + w' \frac{\partial}{\partial z} \right) u' = - \frac{\partial p}{\partial x} + \frac{1}{Re} \nabla^2 u' \text{ eq 8.}$

$$\begin{aligned} & \bar{U}_t + u'_t + \bar{U} \bar{U}_x + \bar{U} u'_x + u' \bar{U}_x + u' u'_x + \bar{V} \bar{U}_y + \bar{V} u'_y + v' \bar{U}_y + v' u'_y + \bar{W} \bar{U}_z + \bar{W} u'_z + w' \bar{U}_z \\ & + w' u'_z = - \bar{P}_x - P'_x + \frac{1}{Re} \nabla^2 \bar{U} + \frac{1}{Re} \nabla^2 u' \end{aligned}$$

After eliminating : $\bar{U} \bar{U}_x + \bar{V} \bar{U}_y + \bar{W} \bar{U}_z + u' u'_x + v' u'_y + w' u'_z = - \bar{P}_x + \frac{1}{Re} \nabla^2 \bar{u}$

By definition, $\overline{u' u'_x} = \frac{\partial \overline{u' u'_x}}{\partial x} - \overline{u' \frac{\partial u'}{\partial x}}$ the same as other two directions, we can get :

$$\bar{U} \bar{U}_x + \bar{V} \bar{U}_y + \bar{W} \bar{U}_z + \frac{\partial \overline{u' u'_x}}{\partial x} + \frac{\partial \overline{v' u'_y}}{\partial y} + \frac{\partial \overline{w' u'_z}}{\partial z} = - \bar{P}_x + \frac{1}{Re} \nabla^2 \bar{u} \text{ eq 9.}$$

We call the sum of the three partial differentials Reynolds stresses, which is the component of the total stress tensor in a fluid.

3.1.2.2. The LES (Large Eddy Simulation)

This approach was proposed by Smagorinsky in 1963. The principle of this model is as follows: by applying a "low-pass filter" to the Navier-Stokes equations, we eliminate the smallest length scales, thus leaving us with a simplified and faster system of equations to solve. This filter can be applied to the space and time domains with the expression :

$$\overline{\phi(x, t)} = \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} \phi(r, \tau) G(x - r, t - \tau) d\tau dr \text{ eq 10.}$$

We first consider the Navier-Stokes equations in the case of an incompressible fluid. By applying the LES method to these equations, we obtain:

The conservation of mass equation: $\frac{\partial \bar{u}_i}{\partial x_i} = 0 \text{ eq 11.}$

The incompressible Navier-Stokes equation $\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j} (\overline{u_i u_j}) = - \frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} (2\nu S_{ij} + \overline{\tau_{ij}}) \text{ eq 12.}$

For the Navier-Stokes equations in the case of a compressible fluid, each equation is filtered with a Favre filtering (density-weighted filtering). We give here the filtered equation of the conservation of mass :

$$\frac{\partial \bar{p}}{\partial t} + \frac{\partial \rho \bar{u}_i}{\partial x_i} = 0 \quad \text{eq 13.}$$

The complexity of this approach lies in the fact that the field is nonlinear: it is, therefore, necessary to know the unfiltered velocity field. Therefore, this velocity field must be modeled separately, which is an expensive and complex task. In conclusion, we have seen that the LES method only considers a range from the size of the field L to the size of the filter Δ : therefore the accuracy of the results given by this method depends on the quality of the initial mesh and the value of the parameter Δ . The most important thing is that the boundary conditions should be chosen very carefully in order to obtain the best results.

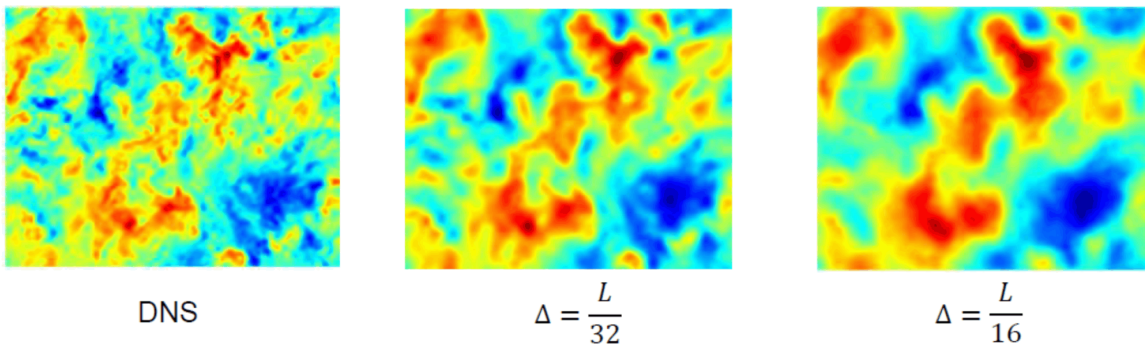


Figure 2: Simulation results depending on the approximations

3.1.2.3. The DNS (Direct Numerical Simulation)

First of all, the DNS method is not based on any turbulence model. Therefore, it has to solve numerically the Kolmogorov scale as well as the Navier-Stokes equations, which leads to rather complex calculations involving many variables and very small scales. The DNS method is the most accurate of the three methods and provides a great deal of detail about the system. The feasibility of this method lies in the fact that the minimum scale of turbulence is not infinite and thus can be modeled. Therefore, the DNS simulation is accurate enough to model these small scales in time and space. It is based on the relationship between the Reynolds number and the Kolmogorov scale:

$$\frac{D}{\eta} \approx Re^{3/4} \quad \text{eq 14. with } \eta = \left(\frac{\nu}{\varepsilon}\right)^{1/4}.$$

In such a simulation, the storage requirements increase rapidly with the Reynolds number. For most industrial applications, this method is too resource-intensive to use. However, for some low Reynolds numbers, this method does work and the simulated flow characteristics are very close to reality.

	RANS	DNS	LES
advantage	-Low computational cost, allowing average statistics	-No need to use models. -Accuracy is limited only	-Only small scales need to be modeled. as these small-scale

	to be obtained in a very short period of time -Can be used for extremely complex geometries.	by computational power. -All fluid characteristics (in time and space) are known.	models are most likely to be insensitive to the specifics of the flow. -Much cheaper than RANS and DNS -Allows to simulate fluids with Reynolds number tending to infinity. -Can obtain as much accuracy as possible by refining our numerical grid.
disadvantage	-Only steady-state flow phenomena can be fully explored with reduced computational effort. -Often, pragmatic "tuning" of the model parameters is required for specific applications. -Most exact turbulence models lead to very complex sets of equations.	-The method is limited to fluids with low Reynolds numbers and simple geometry. -Very time-consuming and requires a large memory capacity. -Usually, very fine meshes are used without proper convergence evaluation.	- The underlying assumption requires that these small (unresolved) scales are independent of the boundary conditions -For complex geometries and highly anisotropic flows, it is hard to solve.

Table 3 : Comparison between the approximation methods

3.1.3. The modelisation of the turbulences

With the Fluent software, in RANS applications, it is possible to model turbulence in different ways with additional transport equations in order to close the system of equations of the problem to solve.

