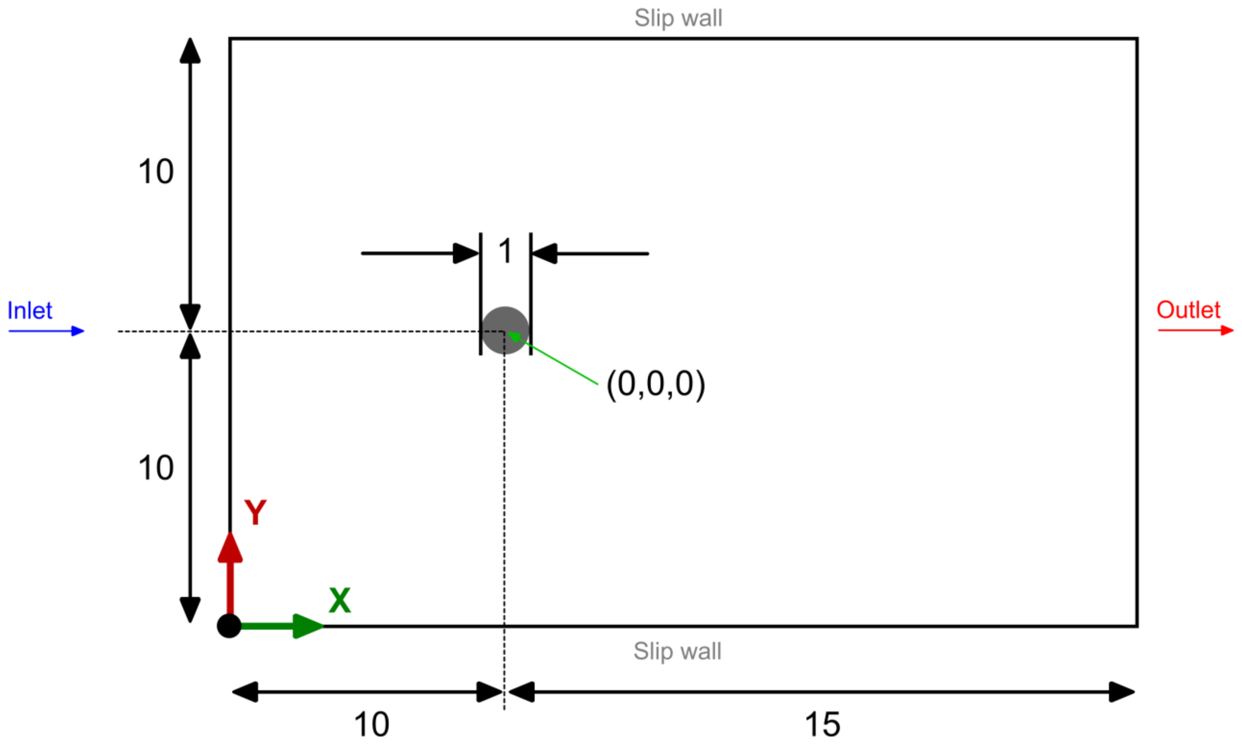
Problem definition

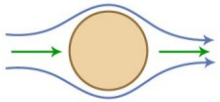


All the dimensions are in meters – Figure not to scale

- Incompressible flow.
- Density: 1 kg/m^3 (constant).
- Inlet velocity: 1 m/s
- Adjust the viscosity to change the Reynolds number.
- Run the case for Reynolds number values ranging from 10 to 1 000 000.
  - In this range of Reynolds number values, you will encounter steady and unsteady regimes.
  - And laminar and turbulent regimes.
- Sample the lift and drag coefficients.
- Sample velocity on the wake of the cylinder.
- Choose any turbulence model with the appropriate boundary and initial conditions.

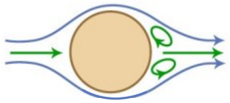
Problem definition

Steady physics



Creeping flow (no separation)
Steady flow

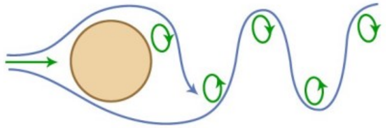
$Re < 5$



A pair of stable vortices in the wake
Steady flow

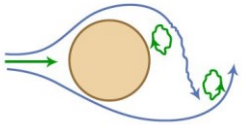
$5 < Re < 40 - 46$

Unsteady physics



Laminar vortex street (Von Karman street)
Unsteady flow

$40 - 46 < Re < 150$

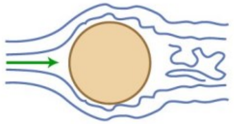


Laminar boundary layer up to the separation point, turbulent wake
Unsteady flow

$150 < Re < 300$

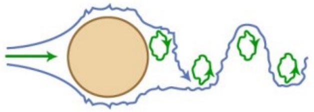
Transition to turbulence

$300 < Re < 3 \times 10^5$



Boundary layer transition to turbulent
Unsteady flow

$3 \times 10^5 < Re < 3 \times 10^6$

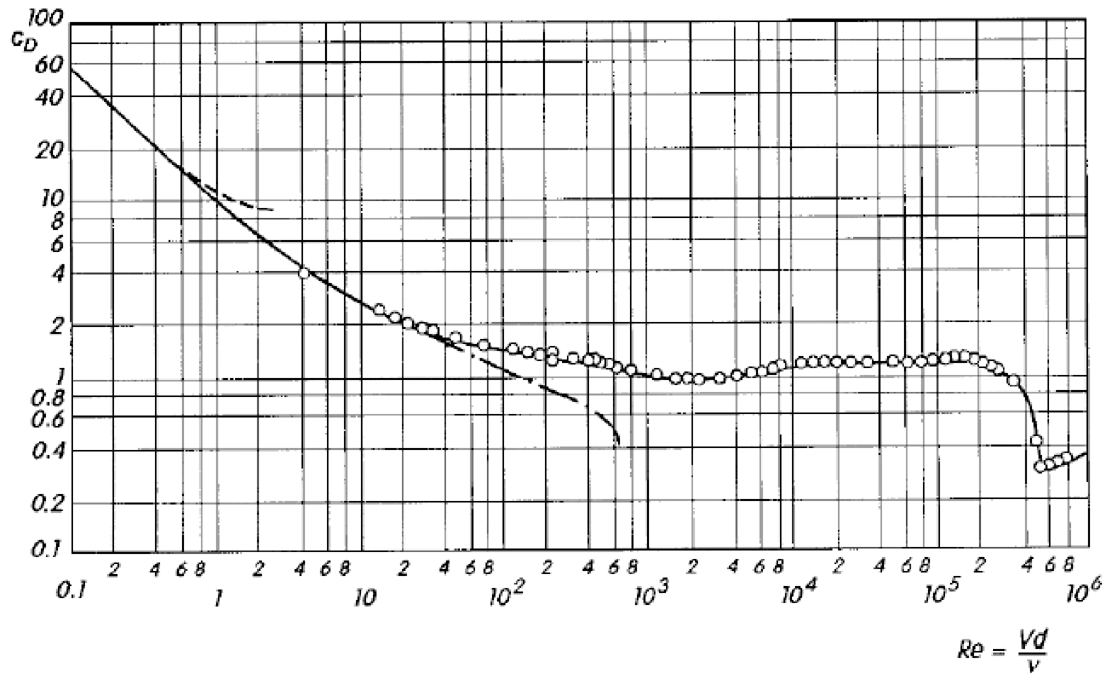


Turbulent vortex street, but the wake is narrower than in the laminar case
Unsteady flow

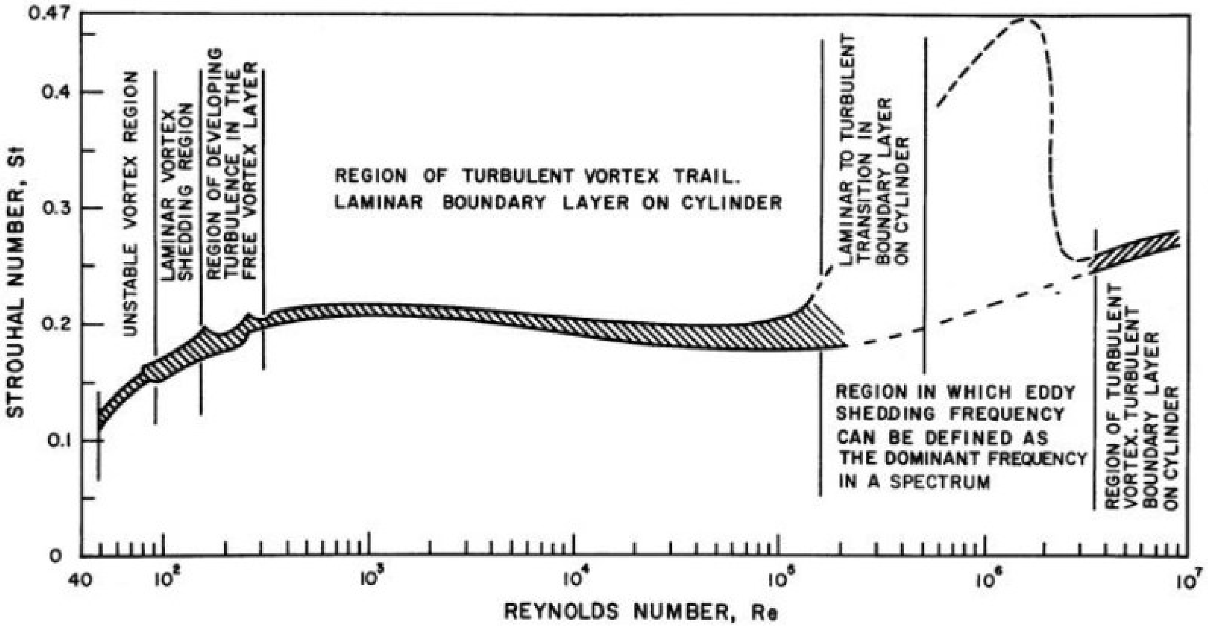
$3 \times 10^6 > Re$

Problem definition

This case has plenty of numerical and experimental data for validation



Drag coefficient versus Reynolds Number for Circular Cylinders [1]



Strouhal Number versus Reynolds Number for Circular Cylinders [2]

[1] H. Schlichting, K. Gersten. Boundary Layer Theory. Springer, 2017.
[2] R. Blevins. Flow-Induced Vibration. Krieger Publishing Company, 2001

Problem definition

Some experimental (E) and numerical (N) results of the flow past a circular cylinder at various Reynolds numbers

Reference	c_d – Re = 20	L_{rb} – Re = 20	c_d – Re = 40	L_{rb} – Re = 40
[1] Tritton (E)	2.22	–	1.48	–
[2] Cuntanceau and Bouard (E)	–	0.73	–	1.89
[3] Russel and Wang (N)	2.13	0.94	1.60	2.29
[4] Calhoun and Wang (N)	2.19	0.91	1.62	2.18
[5] Ye et al. (N)	2.03	0.92	1.52	2.27
[6] Fornbern (N)	2.00	0.92	1.50	2.24
[7] Guerrero (N)	2.20	0.92	1.62	2.21

$\mathbf{L_{rb}}$ = length of recirculation bubble, $\mathbf{c_d}$ = drag coefficient, **Re** = Reynolds number,

[1] D. Tritton. Experiments on the flow past a circular cylinder at low Reynolds numbers. Journal of Fluid Mechanics, 6:547-567, 1959.
[2] M. Cuntanceau and R. Bouard. Experimental determination of the main features of the viscous flow in the wake of a circular cylinder in uniform translation. Part 1. Steady flow. Journal of Fluid Mechanics, 79:257-272, 1973.
[3] D. Rusell and Z. Wang. A cartesian grid method for modeling multiple moving objects in 2D incompressible viscous flow. Journal of Computational Physics, 191:177-205, 2003.
[4] D. Calhoun and Z. Wang. A cartesian grid method for solving the two-dimensional streamfunction-vorticity equations in irregular regions. Journal of Computational Physics. 176:231-275, 2002.
[5] T. Ye, R. Mittal, H. Udaykumar, and W. Shyy. An accurate cartesian grid method for viscous incompressible flows with complex immersed boundaries. Journal of Computational Physics, 156:209-240, 1999.
[6] B. Fornberg. A numerical study of steady viscous flow past a circular cylinder. Journal of Fluid Mechanics, 98:819-855, 1980.
[7] J. Guerrero. Numerical simulation of the unsteady aerodynamics of flapping flight. PhD Thesis, University of Genoa, 2009.

