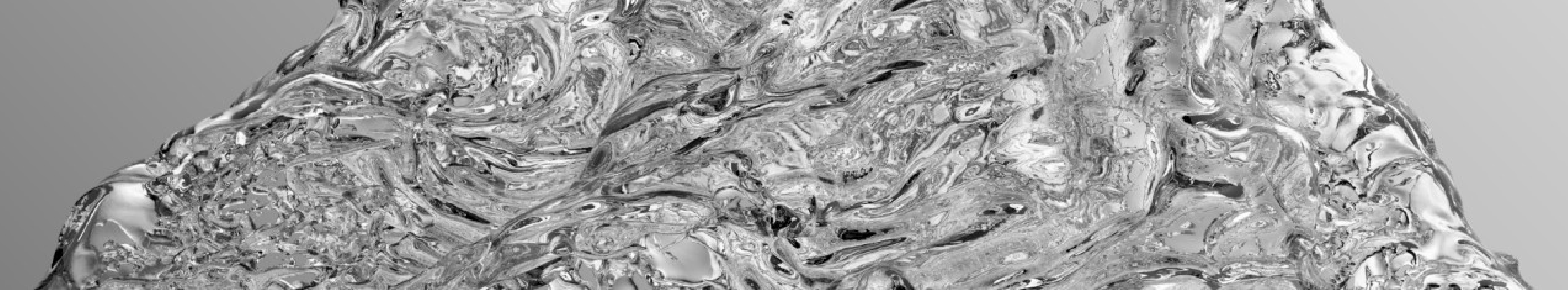


EEN-E2001 Computational Fluid Dynamics

Lecture 5: Fluid physical phenomena

Prof. Ville Vuorinen

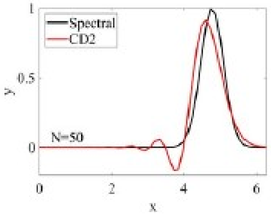
February 13th 2023

Aalto University, School of Engineering

Lecture 1: Linear PDEs and finite difference method

$$\frac{\partial T}{\partial t}+\nabla\cdot T\boldsymbol{u}=\alpha\nabla^2 T$$

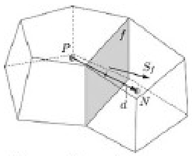
$$\frac{\partial T}{\partial x}\approx\frac{T_{i+1}-T_{i-1}}{2\Delta x}$$



Lecture 2: Gauss' theorem and finite volume method

$$\int_\Omega \nabla\cdot(T\boldsymbol{u})\,d\Omega=\int_{\partial\Omega}(T\boldsymbol{u})\cdot\boldsymbol{n}\,dS$$

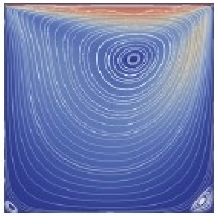
$$\int_{\partial\Omega}\boldsymbol{u}\cdot\boldsymbol{n}\,dS\approx\Sigma_f\boldsymbol{u}_f\cdot\boldsymbol{n}_f\,dS_f$$



Lecture 3: Navier-Stokes equation and pressure

$$\frac{\partial\boldsymbol{u}}{\partial t}+\nabla\cdot\boldsymbol{u}\boldsymbol{u}=-\nabla p+\nu\nabla^2\boldsymbol{u}$$

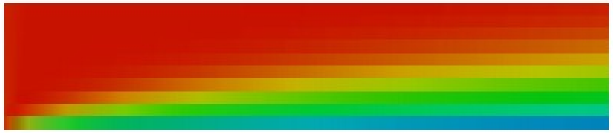
$$-\nabla^2 p=\nabla\cdot\nabla\cdot\boldsymbol{u}\boldsymbol{u}$$

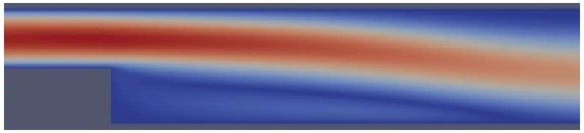


Lecture 4: OpenFOAM code and structure

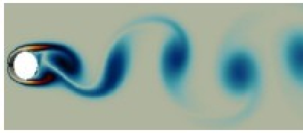
```
fvVectorMatrix UEqn
(
      fvm::ddt(U)
    + fvm::div(phi, U)
    - fvm::laplacian(nu, U)
);
```

Lecture 5: Simulating fluid physical phenomena: part A





Lecture 6: Simulating fluid physical phenomena: part B



Intended learning objectives of the lecture

After the lecture the student:

- Can name basic fluid physical phenomena and connect them to HW3-HW5.