3.1.3.1. The k-ε model

The k-ε turbulence model is the most used model in CFD. The k-epsilon model is a semi-empirical model based on the Boussinesq concept relating Reynolds stresses to average strain rate. It is a two-equation model that gives a general description of turbulence by means of two transport equations. k is the turbulent kinetic energy and ε is the rate of dissipation of turbulent kinetic energy. The two equations are :

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right] + 2\mu_t E_{ij} E_{ij} - \rho \epsilon \text{ for } k \quad \text{eq 15.}$$

$$\text{and } \frac{\partial(\rho \epsilon)}{\partial t} + \frac{\partial(\rho k \epsilon)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\frac{\mu_t}{\sigma_\epsilon} \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} 2\mu_t E_{ij} E_{ij} - C_{2\epsilon} \rho \frac{\epsilon^2}{k} \text{ for } \epsilon \quad \text{eq 16.}$$

$$\text{with } \mu_t = \rho C_\mu \frac{k^2}{\epsilon}$$

The equations also consist of some adjustable constants $\sigma_k, \sigma_\epsilon, C_{1\epsilon}$ and $C_{2\epsilon}$. We can take for these constants : $C_\mu = 0.09$ $\sigma_k = 1.00$ $\sigma_\epsilon = 1.30$ $C_{1\epsilon} = 1.44$ $C_{2\epsilon} = 1.92$

3.1.3.2. The k- ω model

The k- ω turbulence model is also a common model and is used as an approximation for RANS. The model attempts to predict turbulence by two partial differential equations for two variables, k and ω , with the first variable being the turbulence kinetic energy (k) as for the first model while the second (ω) is the specific rate of dissipation. The eddy viscosity ν_T , as needed in the RANS equations, is given by: $\nu_T = \frac{k}{\omega}$, while the evolution of k and ω is modeled as :

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = \rho P - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[\left(\mu + \sigma_k \frac{\rho k}{\omega} \right) \frac{\partial k}{\partial x_j} \right] \text{ for } k \quad \text{eq 17.}$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u_j \omega)}{\partial x_j} = \frac{\alpha \omega}{k} P - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[\left(\mu + \sigma_\omega \frac{\rho k}{\omega} \right) \frac{\partial \omega}{\partial x_j} \right] + \frac{\rho \sigma_d}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \text{ for } \omega \quad \text{eq 18.}$$

$$\text{with } P = \tau_{ij} \frac{\partial u_i}{\partial x_j}$$

3.1.3.3. The laminar model

In fluid dynamics, laminar flow is characterized by fluid particles following smooth paths in layers, with each layer moving smoothly past the adjacent layers with little or no mixing. At low velocities, the fluid tends to flow without lateral mixing, and adjacent layers slide past one another. There are no cross-currents perpendicular to the direction of flow, nor eddies or swirls of fluids. In laminar flow, the motion of the particles of the fluid is very orderly with particles close to a solid surface moving in straight lines parallel to that surface. This is the model that we use for the simulation because of the time of computation with a turbulent model and k epsilon.

3.2. Fluent resolution (Pre-processing)

3.2.1. Fluent

As we saw in the theory section, complete solutions to the fluid equations are very difficult to reach, so it is necessary to approximate them with computer tools. For this purpose, we use software like ANSYS Fluent. This is a simulation software used to predict the behavior of a fluid (liquid or gas) in its environment; it implements the finite element method. The Fluent code is a program for numerical simulation of fluid flows compressible, incompressible, stationary or unsteady and fluid flows involving various physical phenomena such as heat transfer, turbulence, chemical reactions, flows in rotating machines, thermal engines, and this for industrial geometries that can be very complex. This code also allows the refinement of the mesh according to the boundary conditions and dimensions conditions. ANSYS fluent is in other words a solver, but to solve fluid dynamics problems we will use ANSYS Workbench to create the geometry and mesh and analyze the results.

3.2.2. Generation of the geometry

The first step was to generate the geometry, for this we used ANSYS Design Modeler. The flow around the cylinder is modeled in 2-D. All fluid flows are three-dimensional, so 3D models must be used to analyze fluids, and fluids with complex geometries in particular. In some cases, appropriate assumptions can be made to reduce a 3D problem to two dimensions to simplify its numerical analysis. As you can see on the *figure 3* the cylinder is modeled by a circle of radius $D10=50\text{mm}$, positioned at

a distance $H11=125\text{mm}$ from the right edge and is located in the domain represented by a rectangle of $H8=900\text{ mm}$ length and $V6+V9=300\text{mm}$ height.

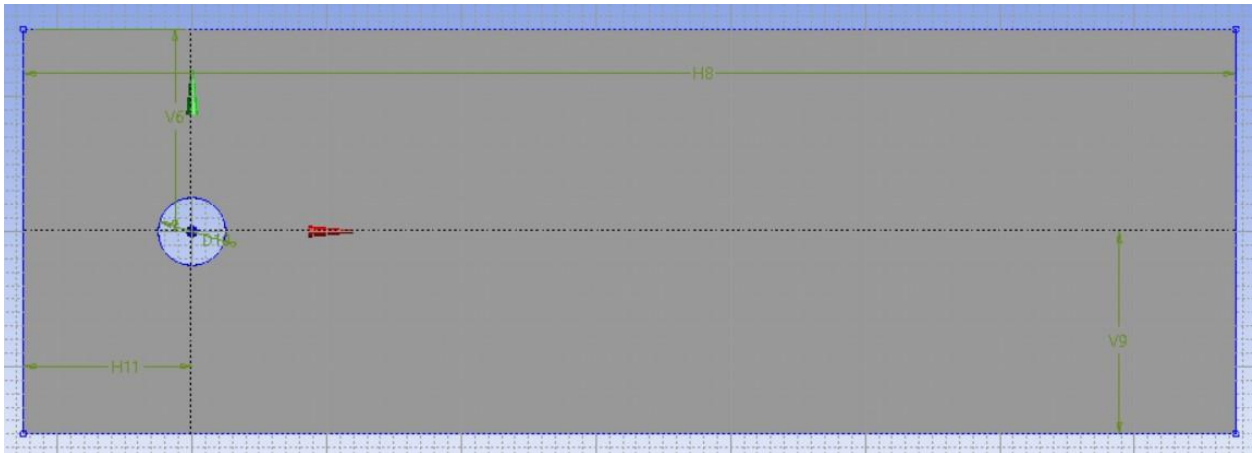


Figure 3: Screenshot of the geometry

3.2.3. Generation of the mesh

To create the mesh we use the second software available on the workbench, namely ANSYS Meshing. The mesh allows us to divide the computational domain into small control volumes. Then we can solve the equations at one point of each cell, and apply this result to the whole cell. Meshing plays a critical role in achieving stable solutions. Two types of meshes can be used. First, we can use structured grids in which the cells have the same shape (for example rectangles, squares or triangles), and are aligned or arranged following the shape that we study. The other type of grid is the unstructured grid, which is a combination of triangles generated randomly (Figure 4).

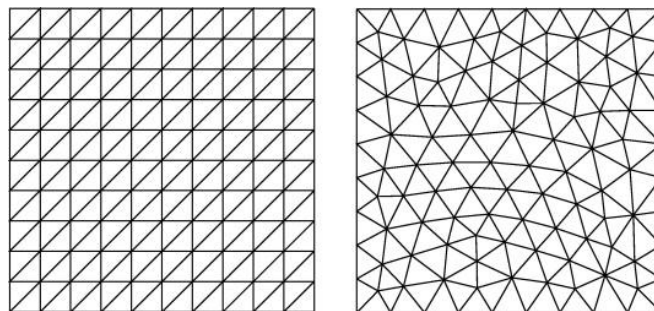


Figure 4: Structured mesh (left) and unstructured mesh (right)

The two grids are used in CFD. The main difference between them is that the unstructured mesh allows us to fit all complex shapes, unlike structured meshes. But it is more complicated to generate, so the calculations are longer to do and the simulation becomes more expensive. In our case, both grids can be used as the geometry of a cylinder is very simple, but we chose to use an unstructured grid because it fits the shape of the cylinder better. We also have the choice of the shape of the control volumes, quadrilaterals or triangles, for the same reason as before we chose to use the triangle method alone. Our mesh contains elements of size 0.006m (Figure 6). In order to be more accurate around the cylinder, we added a dimensioning around the cylinder edge with an element size 0.00009m with a growth rate of 1.1. Furthermore, we added inflation around the cylinder edge of 10

layers with a growth rate of 1.2 (*Figure 5*), in order to see the boundary layer clearly. The mesh must be fine enough to capture in detail the turbulence and the boundary layer for example, but at the same time, it must not be too fine to have a reasonable calculation time. In the end, our mesh contains 67362 elements and 43330 nodes.