Problem definition

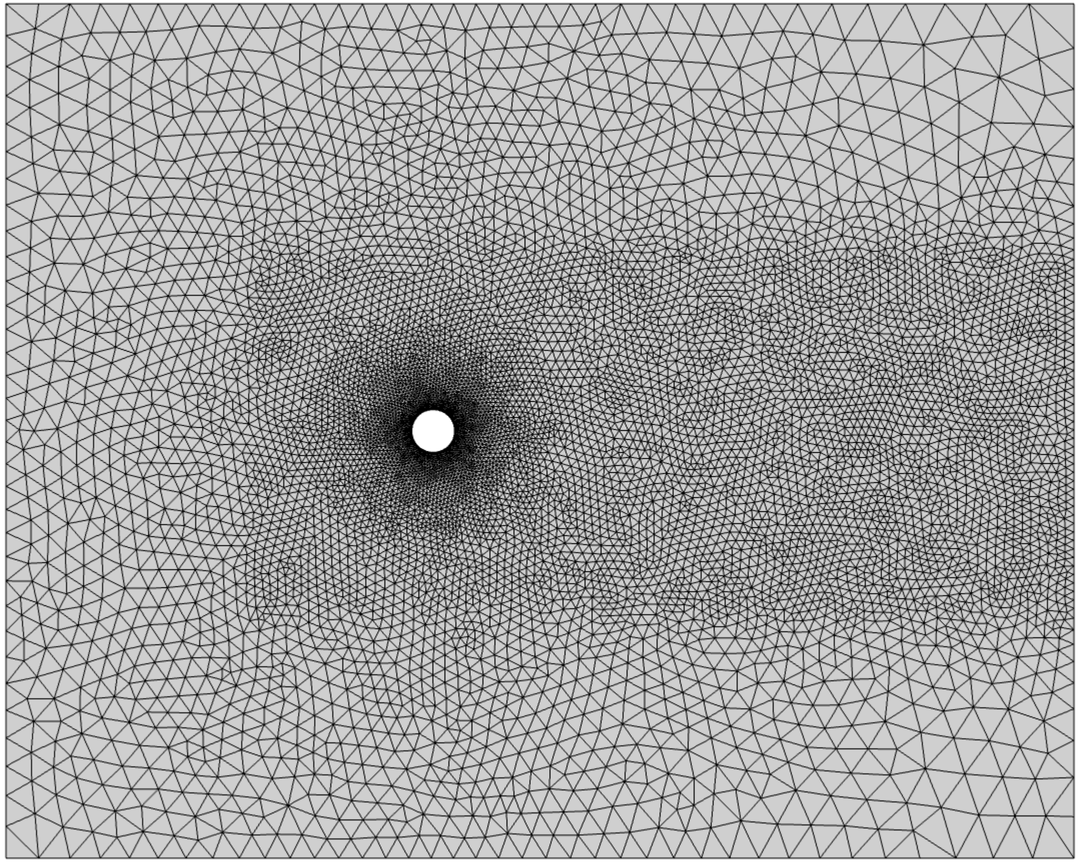
Some experimental (E) and numerical (N) results of the flow past a circular cylinder at various Reynolds numbers

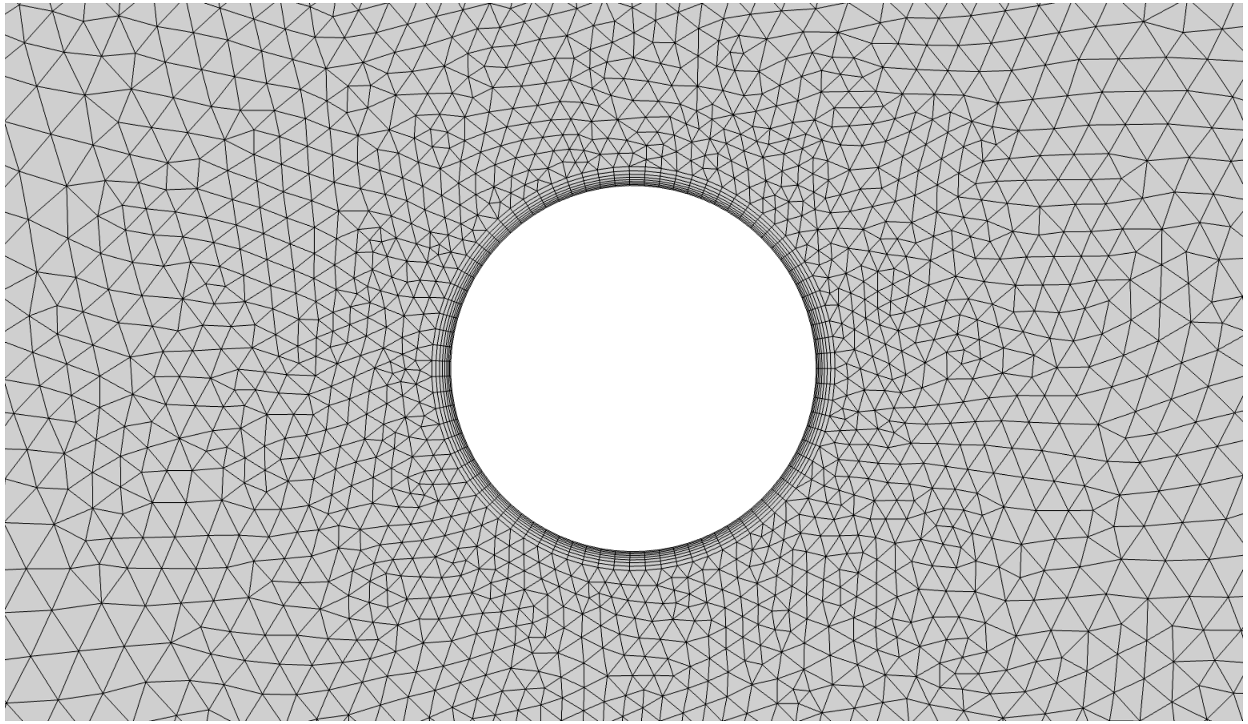
Reference	c_d – Re = 100	c_l – Re = 100	c_d – Re = 200	c_l – Re = 200
[1] Russel and Wang (N)	1.38 ± 0.007	± 0.322	1.29 ± 0.022	± 0.50
[2] Calhoun and Wang (N)	1.35 ± 0.014	± 0.30	1.17 ± 0.058	± 0.67
[3] Braza et al. (N)	1.386± 0.015	± 0.25	1.40 ± 0.05	± 0.75
[4] Choi et al. (N)	1.34 ± 0.011	± 0.315	1.36 ± 0.048	± 0.64
[5] Liu et al. (N)	1.35 ± 0.012	± 0.339	1.31 ± 0.049	± 0.69
[6] Guerrero (N)	1.38 ± 0.012	± 0.333	1.408 ± 0.048	± 0.725

$\mathbf{c_l}$ = lift coefficient, $\mathbf{c_d}$ = drag coefficient, **Re** = Reynolds number

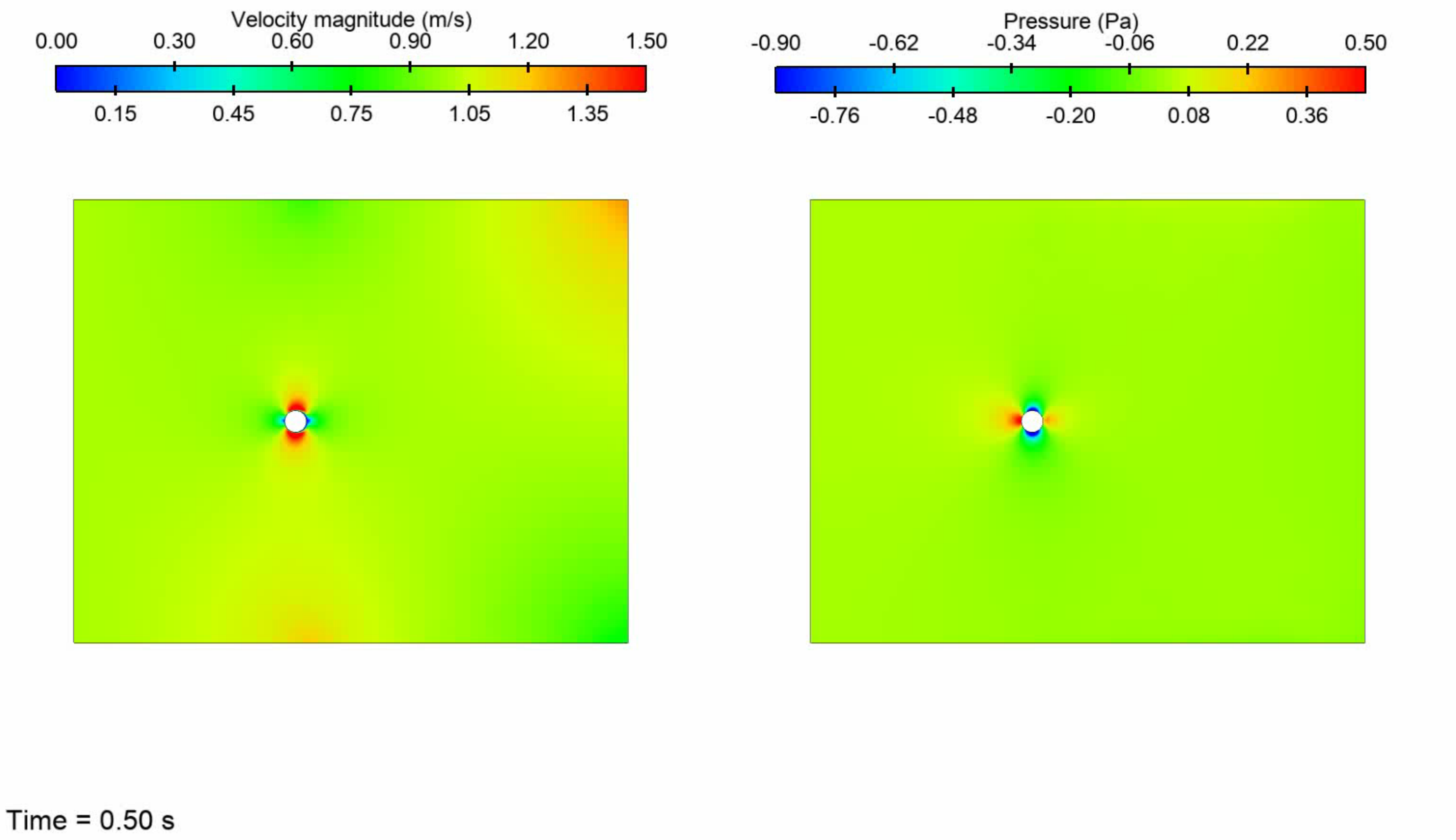
[1] D. Rusell and Z. Wang. A cartesian grid method for modeling multiple moving objects in 2D incompressible viscous flow. Journal of Computational Physics, 191:177-205, 2003.
[2] D. Calhoun and Z. Wang. A cartesian grid method for solving the two-dimensional streamfunction-vorticity equations in irregular regions. Journal of Computational Physics. 176:231-275, 2002.
[3] M. Braza, P. Chassaing, and H. Hinh. Numerical study and physical analysis of the pressure and velocity fields in the near wake of a circular cylinder. Journal of Fluid Mechanics, 165:79-130, 1986.
[4] J. Choi, R. Oberoi, J. Edwards, an J. Rosati. An immersed boundary method for complex incompressible flows. Journal of Computational Physics, 224:757-784, 2007.
[5] C. Liu, X. Zheng, and C. Sung. Preconditioned multigrid methods for unsteady incompressible flows. Journal of Computational Physics, 139:33-57, 1998.
[6] J. Guerrero. Numerical Simulation of the unsteady aerodynamics of flapping flight. PhD Thesis, University of Genoa, 2009.

