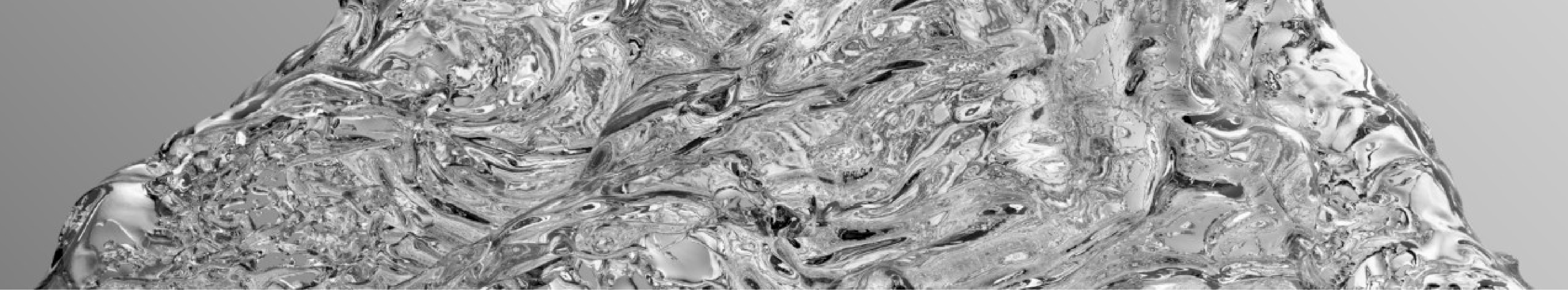


EEN-E2001 Computational Fluid Dynamics

Lecture 5: Fluid physical phenomena

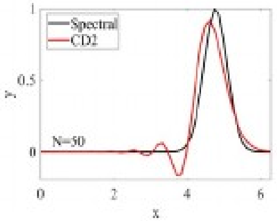
Prof. Ville Vuorinen
February 12th 2024
Aalto University, School of Engineering

Lecture 1: Linear PDEs and finite difference method

HW1

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

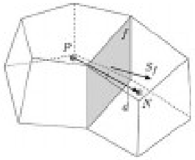
$$\boxed{\frac{\partial T}{\partial x}\approx\frac{T_{i+1}-T_{i-1}}{2\Delta x}}$$



Lecture 2: Gauss' theorem and finite volume method

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

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

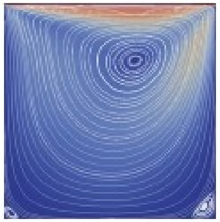


Lecture 3: Navier-Stokes equation and pressure

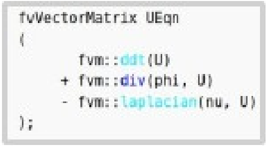
HW2

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

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

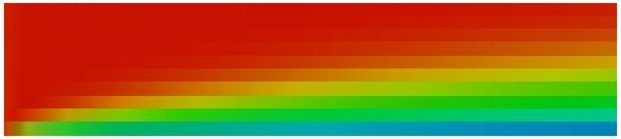


Lecture 4: OpenFOAM code and structure



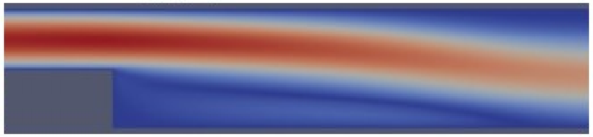
```
fvVectorMatrix UEqn
(
      fvm::ddt(U)
    + fvm::div(phi, U)
    - fvm::laplacian(nu, U)
);
```

HW3



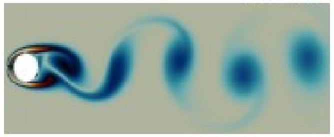
Lecture 5: Fluid physical phenomena: (laminar and turbulent flow)

HW4



Lecture 6: Matrix equations Ax=b and final assignment

HW5



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.