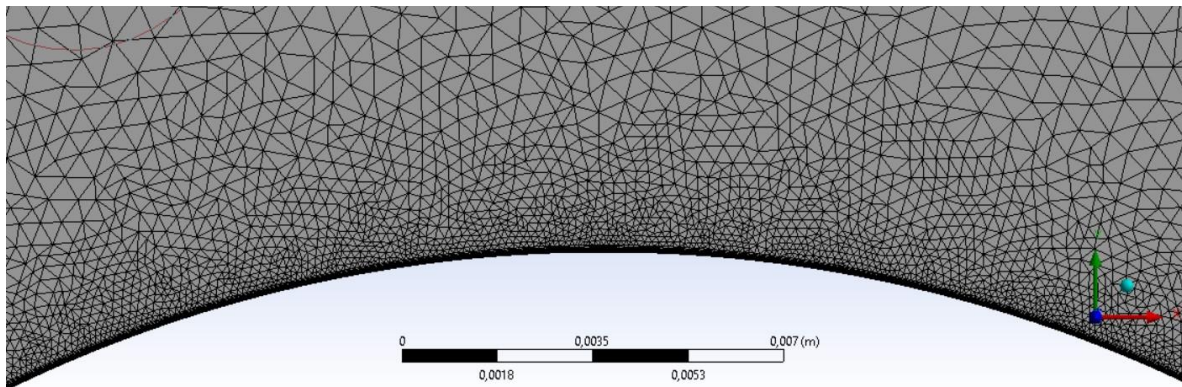


Figure 5: Focus on boundary layers and edge sizing

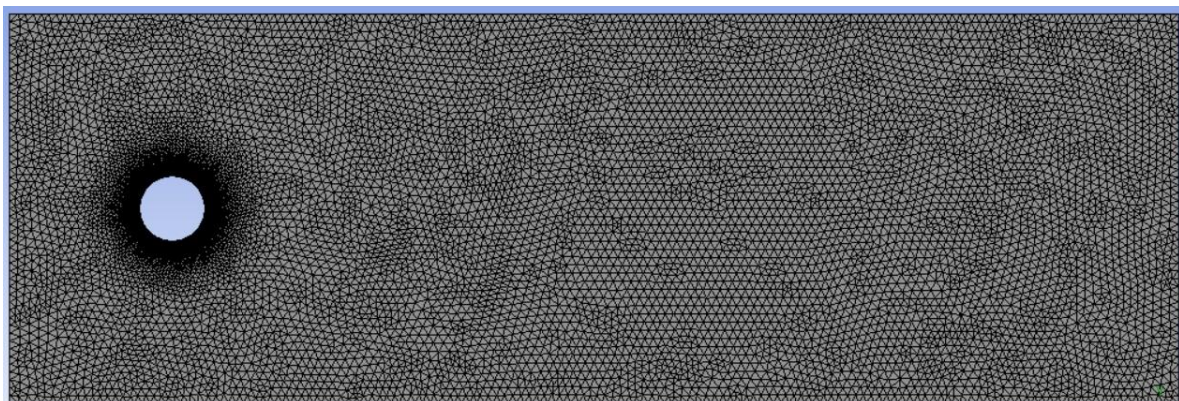


Figure 6: Screenshot of the mesh

3.2.4. Choice of boundary conditions and initial conditions

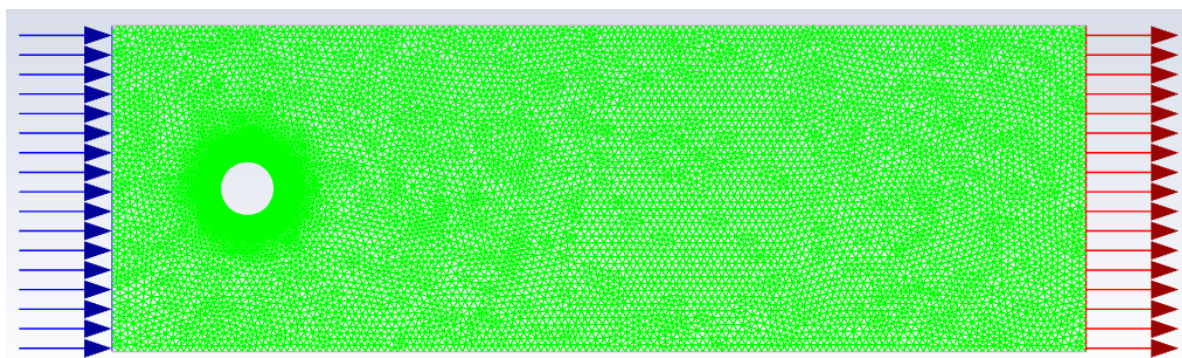


Figure 7: Screenshot of the mesh and boundary conditions seen on Fluent

The next step was to define the types of boundaries. Not all edges in geometry have the same function, so it was important to determine how Fluent would interpret them. First, we had to add some named selections in ANSYS Meshing, i.e. Wall Inlet Outlet Cylinder, and then we could change the function of these selections in Fluent. The left edge is called inlet, the right edge outlet, the cylinder edge cylinder, and the two remaining edges were designated Wall (*Figure 7*). It is important to distinguish between wall and cylinder as they do not have the same boundary conditions. Indeed the wall only delimits the domain, therefore they are defined by slip conditions, for this we select specified shear and enter zero shear stress components. While the cylinder must be specified without slippage in the boundary conditions to observe the boundary layer on the edge of the cylinder.

Velocity (m/s)		0.000291	0.01168	0.04090	5,843
Boundary conditions	Velocity inlet (m/s)	0.000291	0.01168	0.04090	5,843
	Pressure outlet(Pa)	0	0	0	0
	Cylinder	no slip	no slip	no slip	no slip
	Wall	specified shear: 0 Pa on x and y axis	specified shear: 0 Pa on x and y axis	specified shear: 0 Pa on x and y axis	specified shear: 0 Pa on x and y axis
Initial conditions		0.000291	0.01168	0.04090	5,843

Table 4: Boundary and initial conditions

At this point, the solution needed to be initialized. To do this, we have chosen a standard initialization with an initial value of velocity equal to the one in Inlet, see table 4.

3.2.5. Specification of numerical parameters

The fluid used for our simulation is air of $\rho=1,225 \text{ kg.m}^{-3}$ and $\mu=1,7894.10^{-5} \text{ kg.ms}^{-1}$. The calculation parameters are listed in the table below. To determine the time step size it is important to take into account the size of the domain and the speed of the input fluid. Indeed this parameter must be optimized in order to reduce the calculation time but to keep accuracy in the visualization of the flowfield.