Domain mesh – 2D mesh and inflation layer around the cylinder





Qualitative results – Contour plots

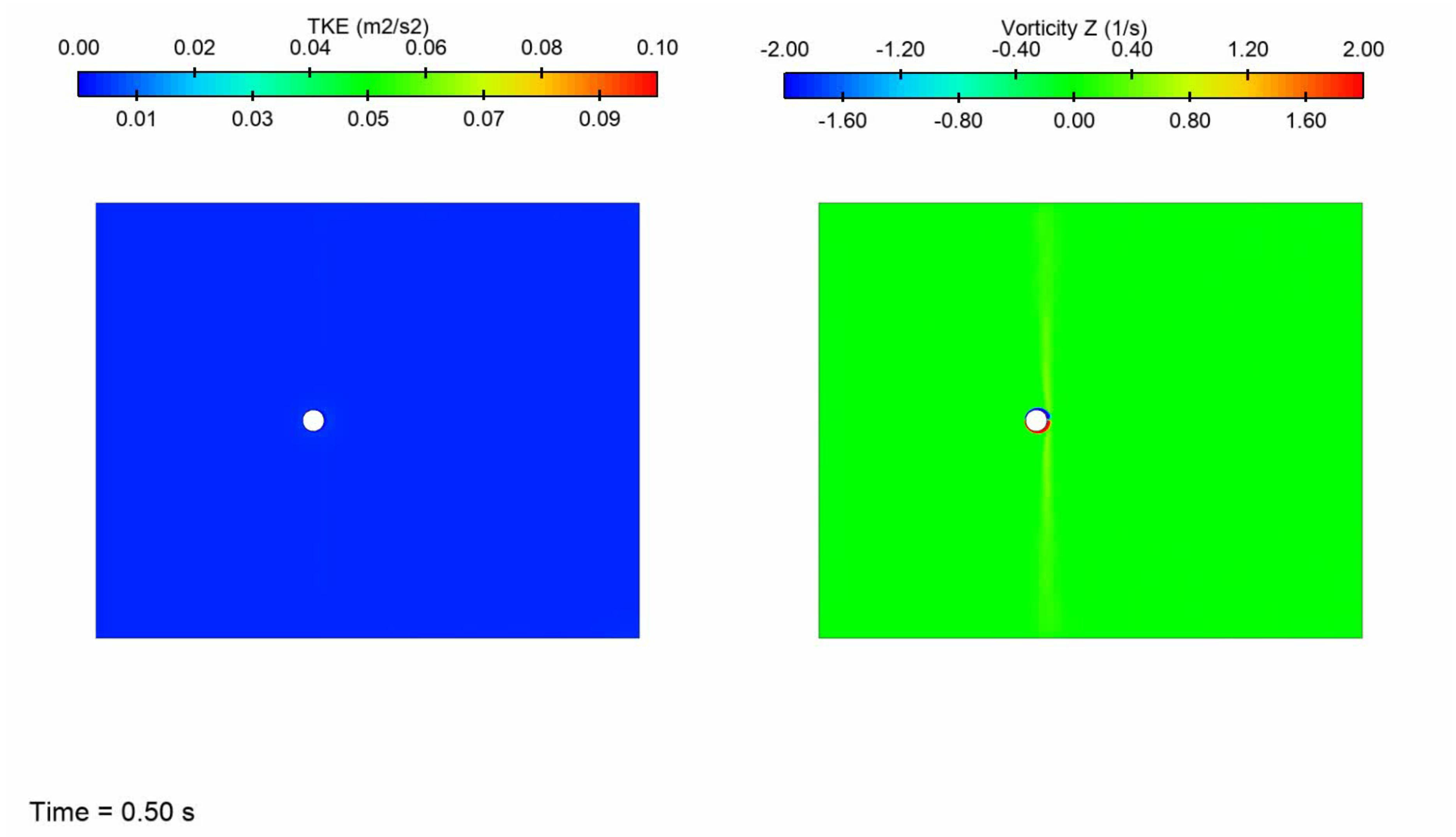


In all cases the Reynolds number in the unsteady and turbulent regime.

Left: velocity magnitude contours – **Right:** relative pressure contours.

www.wolfdynamics.com/training/turbulence/movies1/mov1.avi

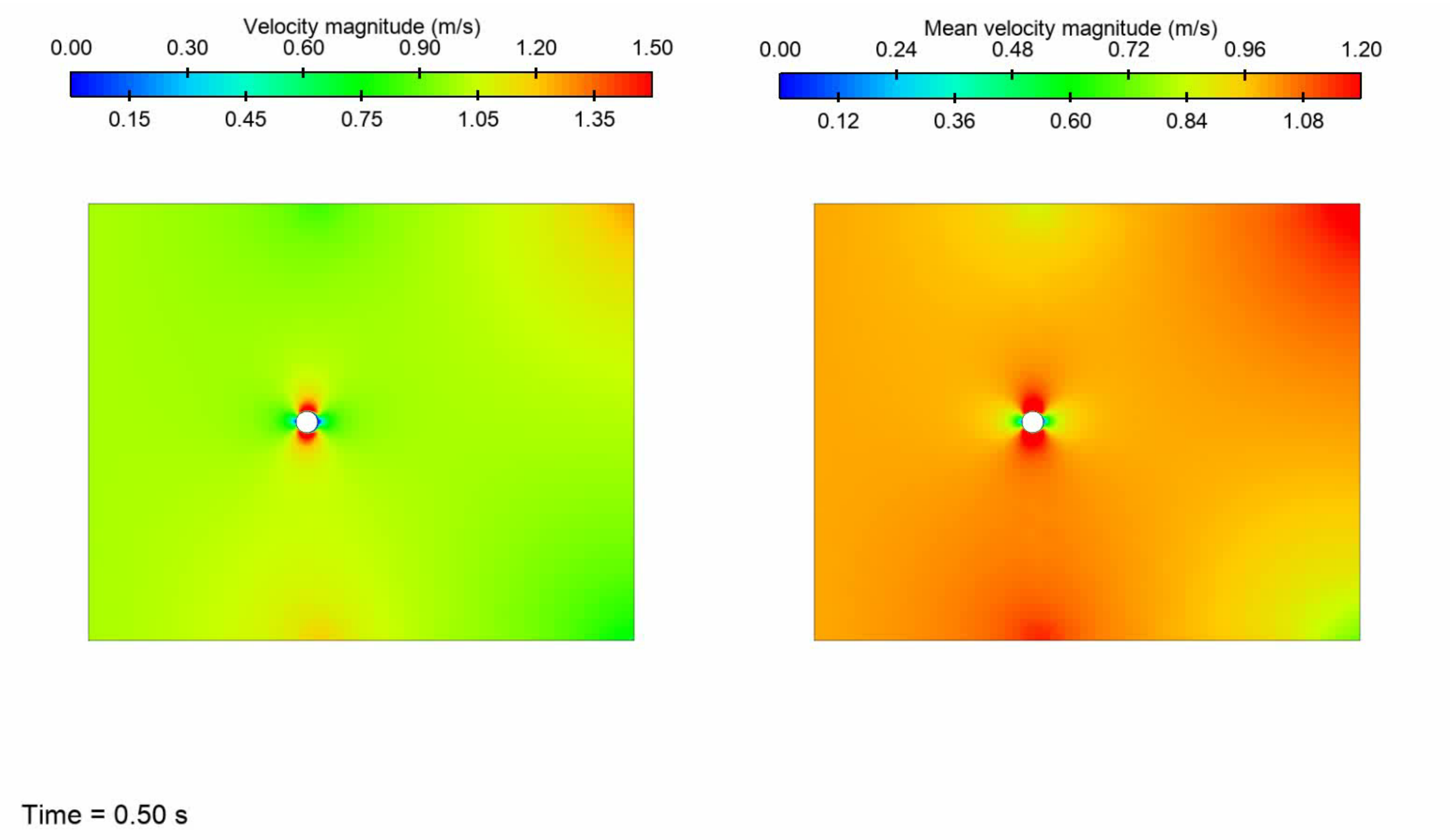
Qualitative results – Contour plots



Left: turbulent kinetic energy contours – **Right:** vorticity Z contours (component normal to the screen).