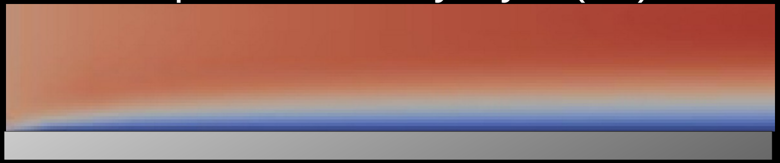
7) **Visualization and post-processing**.

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

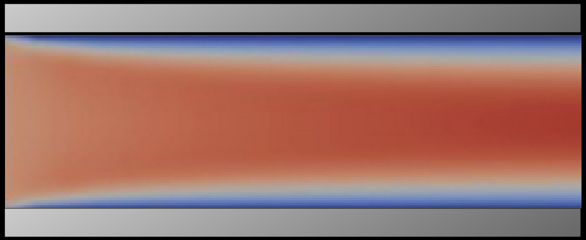
Examples on physics phenomena in fluid dynamics

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

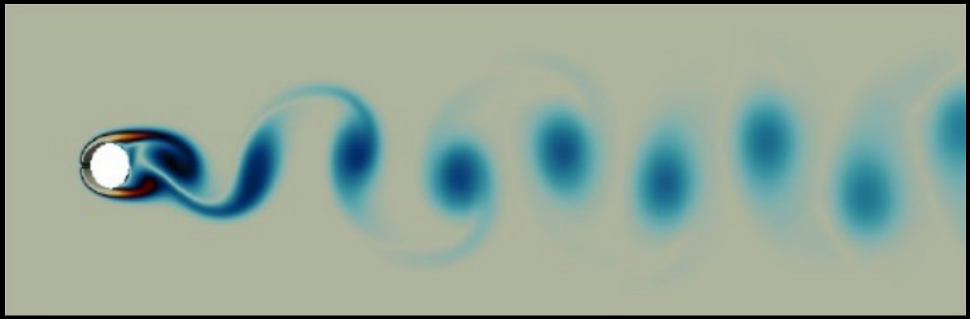
Free flat plate boundary layer (BL)



Channel flow

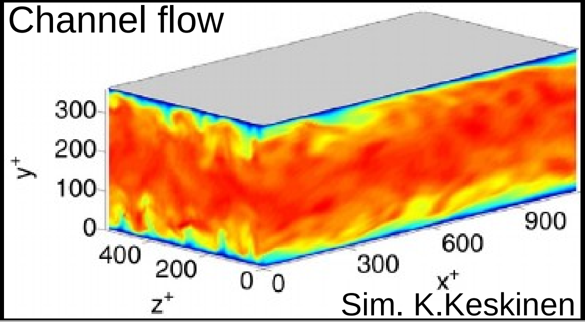


Unsteady, laminar flow. **Key physics**: von Karman instability and vortex shedding



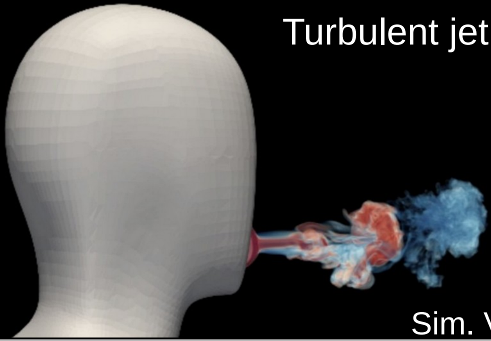
Unsteady, turbulent flow. **Key physics:** turbulence production at walls and shear layers