Flow regime	Reynolds number	Velocity (m/s)	Model	Resolution	
Laminar steady flow	1	0.000291	Laminar	Steady	
Pair of vortices, laminar steady flow	40	0,01168	Laminar	Transient	
				Time step size (s):	0.05
				Time step:	1000
				Iterations/time step:	20

Laminar vortex street, unsteady flow	140	0,04090	Laminar	Transient	
				Time step size (s):	0.1
				Time step:	500
				Iterations/time step:	20
Transitional unsteady flow	20000	5,843	K-ε Cmu = 0.09 C1 = 1.44 C2 = 1.92 TKE = 1 TDR = 1.3	Transient	
				Time step size (s):	0.0005
				Time step:	500
				Iterations/time step:	50

Table 5: Simulation parameters

3.3. Results and discussion

First, we introduce a concept named residues which are employed to monitor the behavior of the numerical process. At the end of each solver iteration, the sum of residuals for each conserved variable is computed and stored, and these residuals will go to zero as the solution converges. In ANSYS, residuals can be displayed using an X-Y plot. The abscissa of the plot corresponds to the number of iterations and the ordinate corresponds to the log-scaled residual values. Most importantly, make sure the value monitors are not jagged but rather smooth (jagged means the time step is too large). Basically, you want to use the largest time step possible as long as it accurately describes/fits the period of your force monitors.

Next, we will talk about two force coefficients. The **lift force** is a force applied to an object in the y-direction. It (represented by the lift coefficient in the plot below) varies between positive and negative peaks in a sinusoidal manner, because the vortex shedding alternates on either side of the cylinder. The positive peak corresponds to the vortex shed on the upper side, and the negative peak to the vortex shed on the lower side. The period equals the time between vortex shedding on the same side. The **drag force** (represented by its coefficient) oscillates at half the period of the lift force, as both vortices shed from the top and bottom induce the drag force in the same direction. Unlike lift, the drag force is always positive. The lift coefficient decreases rapidly and the drag coefficient increases rapidly when the Reynolds number is decreased below critical range.

Afterwards, we have to indicate that the vortex is a region in the fluid domain which causes a swirl in a region where there is asymmetric drag in the flow field. Vortices are formed behind the cylinder and shed periodically from either side. Then the alternating vortices travel along with the main flow towards the downstream region.

Finally, we can make a rough classification with stable and unstable. If you try to simulate all of these different flows with a **steady simulation** you will succeed with the 1. and 2. because they inherently don't exhibit that much transient behavior. What happens with 3. and 4. is that your residuals are going to start oscillating and your solution will not converge. This is when you switch to an **unsteady solution** and guesstimate a time step based on the physics of the flow (higher frequency = smaller time step).

3.3.1. Re=1(Laminar steady flow):

Laminar Flow ($Re \leq 2300$) occurs at low velocity, both the drag forces and the lift forces converge to 0 (see *Annexe 1*). That means the layers of fluid seem to slide smoothly by one another without vortices being present (there is no convective mixing).

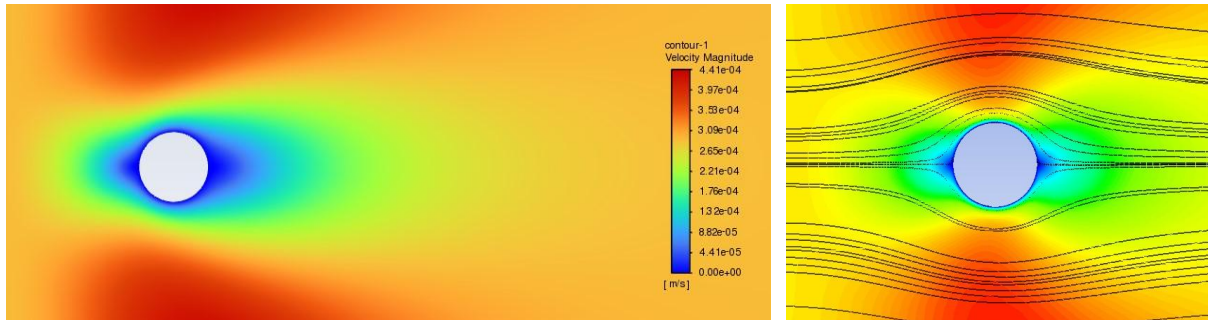


Figure 8: Screenshot of the velocity magnitude for $Re=1$ (left) and streamlines (right)

There are two stagnation points on two edges of the cylinder where the velocity is 0. As the fluid goes around the cylinder, the velocities are literally high and then due to the separation you get a low velocity region right behind the cylinder. At $Re=1$ we are in a special case, as we can see from the current line shown in *figure 8* the flow is purely viscous and almost vertically symmetric.

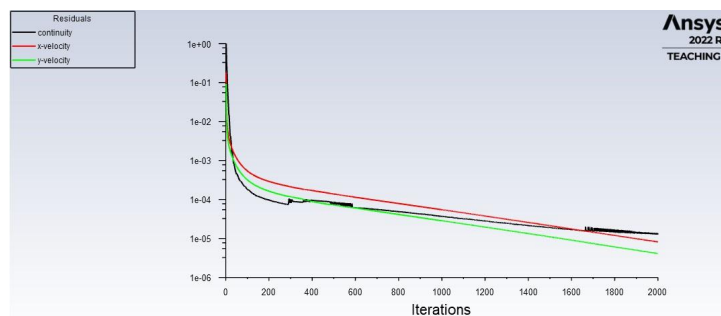


Figure 9: Screenshot of the residuals for $Re=1$

The residues fall smoothly after iterations and we call it perfectly convergence. (*Figure 9*)

3.3.2. Re=40 (Pair of vortices, laminar steady flow):

For fluid flow with low Re (≤ 2300), the influence of vortex currents is significant. We first consider a pair of vortices. The lift force oscillates for half a cycle (see *Annexe 2*). This means there will be no sustained oscillation, we call this case local instability, and we can still use steady simulation.

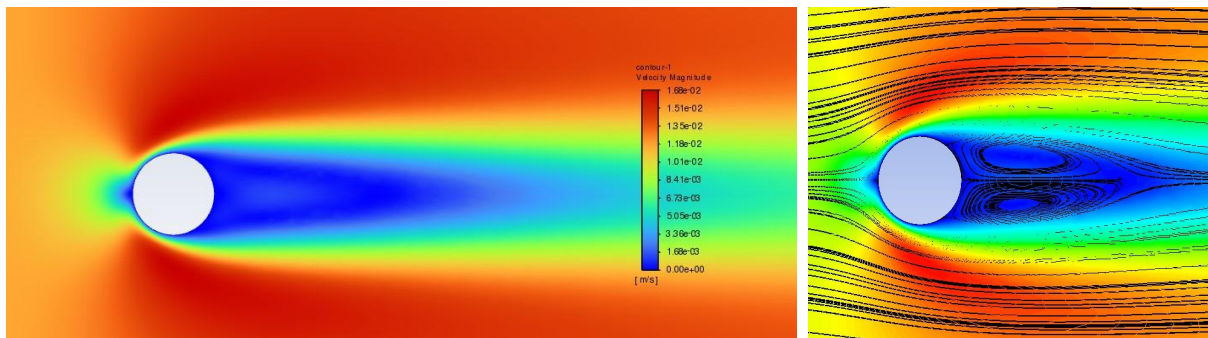


Figure 10: Screenshot of the velocity magnitude for $Re=40$ (left) and streamlines (right)

Obviously, there are two stagnation parts on two edges of the cylinder. As the fluid flow goes around the cylinder, the velocities are literally high and then due to the separation you get a low velocity region right behind the cylinder as before. However, because of a pair of vortices behind the cylinder highlighted by the streamlines (Figure 10), between the two of them, there is a small increase of velocity.