www.wolfdynamics.com/training/turbulence/movies1/mov2.avi

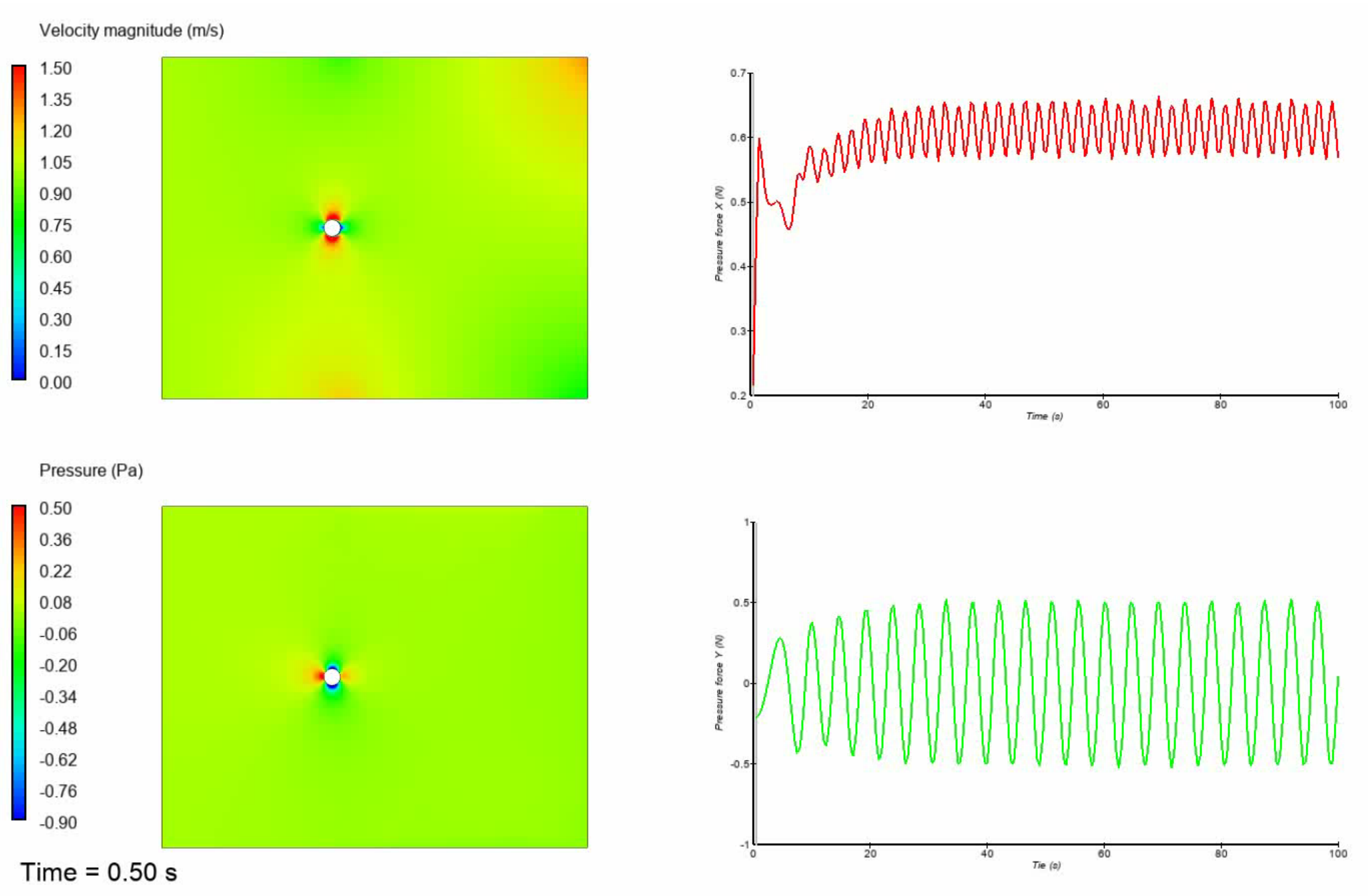
Qualitative results – Contour plots



Left: velocity magnitude contours – **Right:** mean velocity magnitude contours

www.wolfdynamics.com/training/turbulence/movies1/mov3.avi

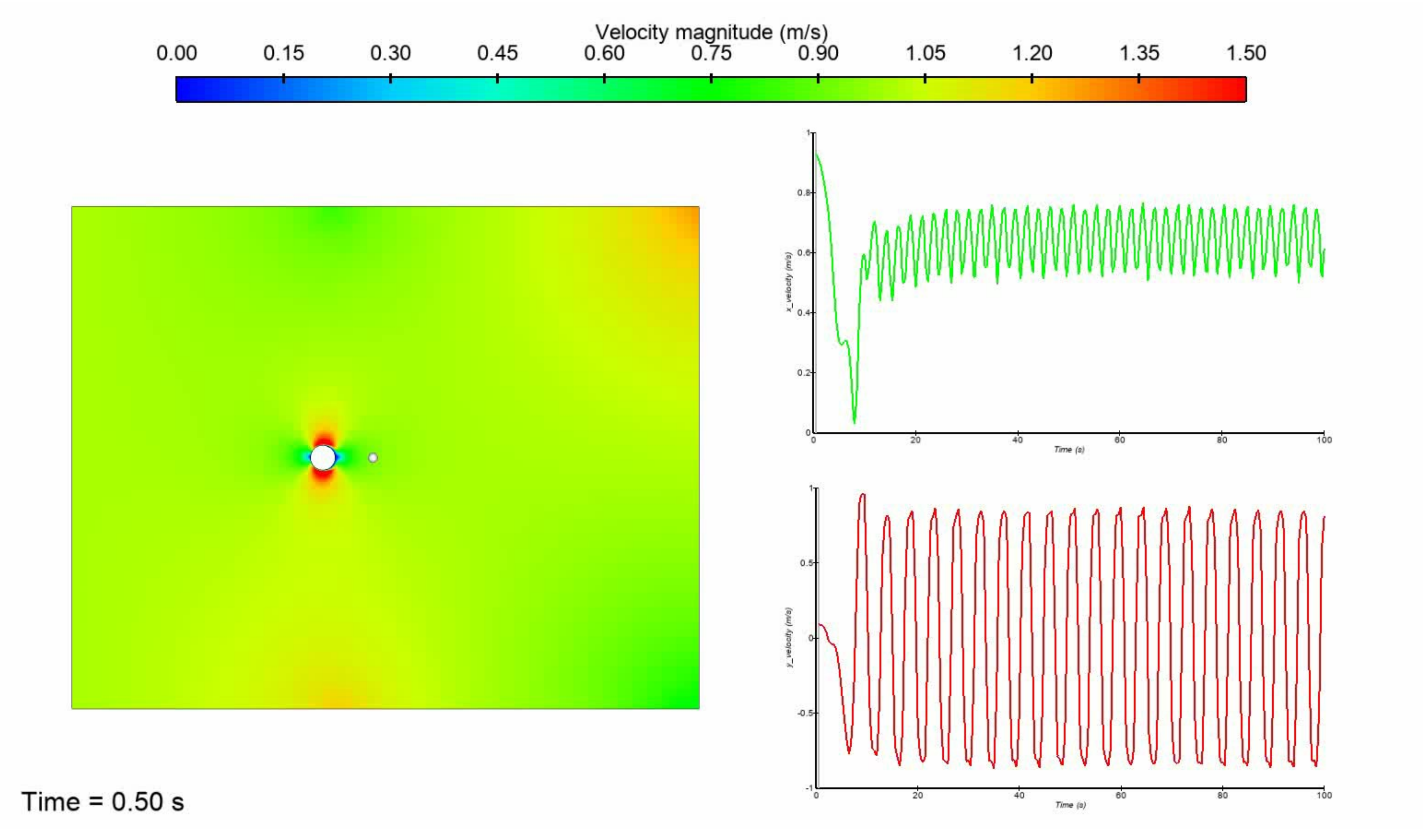
Qualitative results – Contour plots



Left: velocity magnitude and pressure contours – **Right:** Components X and Y of pressure force over the cylinder

www.wolfdynamics.com/training/turbulence/movies1/mov4.avi

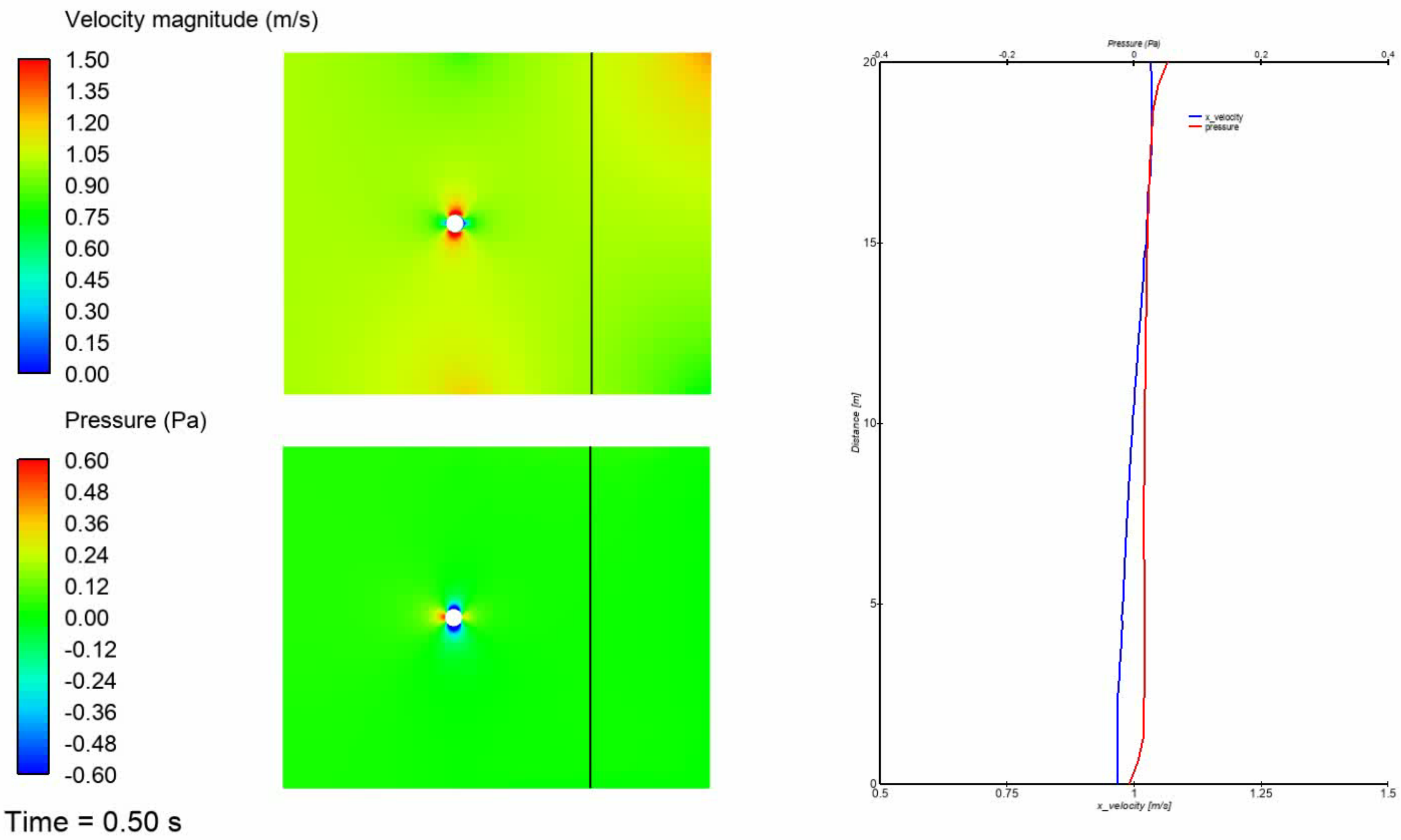
Qualitative results – Contour plots



Left: velocity magnitude contours – **Right:** velocity sampled at a location in the wake of the cylinder