CFD simulation and PDE solution includes at least the following aspects covered on the course

1) **Physics** identification.

2) **Mathematical equations and physics interpretation.** Boundary/initial conditions.

3) **Objectives, feasibility, and time-constraints**.

4) **Numerical method and modeling assumptions**.

5) **Geometry and mesh generation**.

6) **Computing** i.e. running simulation.

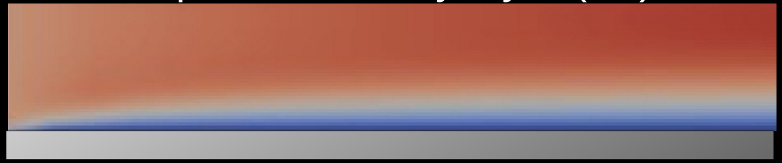
7) **Visualization and post-processing**.

8) **Validation and verification, reference data**. Reporting, analysis and discussion of the results. Are the results sane?

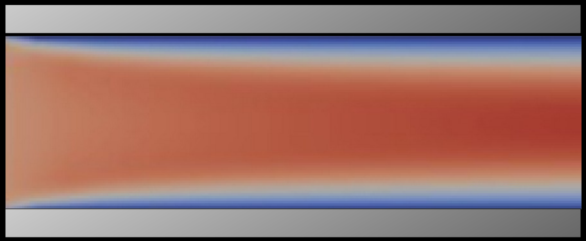
Examples on physics phenomena in fluid dynamics

Steady, laminar flow: **Key physics:** diffusion builds up the BL structure

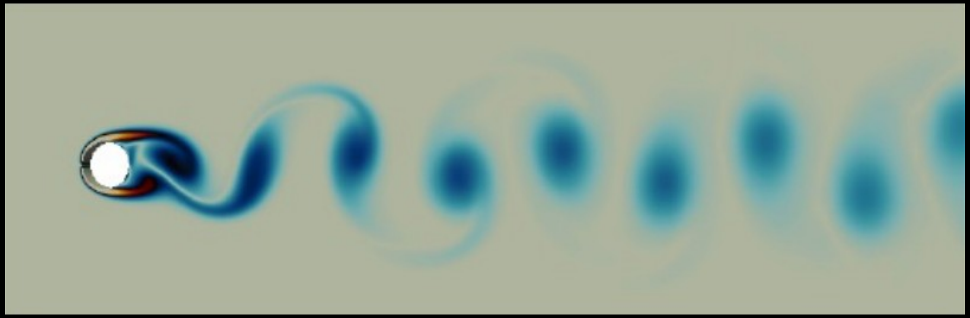
Free flat plate boundary layer (BL)



Channel flow

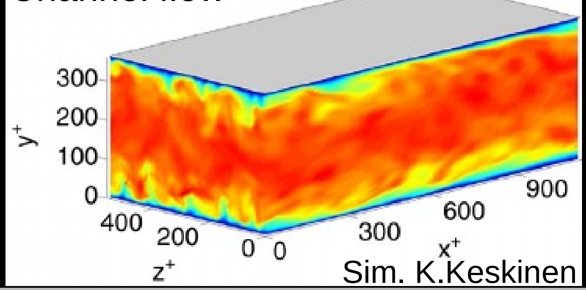


Unsteady, laminar flow. **Key physics**: von Karman instability and vortex shedding



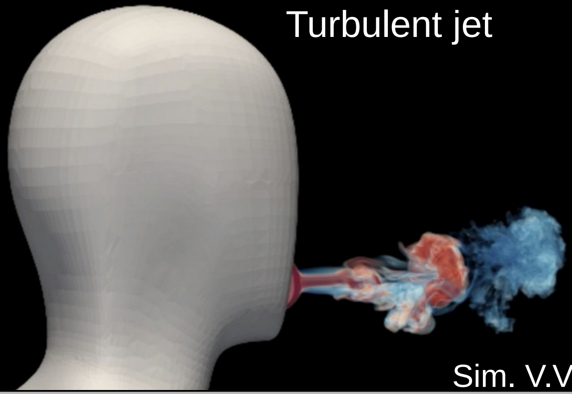
Unsteady, turbulent flow. **Keyword:** turbulence production at walls and shear layers