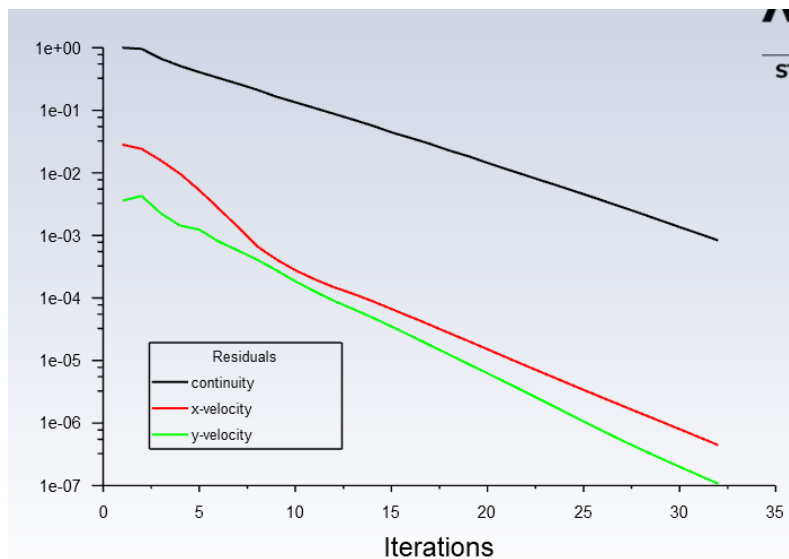


Figure 11: Screenshot of the residuals for $Re=40$

The residuals start oscillating at first, but as time goes on, it eventually converges. (Figure 11) The residues are very good as they exceed 1×10^{-6} .

3.3.3. Re=140 (Laminar vortex street, unsteady flow):

We next consider vortex street. As shown in the image above, the lift force continues to oscillate. Meanwhile, the drag force keeps oscillating at half its cycle (see *Annexe 3*). Thus, the residuals are going to keep oscillating and our solution will not converge. We call this case global instability. Eventually, we switch to an unsteady solution.

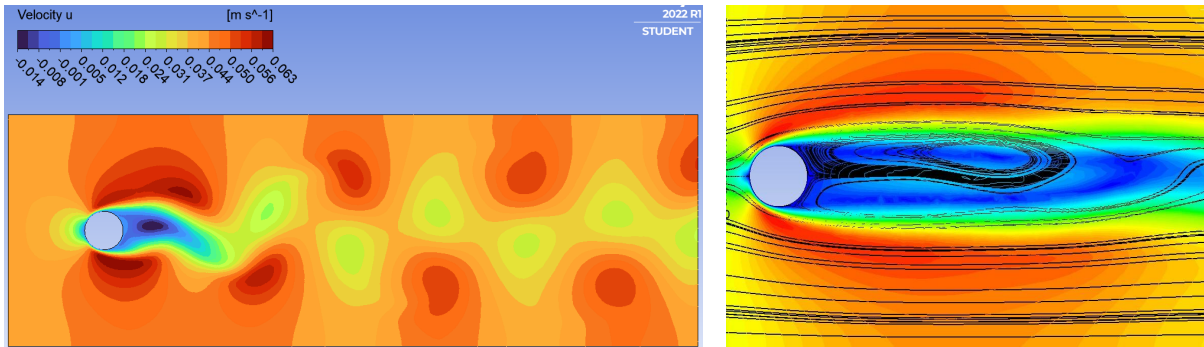


Figure 12: Screenshot of the u -velocity magnitude for $Re=140$ (left) and streamlines (right)

Two stagnation parts on two edges of the cylinder as before and high velocity in the part through which the fluid passes directly. Due to the vortex street, it is more obvious than a pair of vortices, we can clearly see that there is an unstable part in the stagnation area.

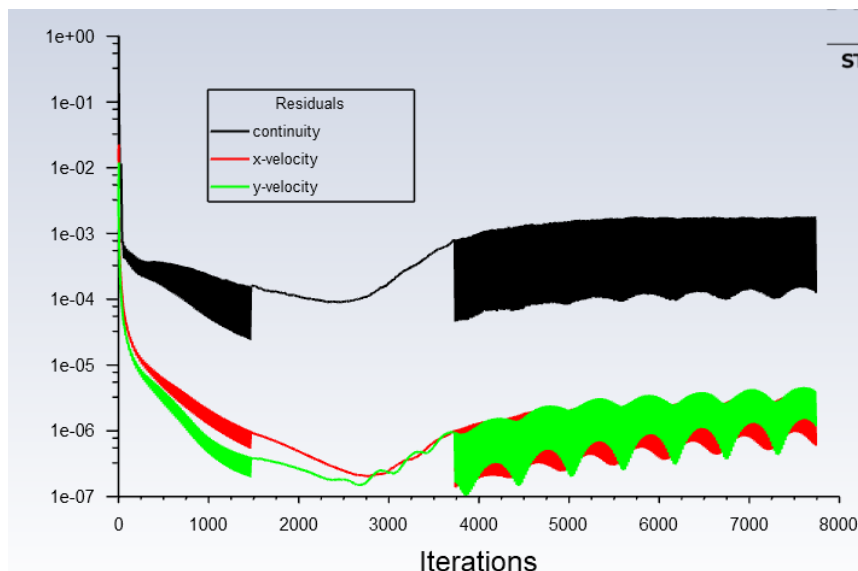


Figure 13: Screenshot of the residuals for $Re=140$

The residuals keep oscillating, and they eventually do not converge. (Figure 13)

3.3.4. Re=20000 (Transitional unsteady flow):

For Transitional unsteady flow, it is a mixture of laminar and turbulent flow, with turbulence in the center of the cylinder, and laminar flow near the edges. With the enormous Re value, we cannot ignore the friction between layers of fluid. That gives the fluid a fluctuating nature.

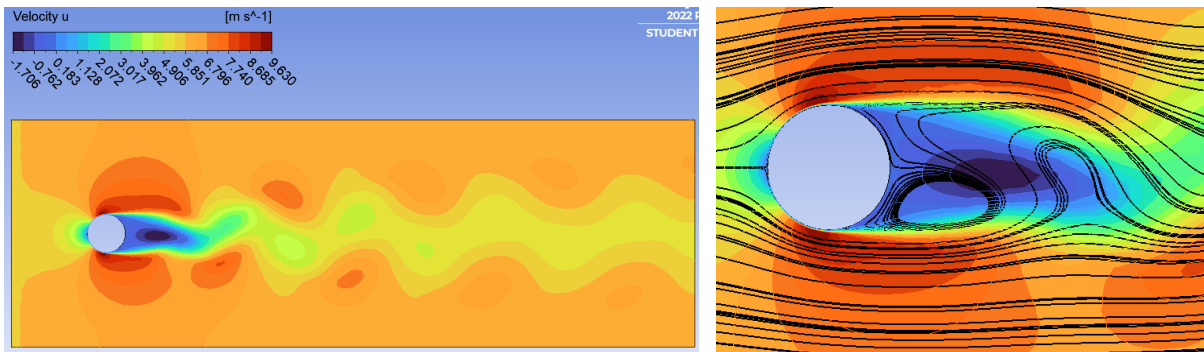


Figure 14: Screenshot of the u -velocity magnitude for $Re=20000$ (left) and streamlines (right)

Transitional unsteady flows are always highly irregular and chaotic. At the high-velocity area, there are also many irregularly shaped low-velocity areas that are all over the space. It is completely different from the previous example.