Channel flow



Sim. K.Keskinen

Turbulent jet

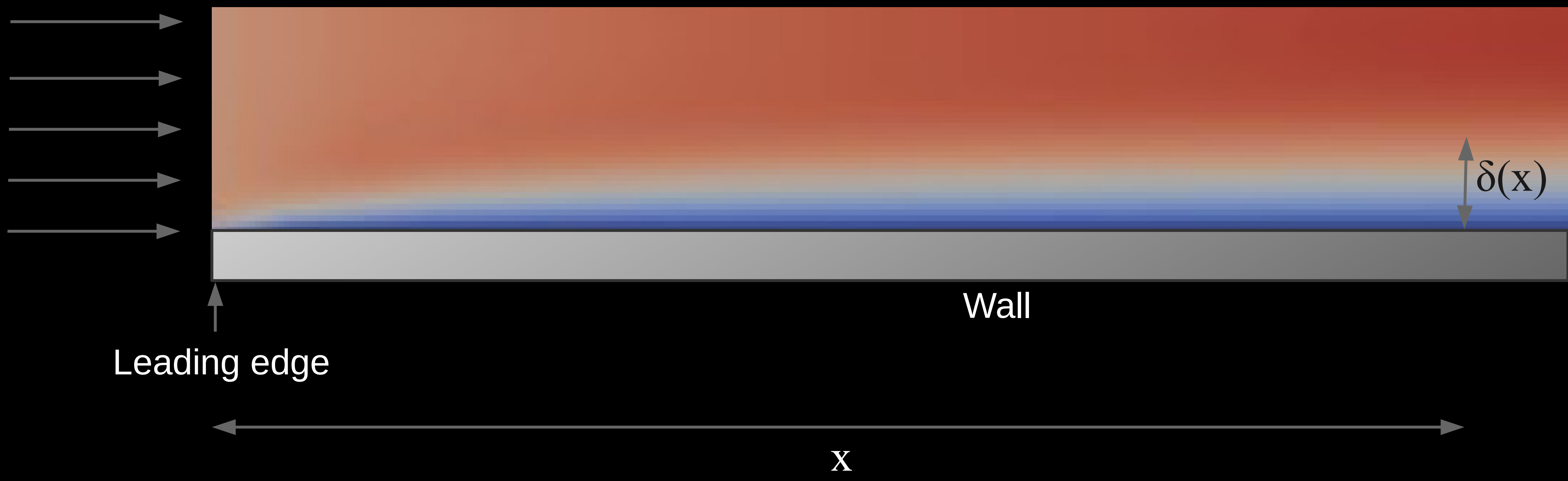


Sim. V.Vuorinen

Free boundary layer

Relevance: HW3

Free boundary layer (relevance: HW3)



Recall: free boundary layer thickness grows as

$$\boxed{\delta \approx 5.0\sqrt{\frac{\nu x}{U_\infty}} = 5.0\, x/\sqrt{\mathrm{Re}_x}}$$