Channel flow



Sim. K.Keskinen

Turbulent jet

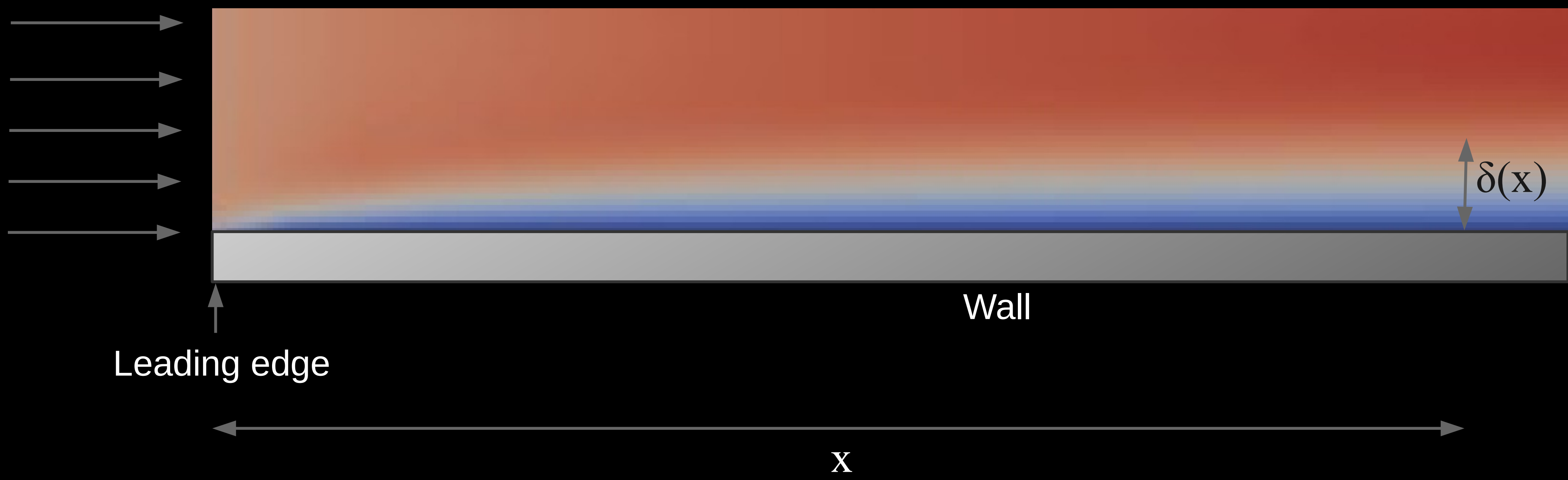


Sim. V.Vuorinen

Free boundary layer

Relevance: HW3

Free boundary layer (relevance: HW3)