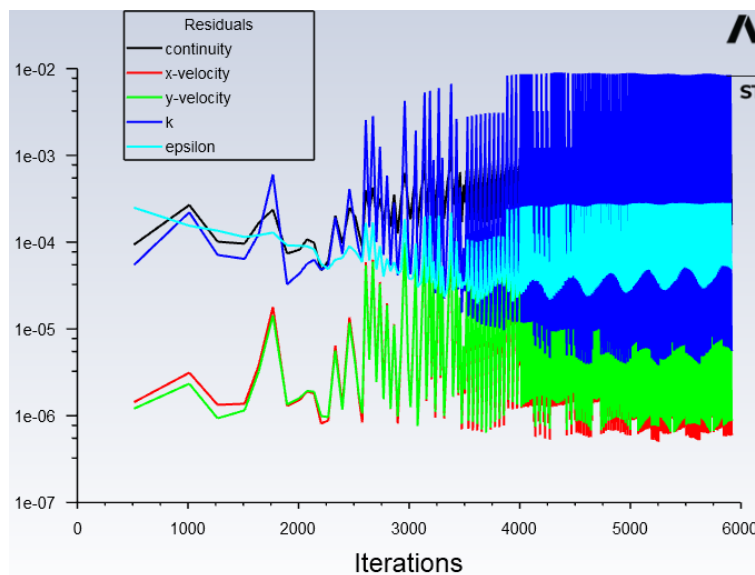


Figure 15: Screenshot of the residuals for $Re=20000$

The residuals keep oscillating more frequently than laminar vortex street, and eventually do not converge too. Each peak corresponds to the time step change, the residuals are still satisfying as they are between 1×10^{-6} and 1×10^{-4} .

To sum up, when the viscous force plays a role and is greater than the inertial force (i.e., the Reynolds number is lower), the flow tends to be laminar. When the inertial force is greater than the viscous force (i.e., the Reynolds number is higher), it leads to flow separation and the flow tends to be turbulent.

4. CONCLUSION AND PERSPECTIVES

In conclusion, we can say that this project allowed us to acquire new notions in the field of CFD. Indeed, it allowed us to learn more concerning fluid dynamics, through the Navier Stokes equations, Reynolds number, boundary layer and others. It allowed us to understand laminar and turbulent flows. We also learned a lot concerning simulation software in general. In fact, we researched how the equations behind the simulations are analyzed, simplified and solved, especially the approximation methods. It is very important to know that before doing simulations because we can predict what the software is able to do, and sometimes some simplifications can hide the phenomena we want to study so it is important to be able to choose the right calculation methods and modelisations of our problem. Knowing that, we focused on Ansys Fluent and learned how to use its different modules in order to create various simulations, and exploit its results. With our results, we can see the impact of the Reynolds number on lift and drag forces and in that way on turbulence. The higher it is, the more turbulence there is.

From a personal point of view, we learned a lot through this project. This project taught us how to carry out group work, managing its organization, the possible constraints and problems encountered. Furthermore, it was really interesting in terms of group work. We carried out a long-term project with a group of people we didn't know before, so we had to share the tasks and adapt ourselves in order to work as harmoniously as possible. We will probably encounter the same situation again in our future lives, so this project has been a good experience.

On top of that, it could be very interesting to carry out experiments with a real cylinder and a wind tunnel. Thus, it would be possible to compare the results of our simulation and of the experiment.

BIBLIOGRAPHY

Books

- [1] J.H. Fergizer, “Computational Methods for Fluid Dynamics-annotated”, Springer International Publishing AG, 2019 (Chapter 2, Chapter 3, Chapter 6)
- [2] J.D. Anderson Jr. (auth.), Prof. Dr. John F. Wendt (eds.), “Computational Fluid Dynamics: An Introduction”, Springer Berlin Heidelberg, 1992 (Chapter 1, Chapter 2)

Websites

- [3] <https://www.simscale.com/blog/2016/03/what-everybody-ought-to-know-about-cfd/> (consulted on 27/02/2022)
- [4] https://en.wikipedia.org/wiki/Reynolds_number#:~:text=The%20Reynolds%20number%20is%20the%20interior%20of%20a%20pipe. (consulted on 24/02/2022)
- [5] <https://courses.ansys.com/index.php/courses/simple-approximations-of-fluid-flows/lessons/simulation-examples-homework-quizzes/topic/unsteady-flow-over-a-cylinder-simulation-example/> (consulted on 25/03)
- [6] https://en.wikipedia.org/wiki/Navier%E2%80%93Stokes_equations (consulted on 24/02/2022)
- [7] https://en.wikipedia.org/wiki/Boundary_layer#:~:text=In%20physics%20and%20fluid%20mechanics,zero%20velocity%20at%20the%20wall. (consulted on 24/02/2022)
- [8] <https://www.manchestercfd.co.uk/post/all-there-is-to-know-about-different-mesh-types-in-cfd> (consulted on 25/02/2022)
- [9] <http://emmanuelplaut.perso.univ-lorraine.fr/mf/pol.pdf> (consulted on 25/02/2022)
- [10] https://en.wikipedia.org/wiki/K%E2%80%93omega_turbulence_model (consulted on 29/05/2022)
- [11] https://en.wikipedia.org/wiki/K-epsilon_turbulence_model (consulted on 29/05/2022)
- [12] <https://www.nagwa.com/fr/explainers/817184192395/> (consulted on 04/06/2022)

Videos

- [13] https://www.youtube.com/watch?v=mDt-HJEXK6g&ab_channel=SteveBrunton
- [14] https://www.youtube.com/watch?v=pe6AxHpvBJU&ab_channel=SteveBrunton
- [15] https://www.youtube.com/watch?v=zlQpxmLwbXQ&ab_channel=SteveBrunton
- [16] <https://www.youtube.com/watch?v=DXWO-X16UIoü>
- [17] <https://www.youtube.com/watch?v=6xVF6qJmOnI>