$$\boxed{\mathrm{Re}_x = \frac{U_\infty x}{\nu}}$$

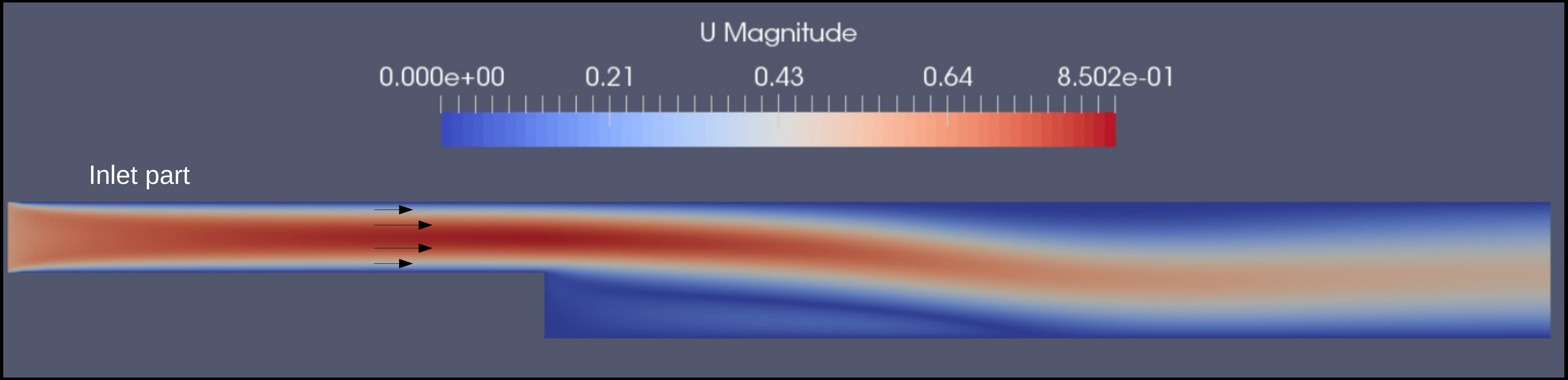
Backward facing step flow

Relevance: HW3-HW4

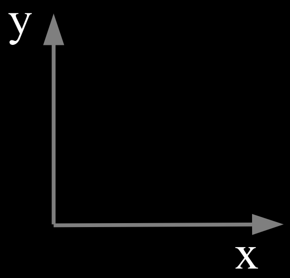
Real world “Backward facing step”

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

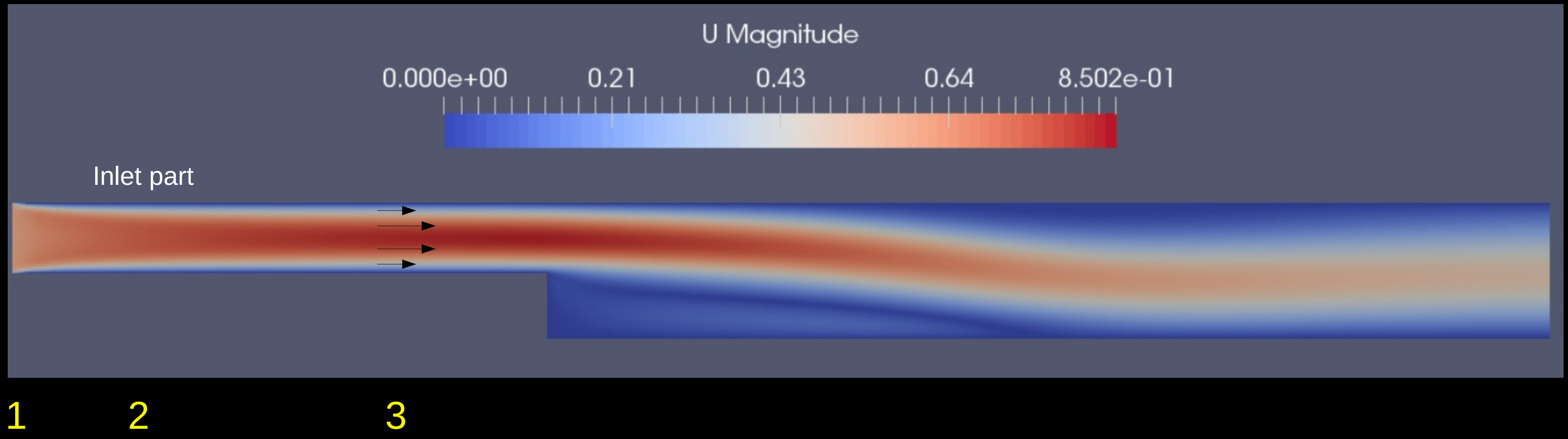


Reynolds number:

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

Viscous entry length after which laminar channel flow fully developed:

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

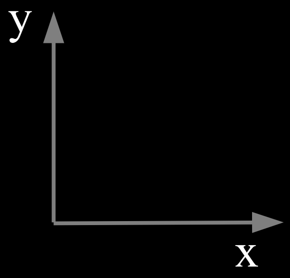


Comment 1: constant velocity at inlet.