Recall: free boundary layer thickness grows as

$$\delta \approx 5.0\sqrt{\frac{\nu x}{U_\infty}} = 5.0\, x / \sqrt{Re_x}$$

$$Re_x = \frac{U_\infty x}{\nu}$$

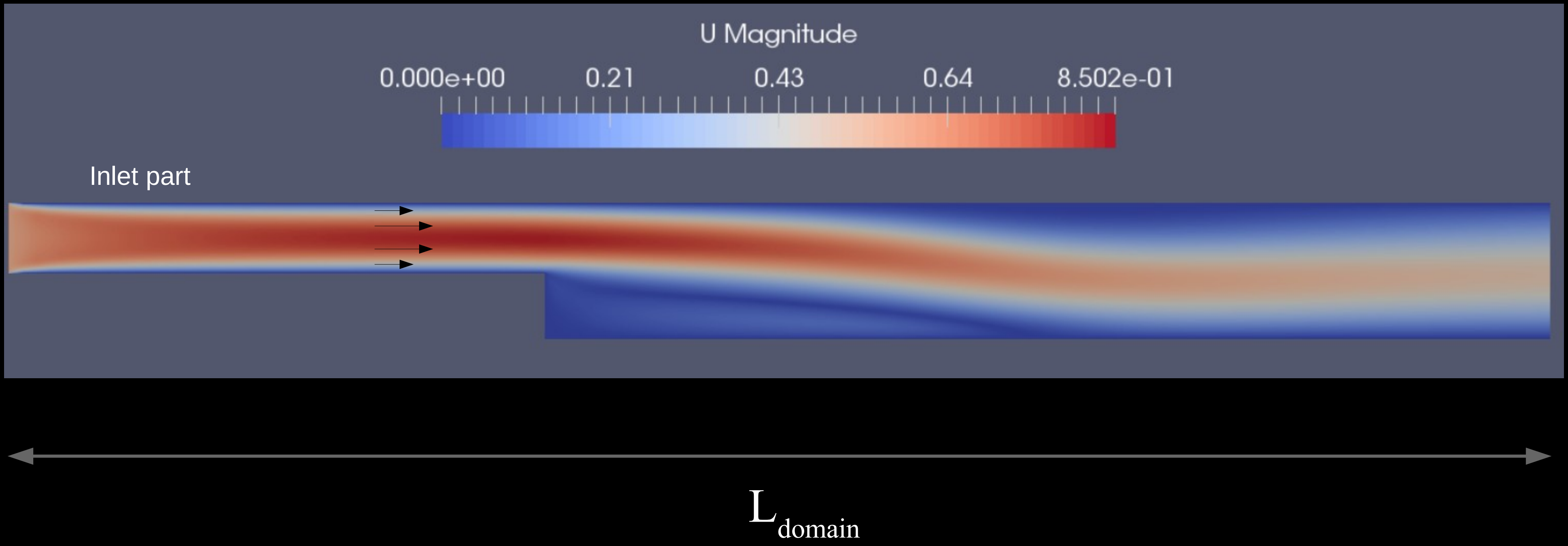
Backward facing step flow

Relevance: HW3-HW4

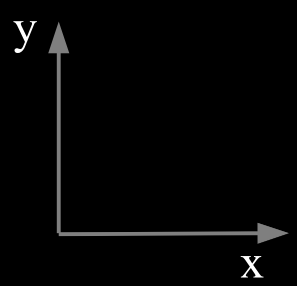
Real world “Backward facing step”

By Ralf Roletschek (talk) - Fahrradtechnik auf fahrradmonteur.de - Own work, FAL, https://commons.wikimedia.org/w/index.php?curid=15

Backward facing step



Backward facing step

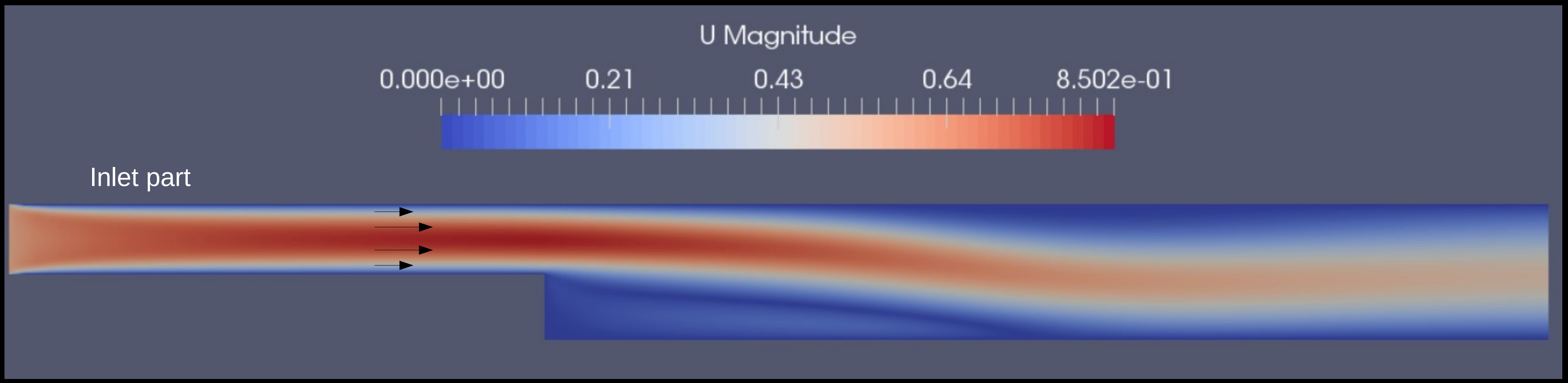


Reynolds number:

$$Re = \frac{UD}{\nu}$$

Viscous entry length after which laminar channel flow fully developed:

$$L/D = 0.05\,Re$$



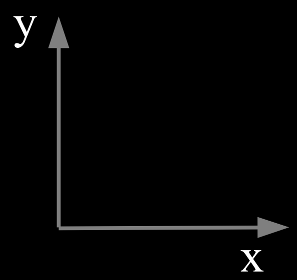
1 2 3

Comment 1: constant velocity at inlet.

Comment 2: laminar boundary layer growth by viscous diffusion.

Comment 3: after viscous entry length, expect close to parabolic velocity profile $U=U(y)$.