TABLE OF ILLUSTRATIONS

Figures

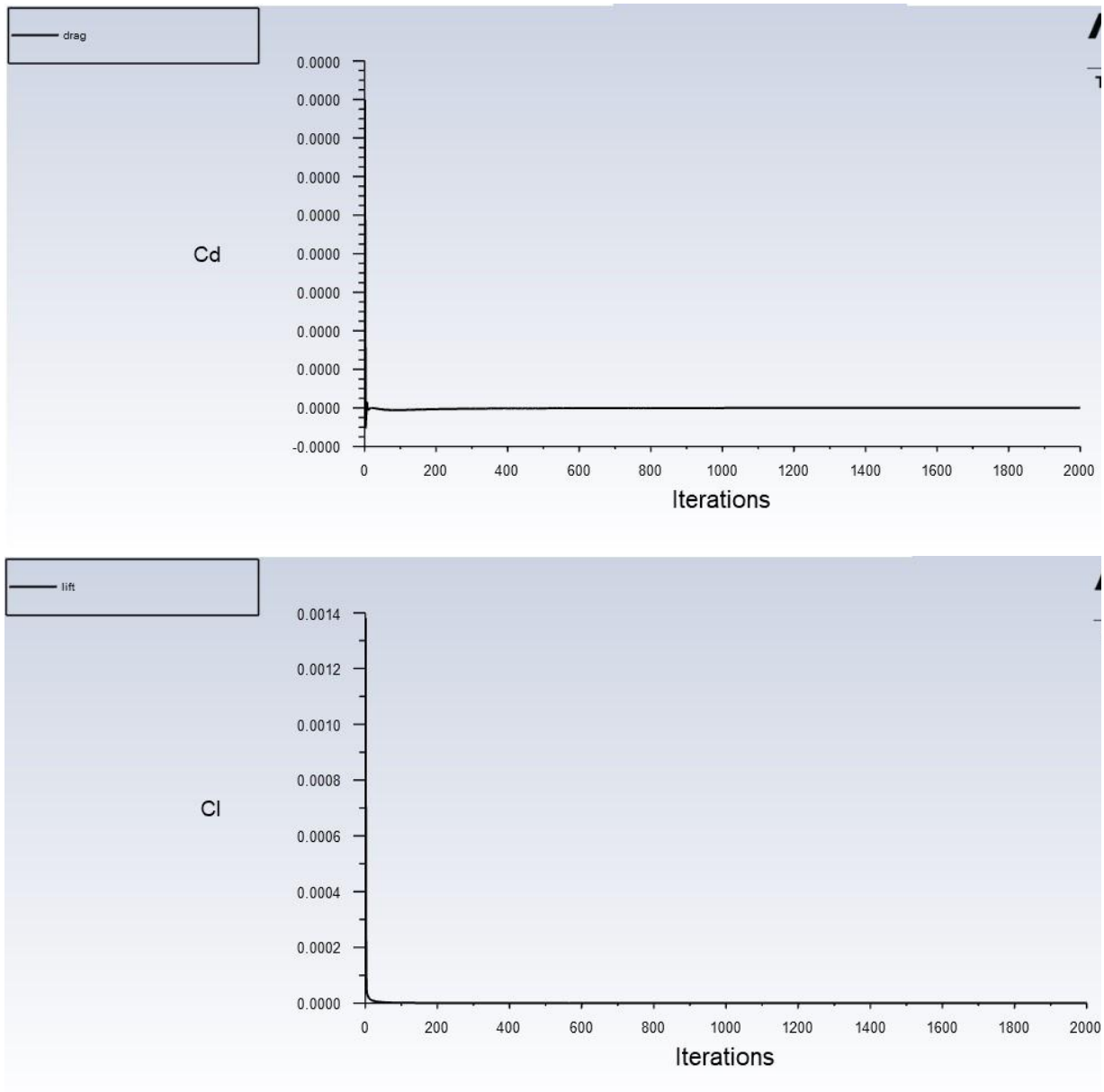
- Figure 1: *Boundary layer pattern*
<https://www.nuclear-power.com/nuclear-engineering/fluid-dynamics/turbulent-flow/turbulent-boundary-layer/>
- Figure 2 : *Simulation results depending on the approximations*
https://en.wikipedia.org/wiki/Large_eddy_simulation#cite_note-1
- Figure 3: *Screenshot of the geometry*
- Figure 4 : *Structured mesh (left) and unstructured mesh (right)*
https://www.researchgate.net/figure/Example-of-a-structured-triangular-mesh-and-an-unstructured-triangular-mesh-in-a-unit-box_fig3_259159803
- Figure 5: *Focus on boundary layers and edge sizing*
- Figure 6: *Screenshot of the mesh*
- Figure 7: *Screenshot of the mesh and boundary conditions seen on Fluent*
- Figure 8: *Screenshot of the velocity magnitude for Re=1*
- Figure 9: *Screenshot of the residuals for Re=1*
- Figure 10: *Screenshot of the velocity magnitude for Re=40*
- Figure 11: *Screenshot of the residuals for Re=40*
- Figure 12: *Screenshot of the velocity magnitude for Re=140*
- Figure 13: *Screenshot of the residuals for Re=140*
- Figure 14: *Screenshot of the velocity magnitude for Re=20000*
- Figure 15: *Screenshot of the residuals for Re=20000 at the right and a zoom in at the left*

Tables

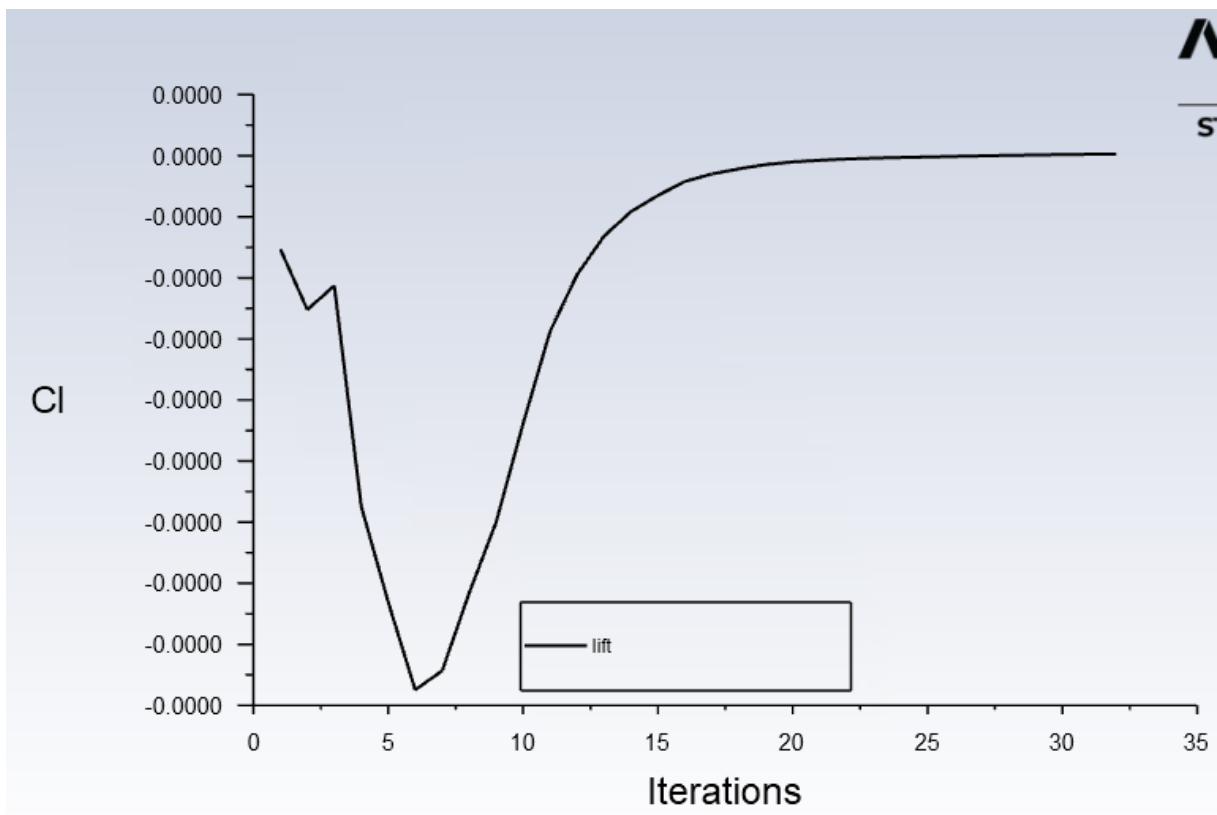
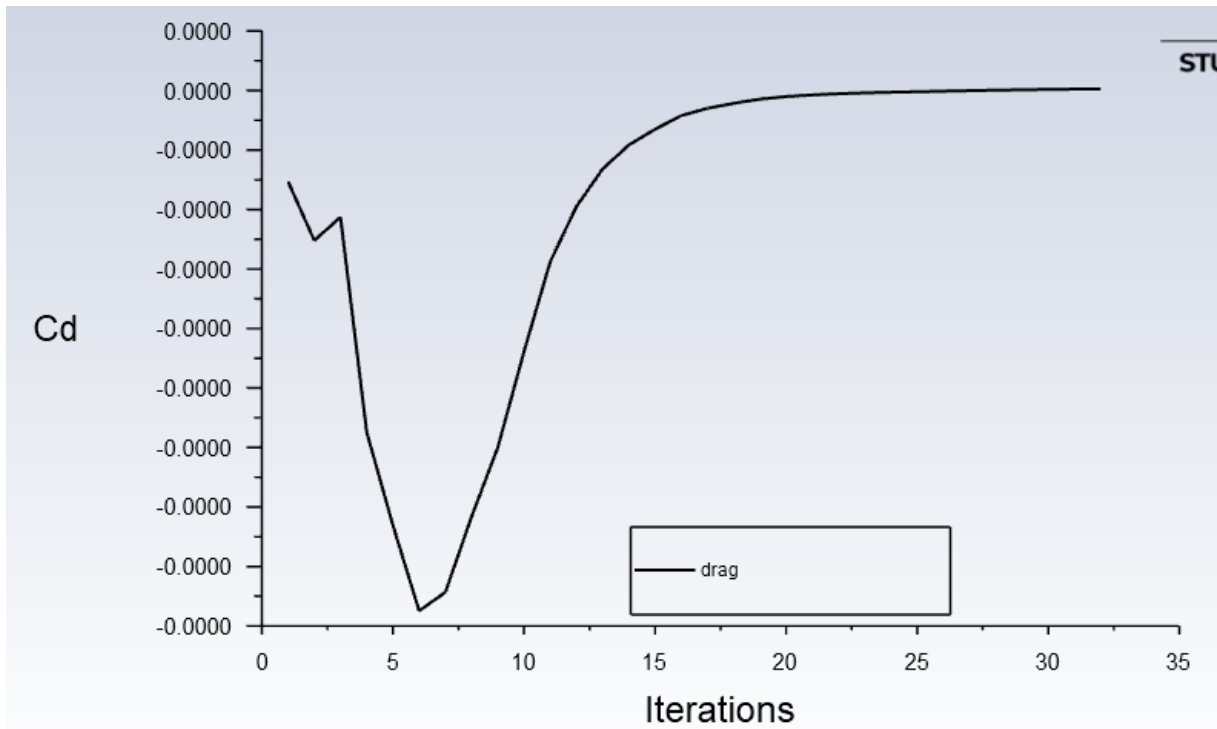
- Table 1: *Distribution of work by week (1.2: cf bibliography)*
- Table 2 : *Flow profiles depending on Reynolds numbers* <https://www.thermopedia.com/content/1216/>
- Table 3 : *Comparison between the approximation method*
- Table 4: *Boundary and initial conditions*
- Table 5: *Simulation parameters*