www.wolfdynamics.com/training/turbulence/movies1/mov5.avi

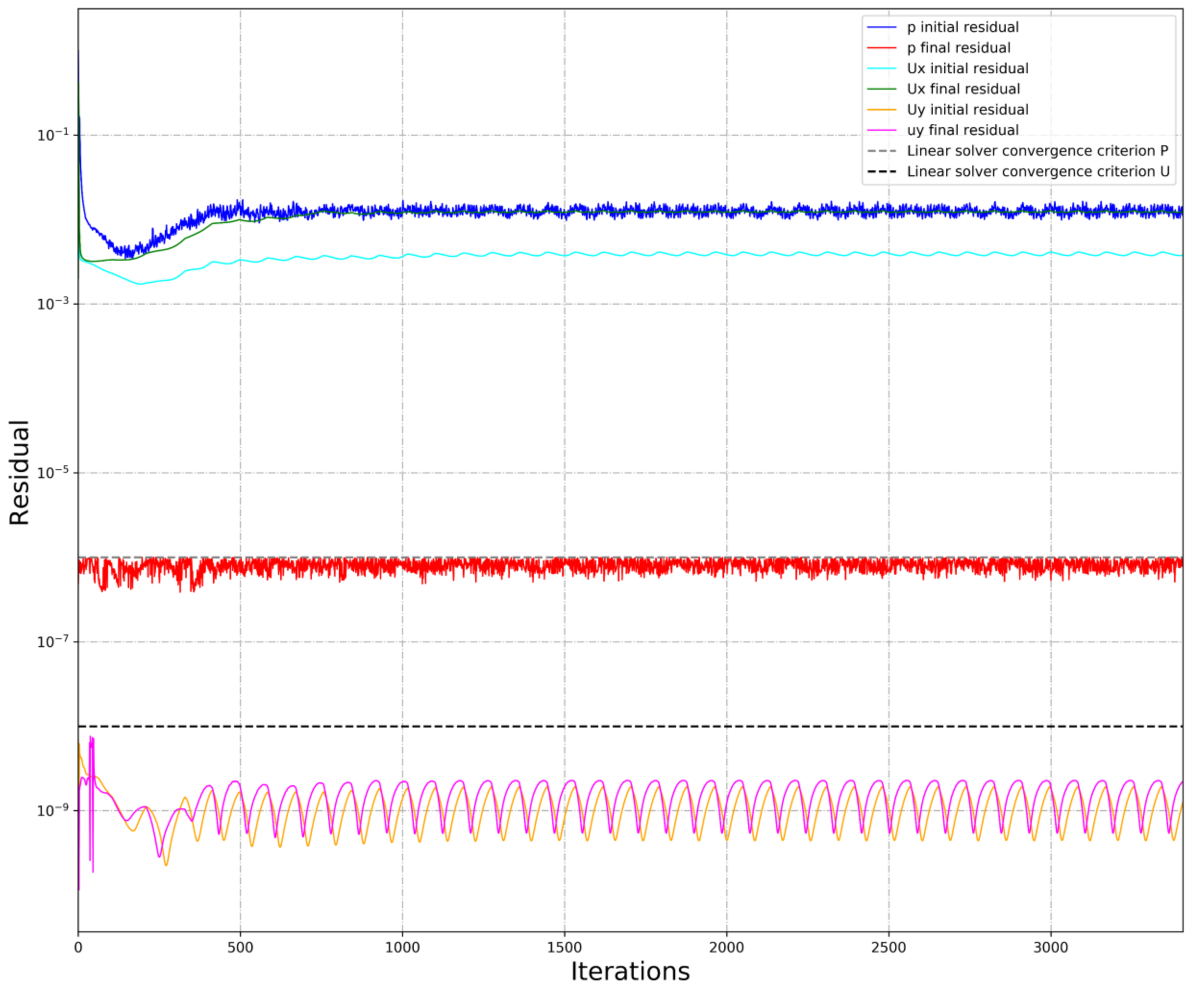
Qualitative results – Contour plots

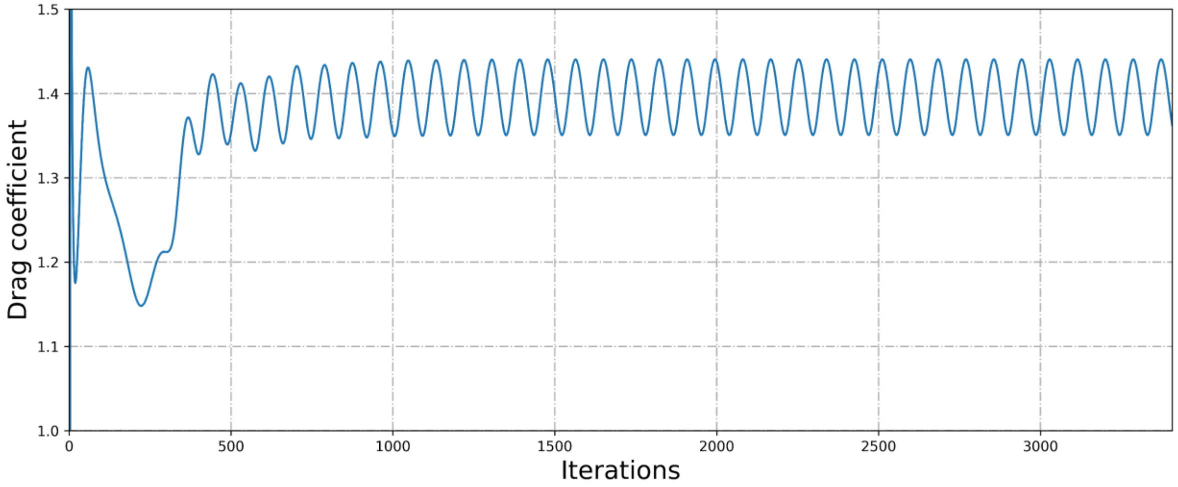


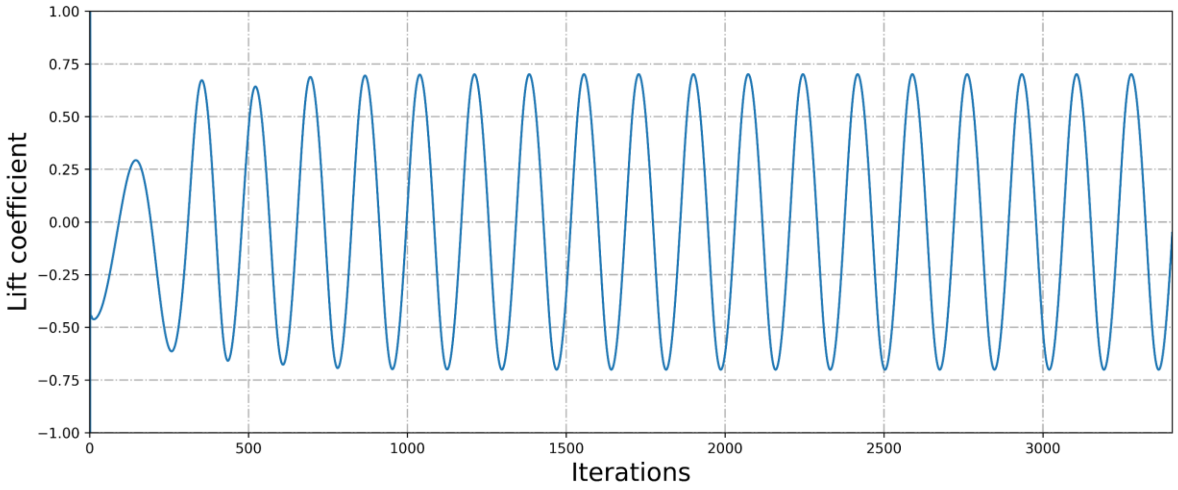
Left: velocity magnitude contours – **Right:** velocity and pressure sample in a vertical line in the wake of the cylinder