Backward facing step

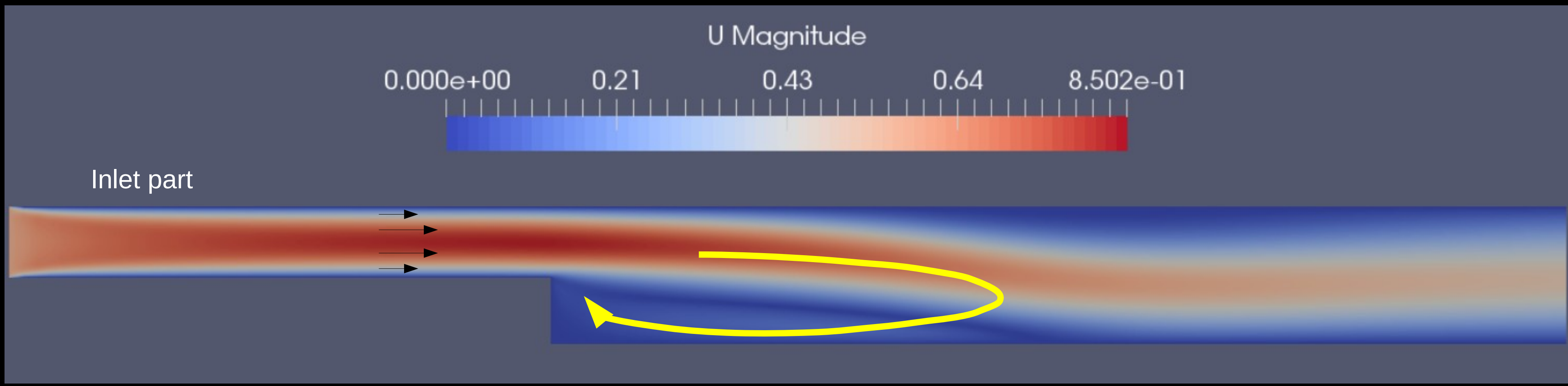


Reynolds number:

$$\mathrm{Re} = \frac{UD}{\nu}$$

Viscous entry length after which laminar channel flow fully developed:

$$L/D = 0.05\,\mathrm{Re}$$



4 5 6

Comment 4: flow recirculation zone (negative x-velocity).

Comment 5: stagnation point (x-velocity=0)

Comment 6: outlet (modeled commonly using `zeroGradient or InletOutlet` bc in OpenFOAM)

Important considerations

“Resolve”:
Space: put enough grid points to resolve the length scales
(we can not take photos with 5*5 pixels).
Time: make timestep small enough. (Courant number, CFL number).

Resolve all the geometrical length scales:
channel height, step height, channel length, main chamber length/height.

Identify physics and refine grid accordingly:
- Laminar or turbulent.
- Turbulent flow: stricter space resolution requirement to resolve vortices of different size.
- Boundary layer thickness and near wall resolution (y+ value) → necessary to understand in order to capture the wall shear stress (friction).
- Is there unsteady behavior e.g. vortex shedding? What is the oscillation frequency?

How long time should we simulate?
→ Depends on the case but rough guidelines to start with:
- Identify vortex turnaround time scale and simulate “at least tens of those time scales”
- Identify “flow through time scale” and simulate “at least a few of them”