ANNEXES

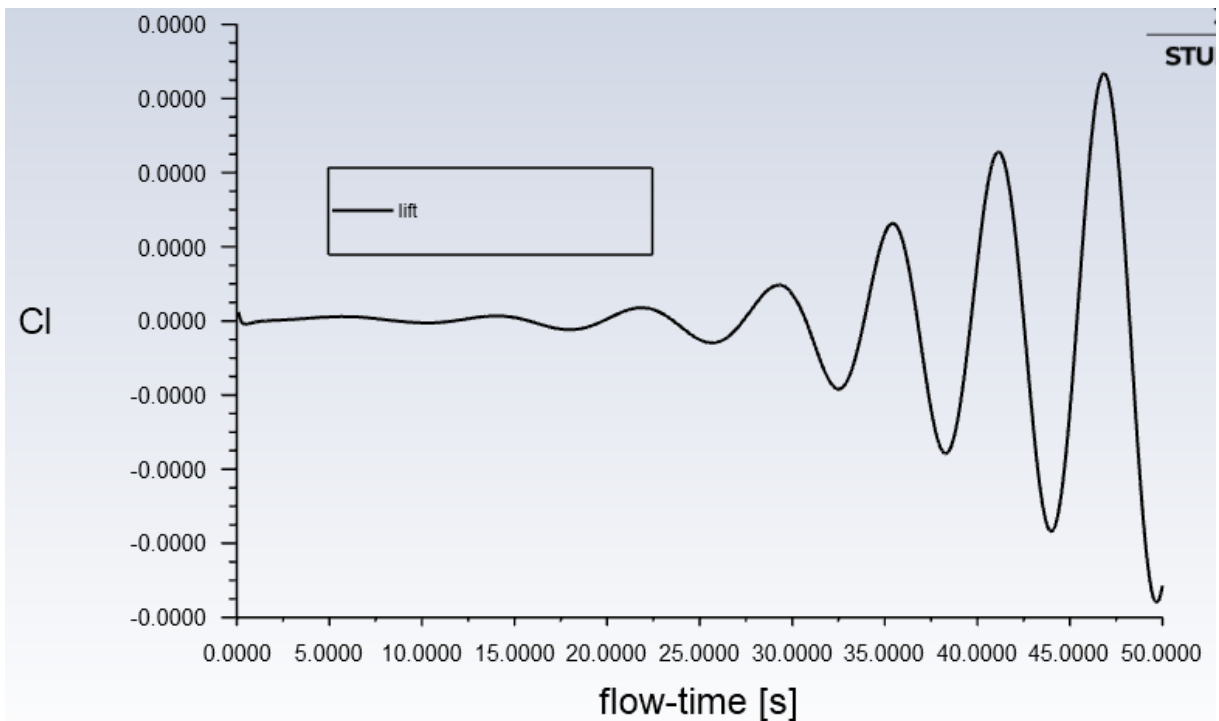
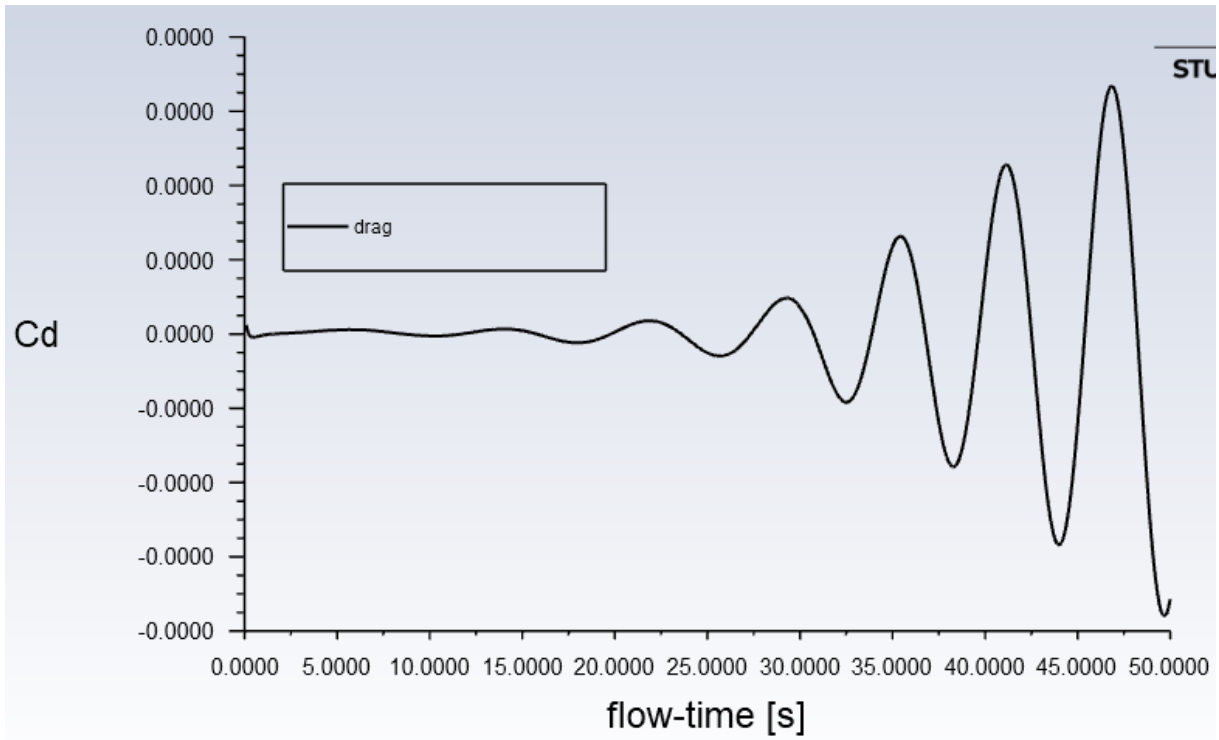
Annexe 1: Drag and lift coefficient for $Re=1$



Annexe 2: Drag and lift coefficient for $Re=40$



Annexe 3: Drag and lift coefficient for Re=140



Annexe 4: Drag and lift coefficient for Re=20000