www.wolfdynamics.com/training/turbulence/movies1/mov7.avi

Quantitative results – Residuals – Lift and drag time series



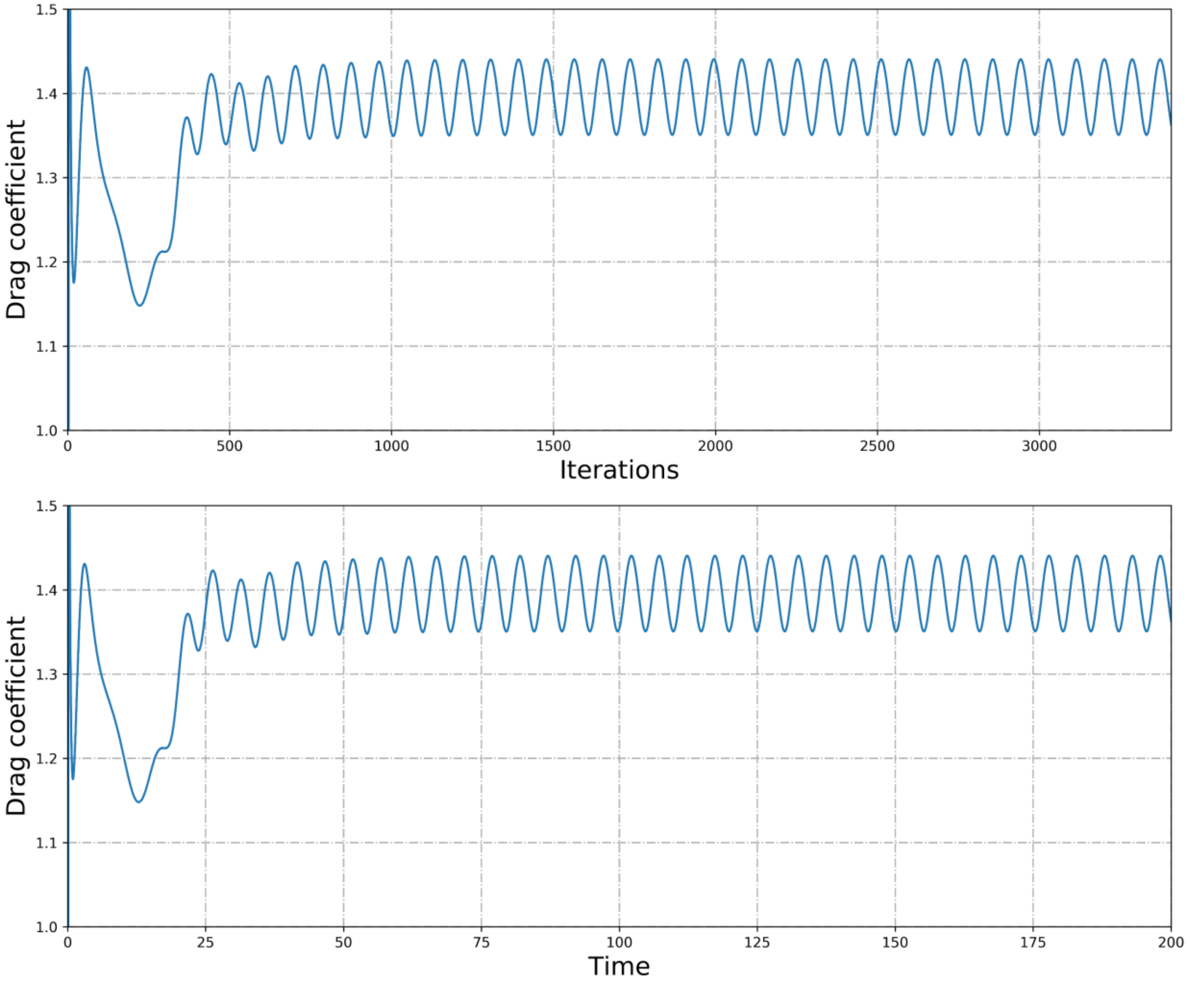




Unsteady solver – Comparison of iterative convergence and iterative QoI

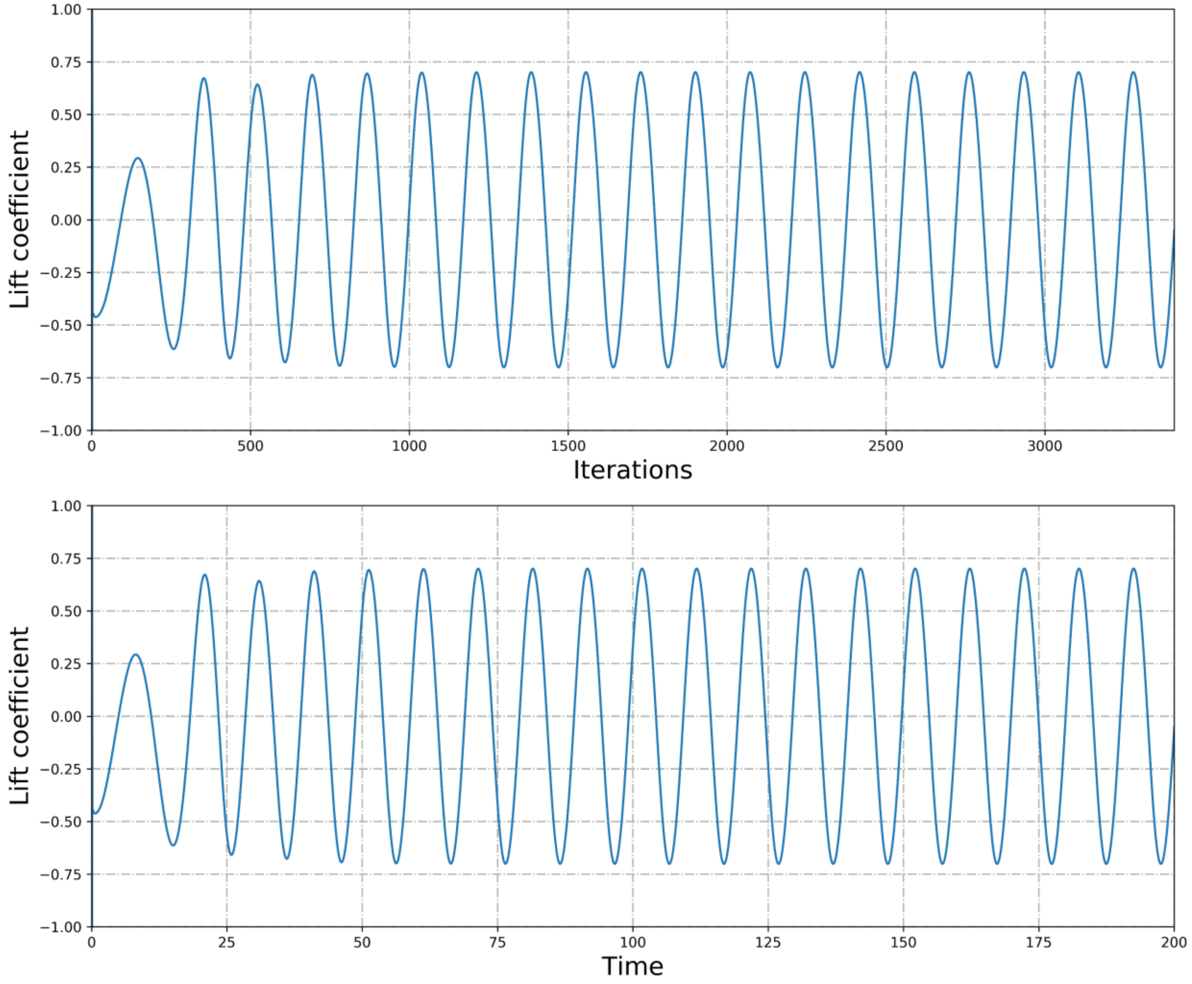
Lift coefficient and drag coefficient signals

Quantitative results – Lift and drag time series



Unsteady solver QoI – Drag coefficient

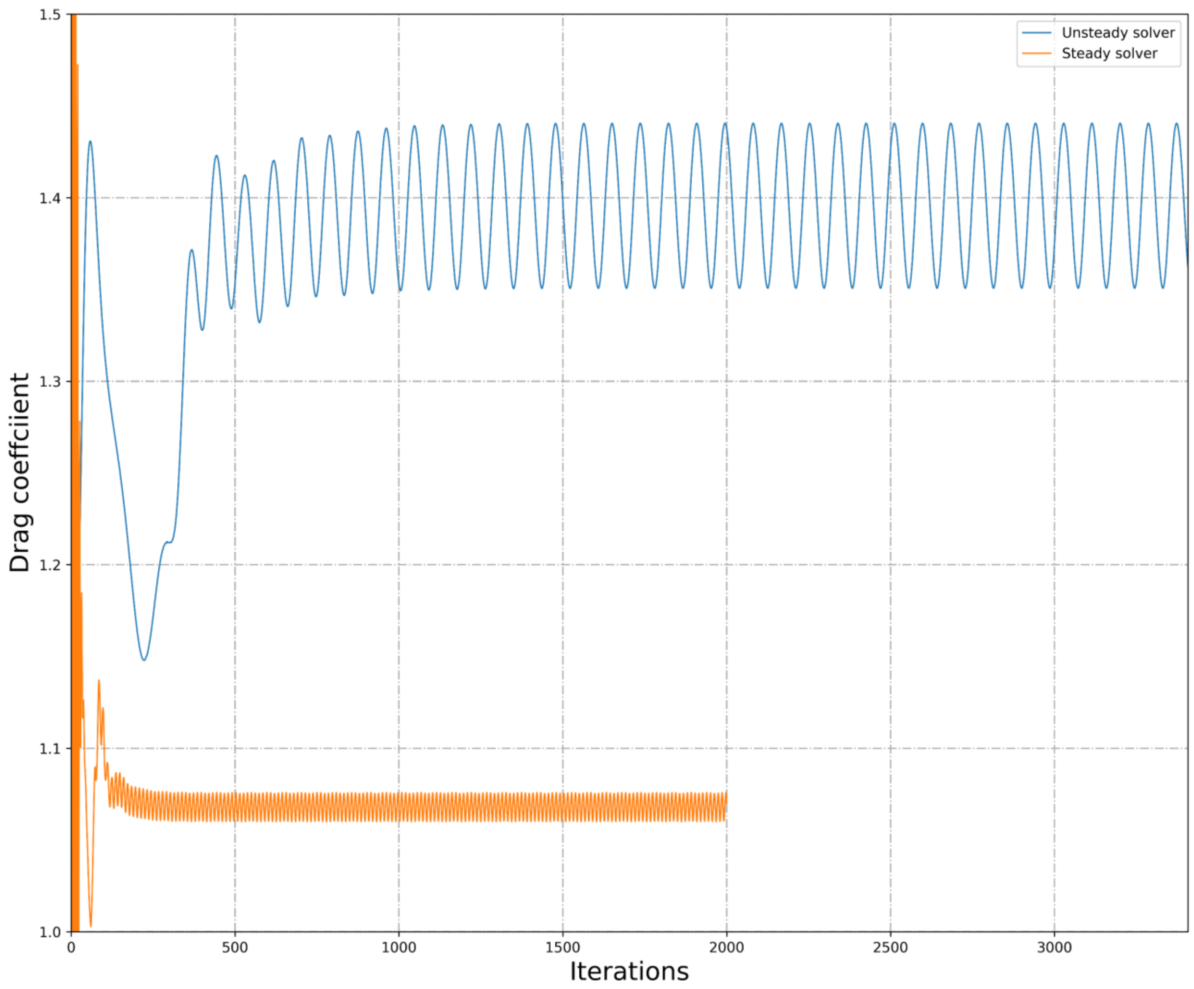
Top image: in function of iterations
Bottom image: in function of time



Unsteady solver QoI – Lift coefficient

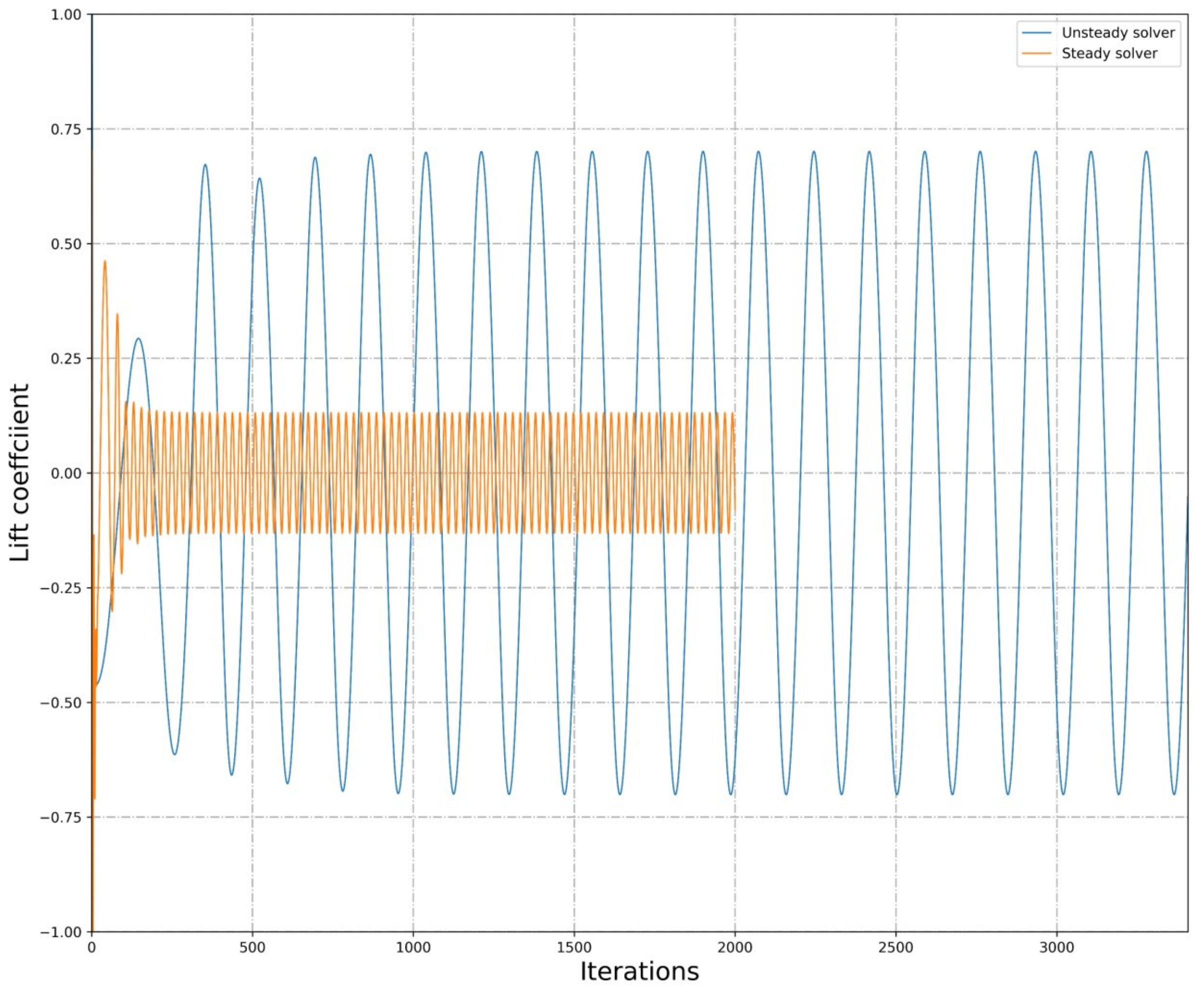
Top image: in function of iterations
Bottom image: in function of time