Vortex turn-around time scale

$$\tau_{vortex} = \frac{d_{vortex}}{U_{vortex}}$$

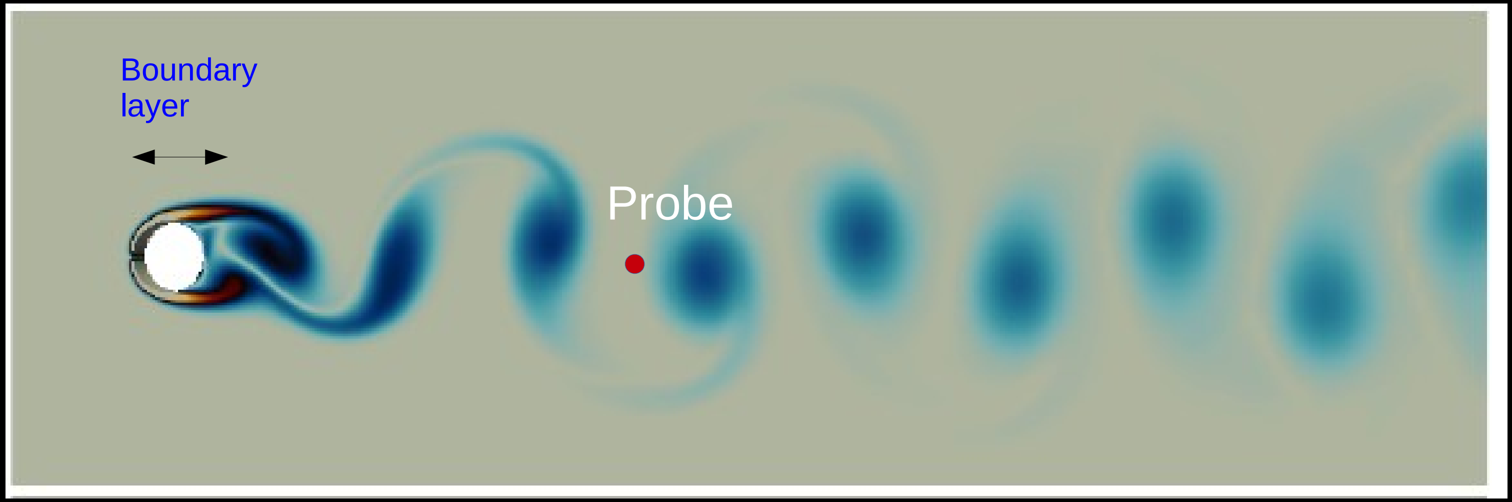
Flow through time scale

$$\tau_{flow} = \frac{L_{domain}}{U}$$

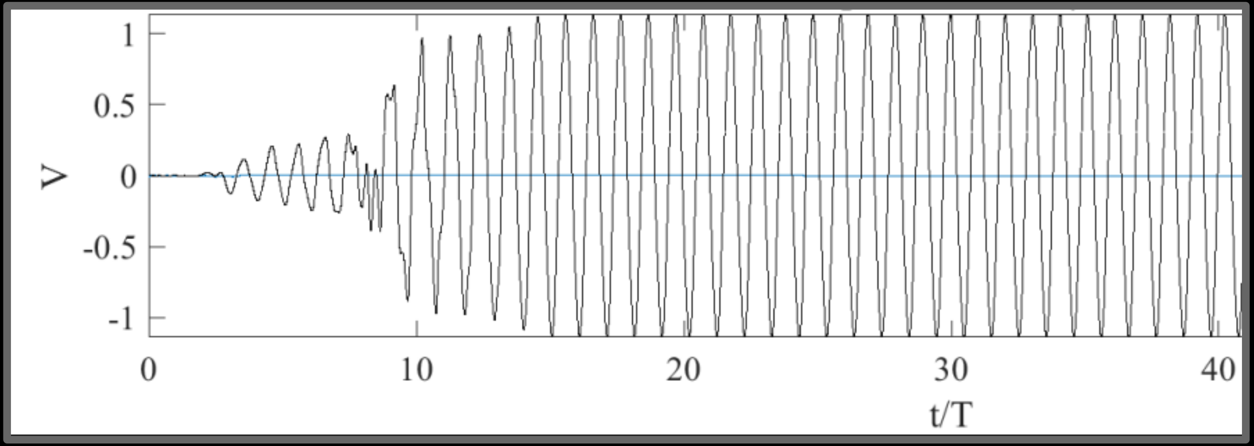
Flow over a cylinder

Relevance: HW5

Flow over a cylinder

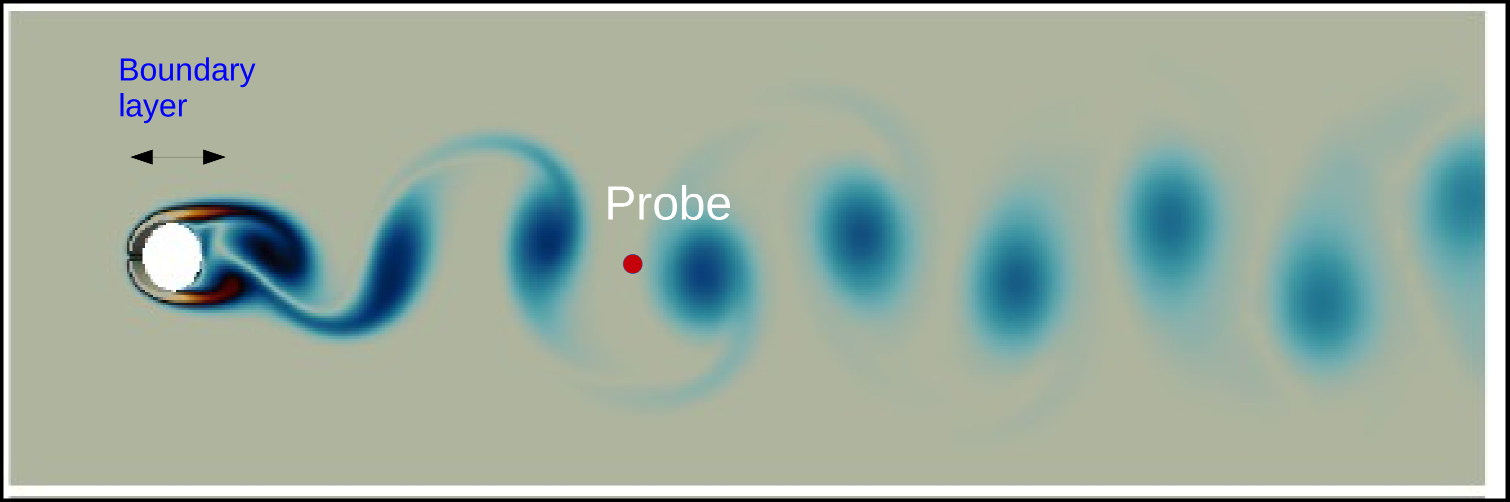


Probe

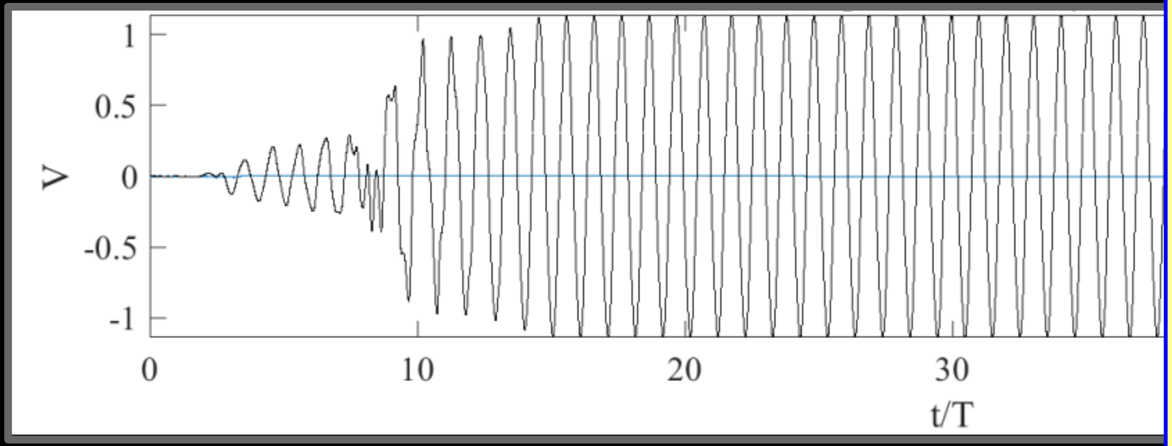


Typical scenario:
After initial transients flow enters a periodic shedding mode with a distinct frequency (Strouhal number) observed from the spanwise velocity component by downstream probing or lift-coefficient variation.

Flow over a cylinder