Quantitative results – Lift and drag time series



Unsteady solver and **steady solver** comparison

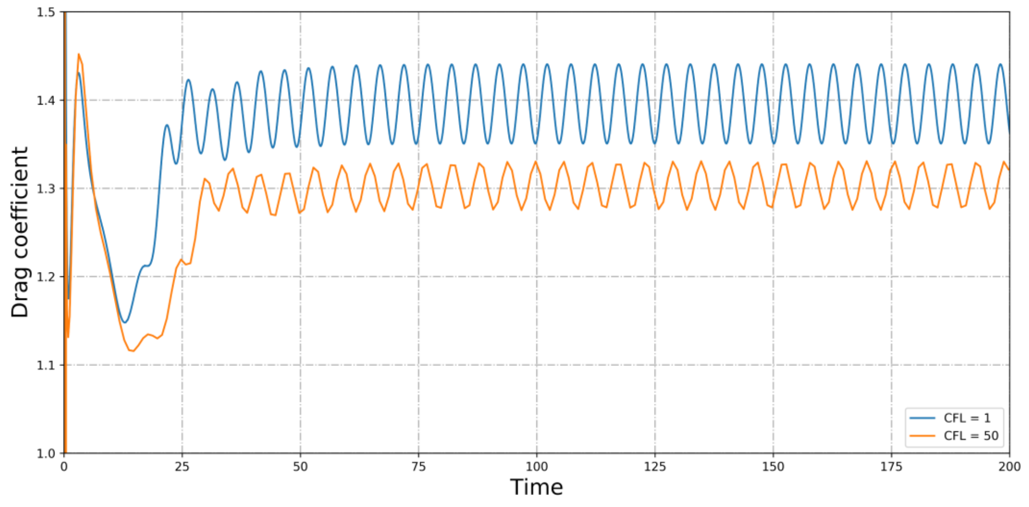
Comparison of the drag coefficient outcome



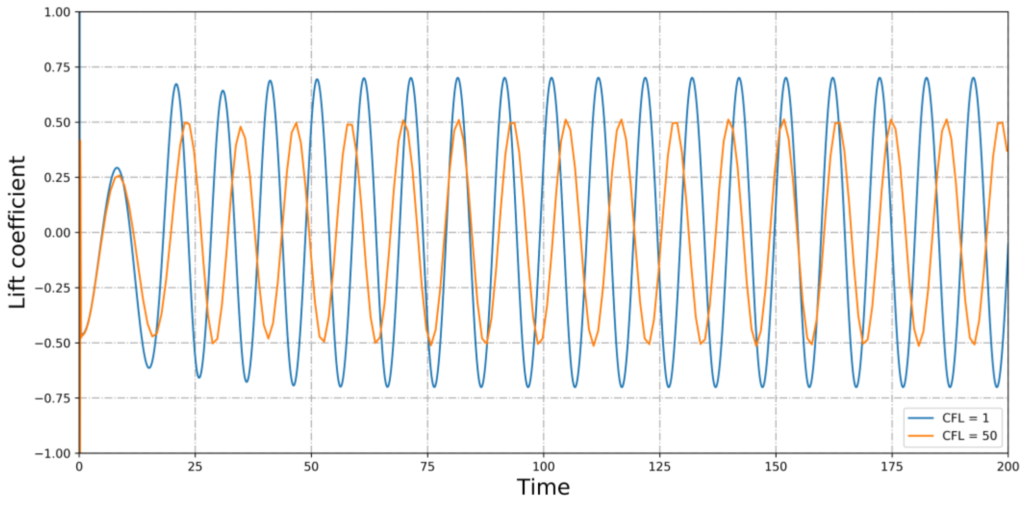
Unsteady solver and **steady solver** comparison

Comparison of the lift coefficient outcome

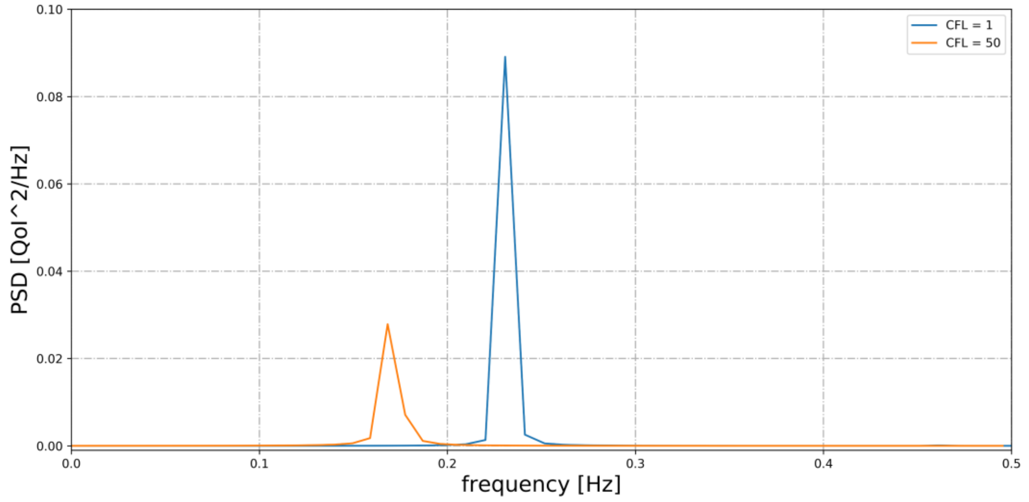
Qualitative results – Contour plots



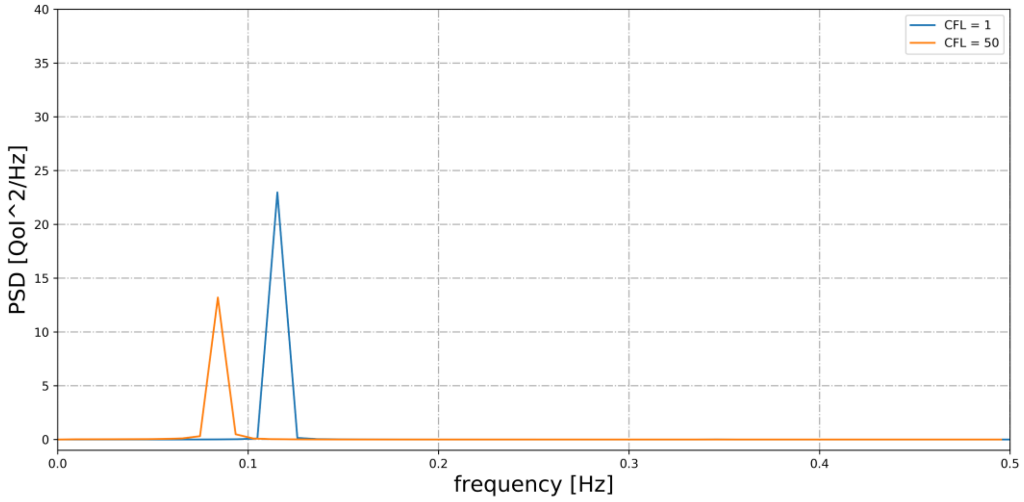
Drag coefficient time series
CFL = 1 and CFL = 50