Probe



Very, very important in numerics to minimize numerical diffusion (but typically it is not possible to get fully rid of it). Pay attention to:

- Discretize convection term **`div(phi,U)`** preferably with "**`Gauss linear`**" (CD2)
- Second best option TVD/NVD limiter: e.g. **`GammaV 0.1`**
- Discretize time with "**`backward`**" (2^{nd} order)

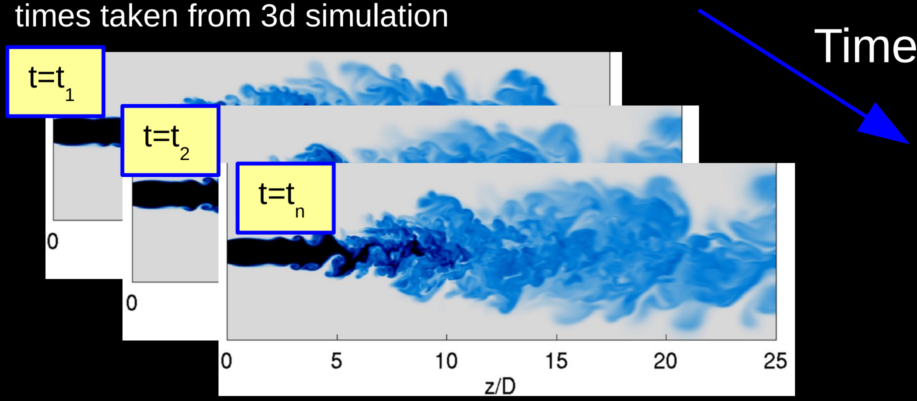
Turbulent flow

Relevance: Important to know later on but not course content.

Turbulent jets

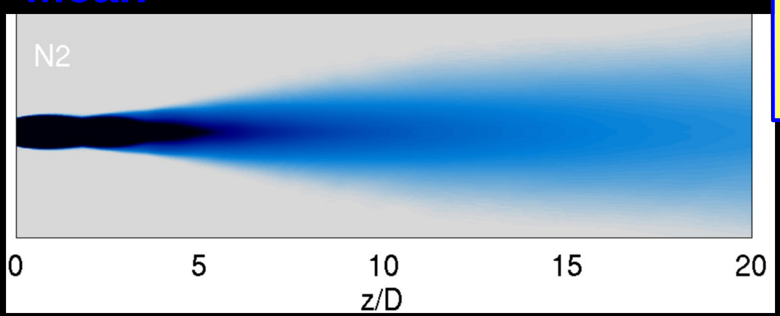
Below: we inject a concentration field in the jet flow to study mixing.

Example: 2d cutplanes from different times taken from 3d simulation



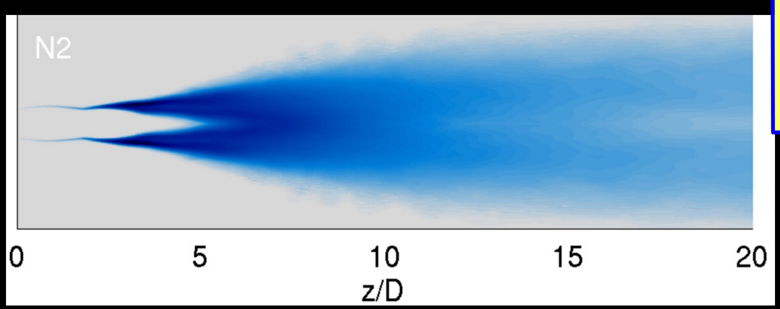
Here: passive scalar field averaged in time over several thousand cutplanes (time snapshots)

Mean



$$<P> = \frac{1}{N} \Sigma_{i=1}^{N} P_i$$

Standard Deviation



$$\sigma_P = \frac{\sqrt{\Sigma_{i=1}^{N} (P_i - <P>)^2}}{\sqrt{N}}$$

Turbulent channel flows

Here is a Matlab code to do turbulent channel flow. OpenFOAM also has relevant tutorials.
https://www.sciencedirect.com/science/article/abs/pii/S0010465516300388

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j^2} + g_i$$

Domain size

$2\pi \times 2 \times \pi$

$$\frac{\partial u_i}{\partial x_i} = 0$$

Vuorinen et al., Com.in Comp.Physics (2016)

$\boldsymbol{g} = (g_x, 0, 0)$

$H=2$

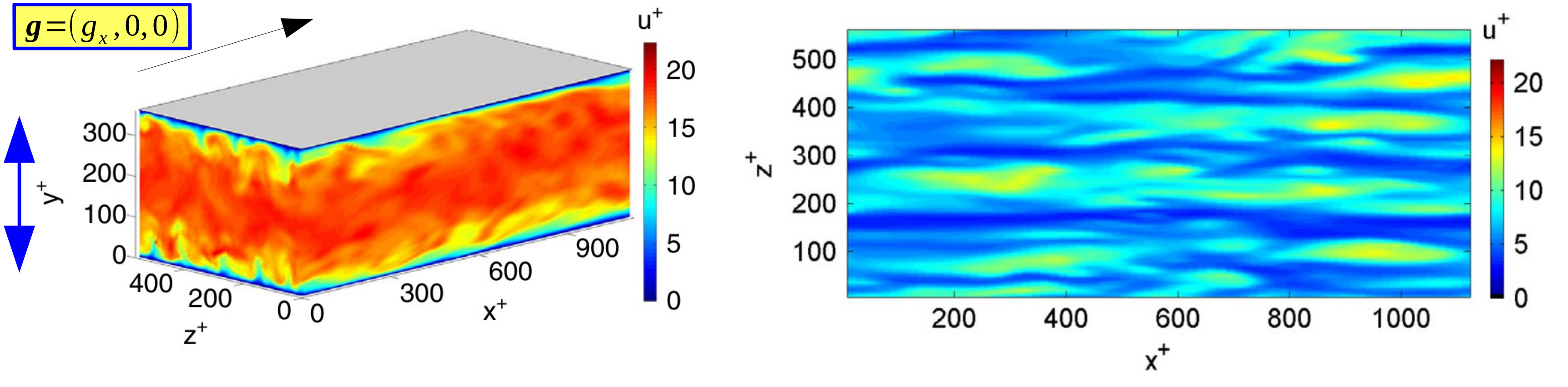


Fig. 5. Matlab simulations of instantaneous velocity in the channel (left), and visualization of the near-wall streaks at $y^+ \approx 10$ (right).

Note:
- Channel flows are very much studied configurations in understanding near-wall turbulence
- The “standard” for LES/DNS code benchmarking
- Steady flow maintained with the external force ***g***
- Periodic boundary conditions

Turbulent channel flows – Mean velocity profiles