Lift coefficient time series
CFL = 1 and CFL = 50

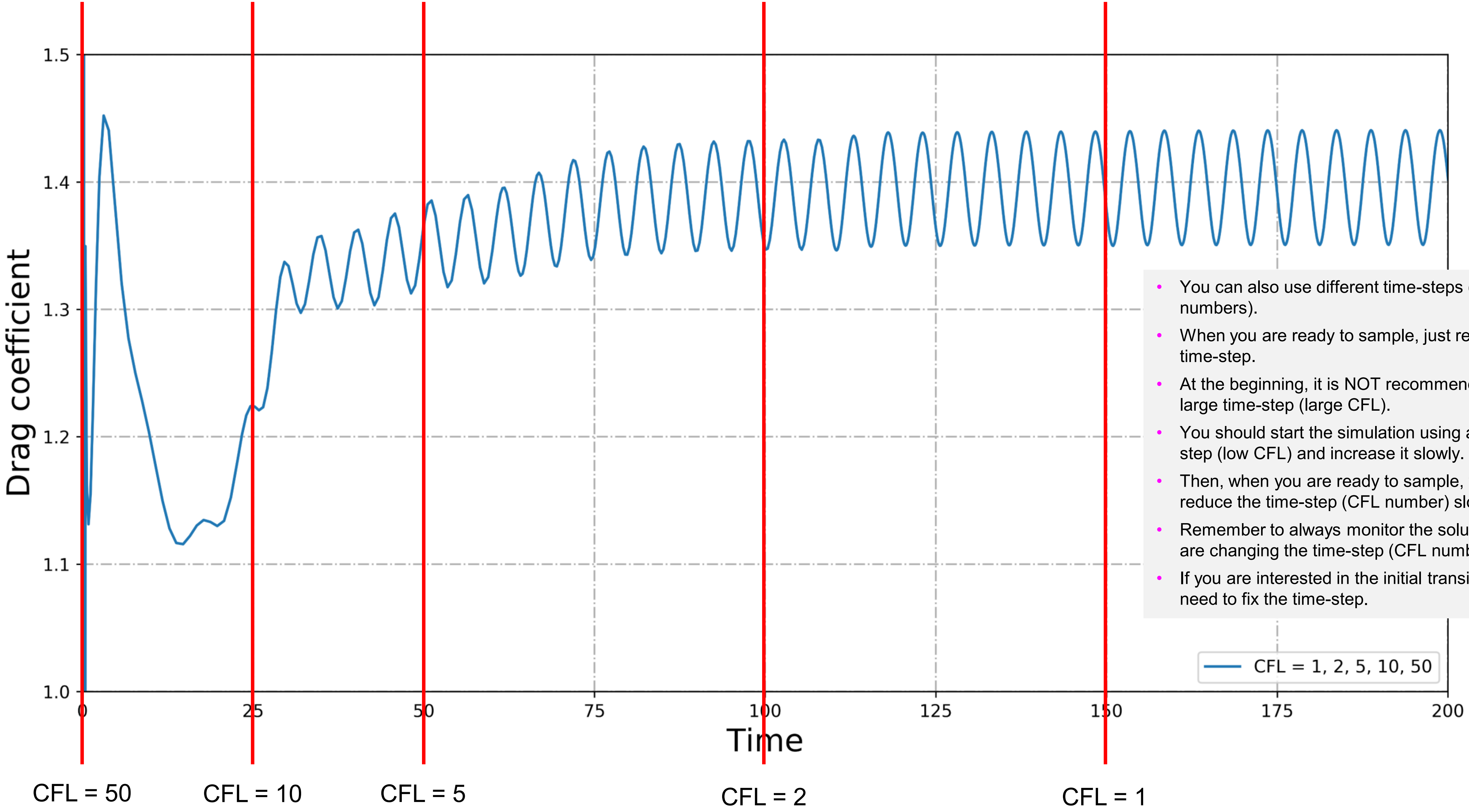


Periodogram of drag coefficient time series
CFL = 1 and CFL = 50



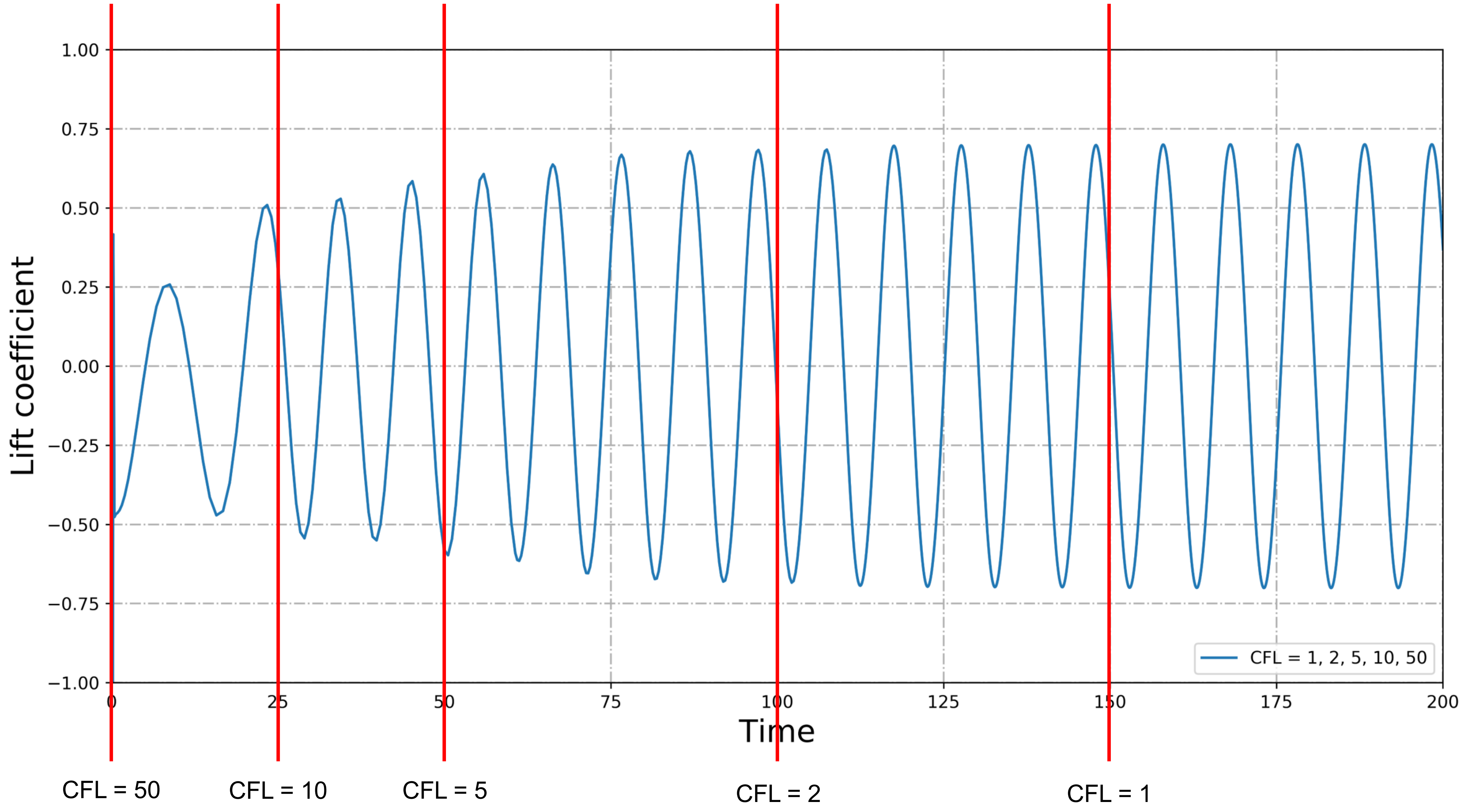
Periodogram of lift coefficient time series
CFL = 1 and CFL = 50

Qualitative results – Contour plots



- You can also use different time-steps (CFL numbers).
- When you are ready to sample, just reduce the time-step.
- At the beginning, it is NOT recommended to use a large time-step (large CFL).
- You should start the simulation using a small time-step (low CFL) and increase it slowly.
- Then, when you are ready to sample, you can reduce the time-step (CFL number) slowly.
- Remember to always monitor the solution while you are changing the time-step (CFL number).
- If you are interested in the initial transient, you will need to fix the time-step.

Qualitative results – Contour plots



On the CFL number

- First of all, what is the **CFL number** or **Courant number**?
- In one dimension, the **CFL number** is defined as,

$$CFL = \frac{u \ \Delta t}{\Delta x}$$

- The **CFL number** is a measure of how much information (u) traverses a computational grid cell (Δx) in a given time-step (Δt).

On the CFL number

- Similarly, we can define the **CFL number condition**, which is related to the **CFL number**.
- For the **N** dimensional case, the **CFL number condition** becomes,

$$CFL = \Delta t \sum_{i=1}^{n} \frac{u_i}{\Delta x_i} \leq CFL_{max}$$

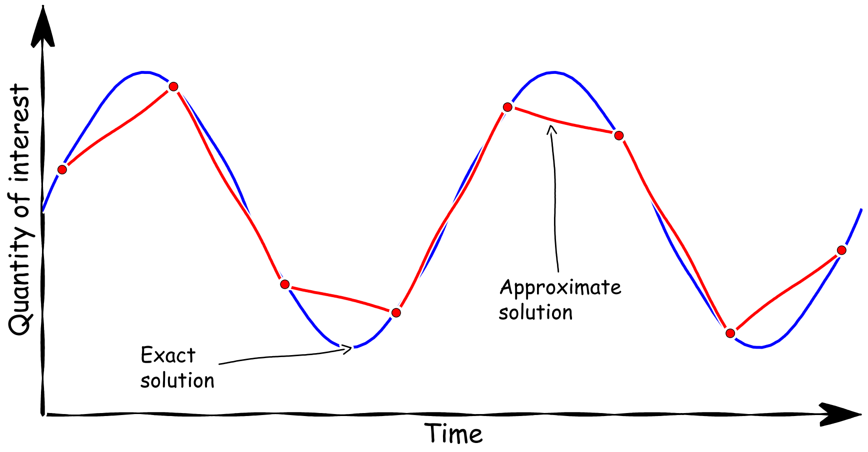
Maximum CFL number allowed by the numerical method

On the CFL number

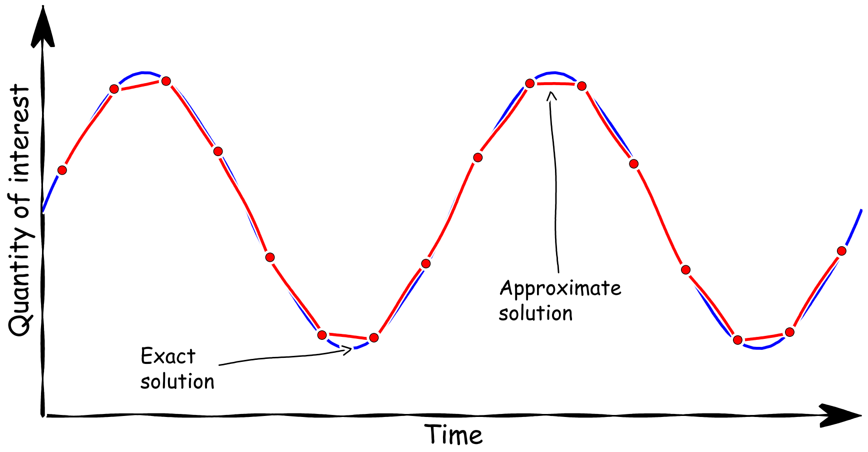
- The **CFL number** is a necessary condition to guarantee the stability of the numerical scheme.
- But not all numerical schemes have the same stability requirements or CFL_{max} requirement (maximum allowable **CFL number**).
- We are going to use implicit numerical methods which are **unconditionally stable**.
- In other words, they are not constrained to the **CFL number condition** or maximum allowable **CFL number**.
- However, the fact that we are going to use a numerical method that is unconditionally stable, does not mean that we can use a time step of any size.
- The time-step must be chosen in such a way that it resolves the time-dependent features, and it maintains the solver stability.

On the CFL number

- When running unsteady simulations, the time-step must be chosen in such a way that it resolves the time-dependent features and maintains solver stability.



When you use large time steps you do not resolve well the physics



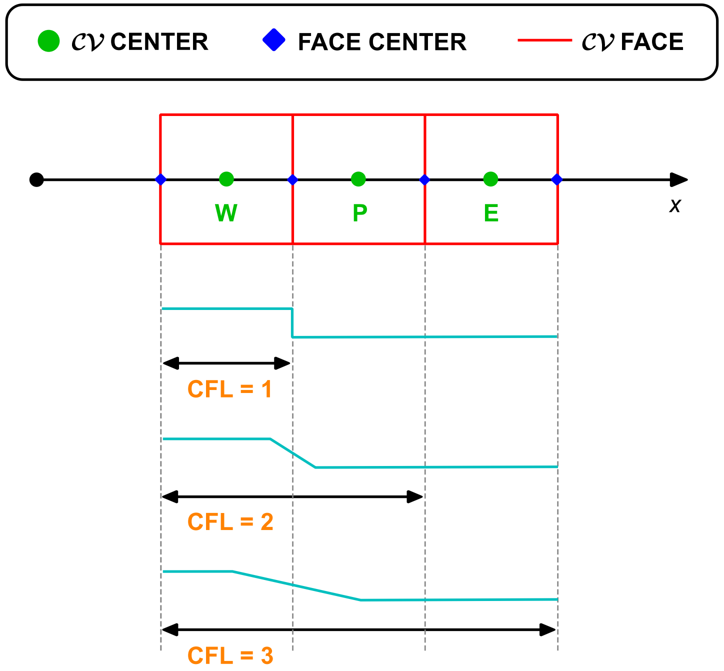
By using a smaller time step you resolve better the physics, and you gain stability

On the CFL number

- I like to see the **CFL number** as follows,

$$CFL = \frac{u\,\Delta t}{\Delta x} = \frac{u}{\Delta x/\Delta t} = \frac{\text{speed of the PDE}}{\text{speed of the mesh}}$$

- It is an indication of the amount of information that propagates through one cell (or many cells), in one time-step.



- The **CFL condition** is a necessary condition for stability (and hence convergence).
- But it is not always sufficient to guarantee stability.
- Other properties of the discretization schemes that you should observe are: conservationess, boundedness, transportiveness, and accuracy.
- The **CFL number** is not a magical number.