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

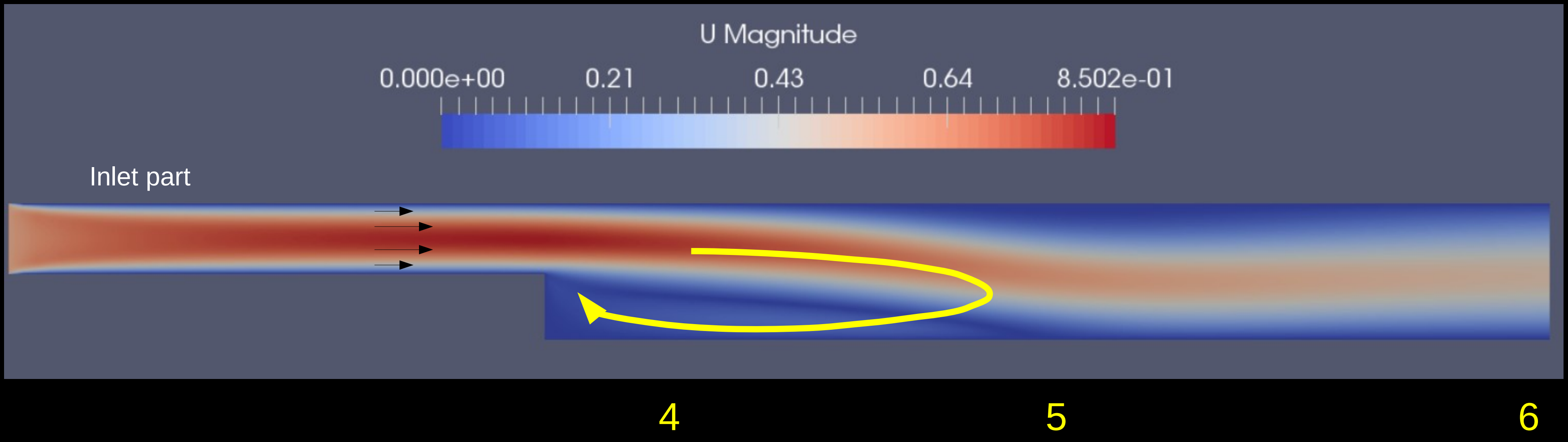


Reynolds number:

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

Viscous entry length after which laminar channel flow fully developed:

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



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

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

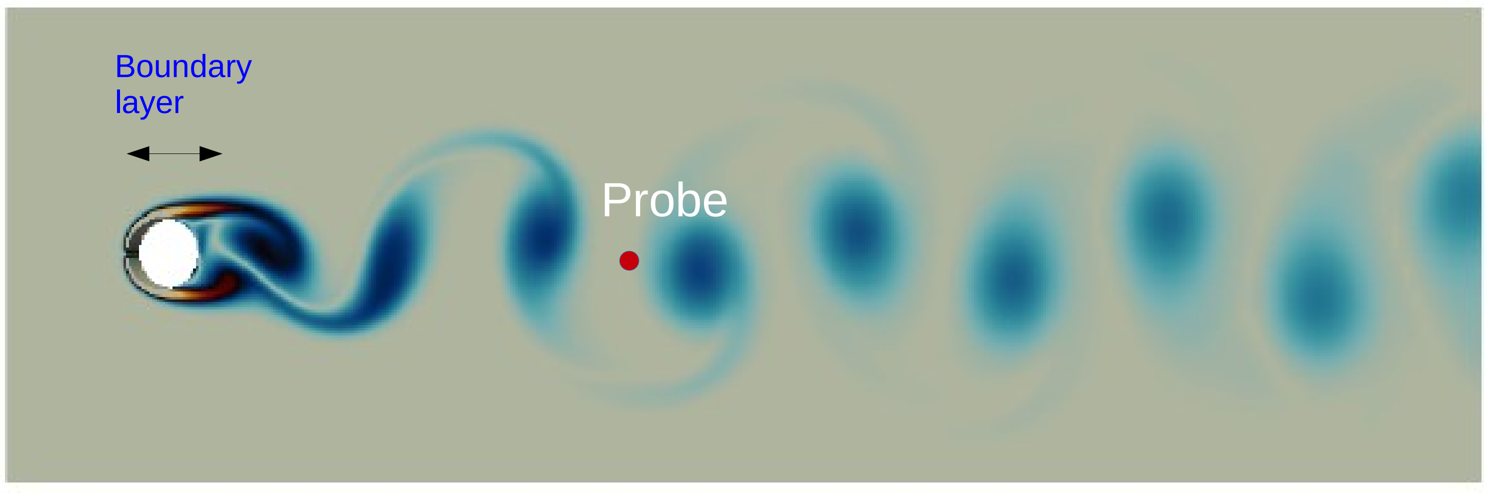
Flow through time scale

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

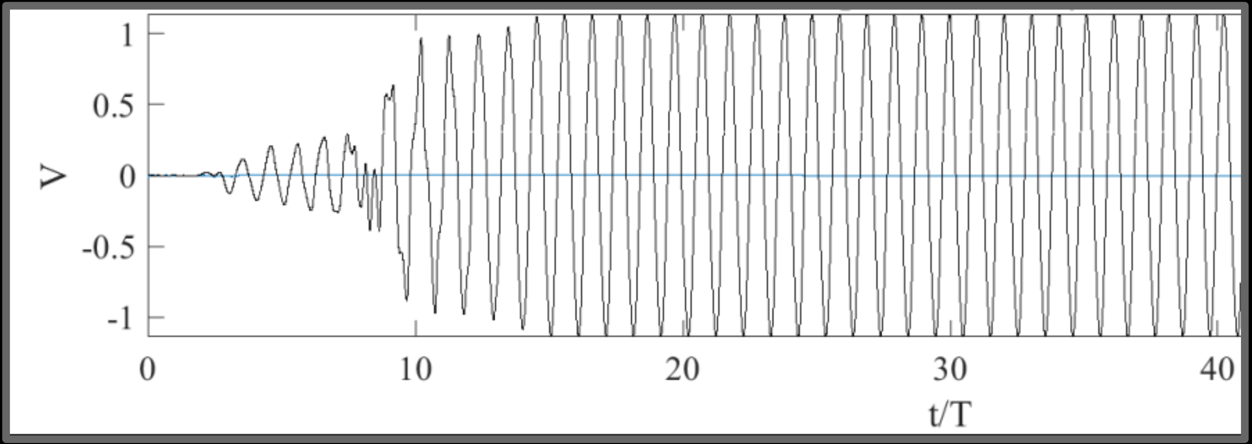
Flow over a cylinder

Relevance: HW5

Flow over a cylinder

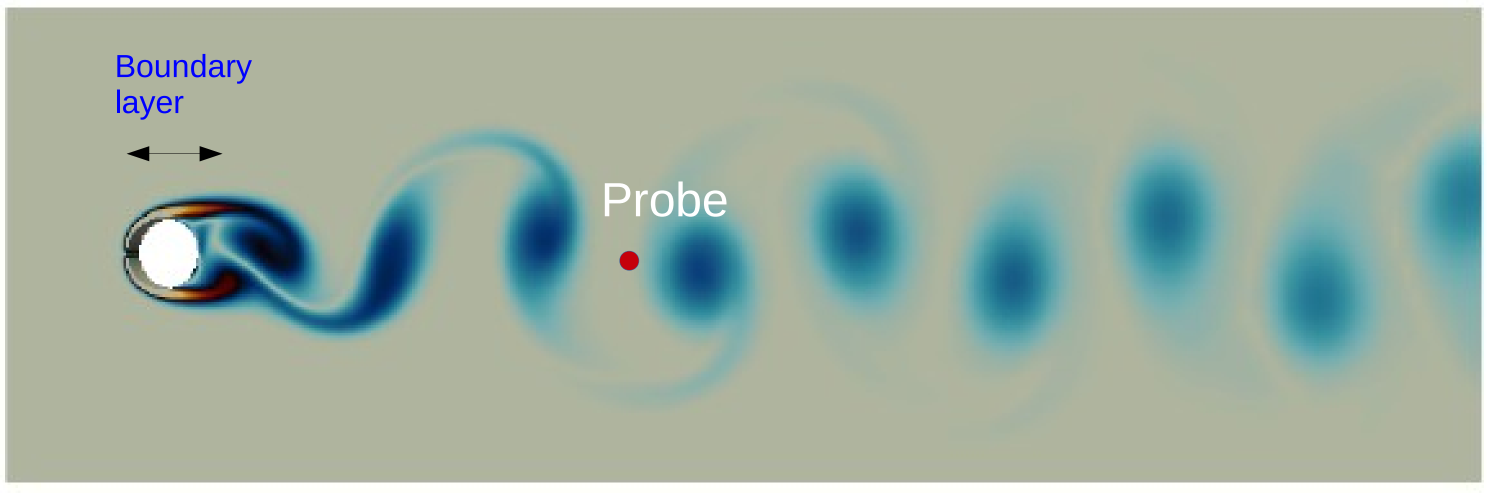


Probe

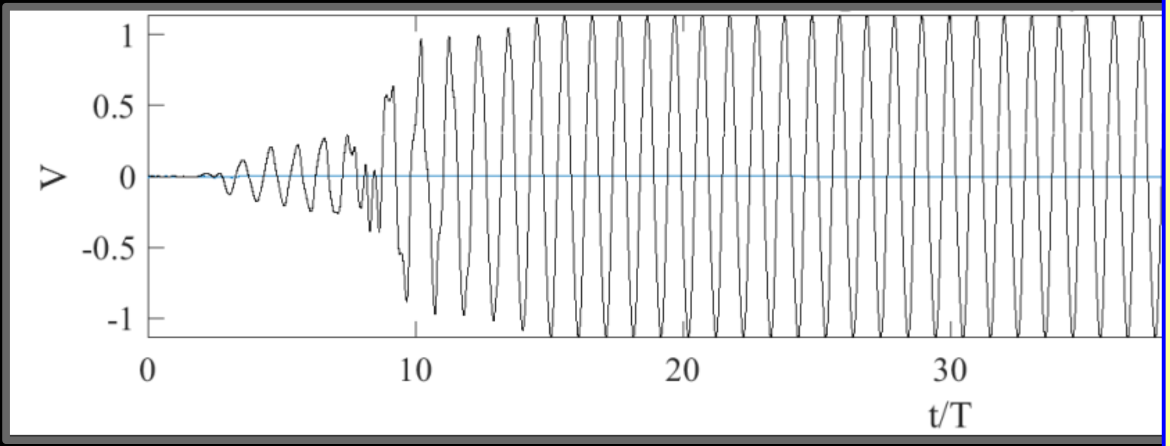


Typical scenario:
After initial transients flow enters a periodic shedding mode with a distinct frequency (Strouhal number) observed from the spanwise velocity component by downstream probing or lift-coefficient variation.

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`**” (2nd order)

Turbulent flow

Insight to turbulent flow

1) CFD course: we directly solve Navier-Stokes eqn under laminar conditions

2) CFD course OpenFOAM solvers: directly applicable for turbulent flows as well under moderate Reynolds # (e.g. Re<5000) without turbulence modeling on very fine grid resolution $\Delta x \approx \eta$. This is called: "**direct numerical simulation**" or "**DNS**"

3) In practice, for turbulent conditions, it is not feasible to refine the grid all the way to the smallest **Kolmogorov length scale** η ("eta") where **kinetic energy dissipates to heat**. DNS is computationally expensive although possible.

4) In turbulent flow, energy flows from large scales to η. **Think:** pour milk to coffee and what happens? What is "large scale" and what are the "medium scales"?

5) In practice, for turbulent flows, in CFD we can "afford" to resolve until $\Delta x > \eta$. Turbulence modeling needed to compensate the "missing" kinetic energy dissipation on scale Δx. Physics: there are "subgrid eddies" of size < Δx.

6) Large-eddy simulation (LES turbulence model): fine resolution CFD, only model the "subgrid" scales. Offers space-time dependent transient information.

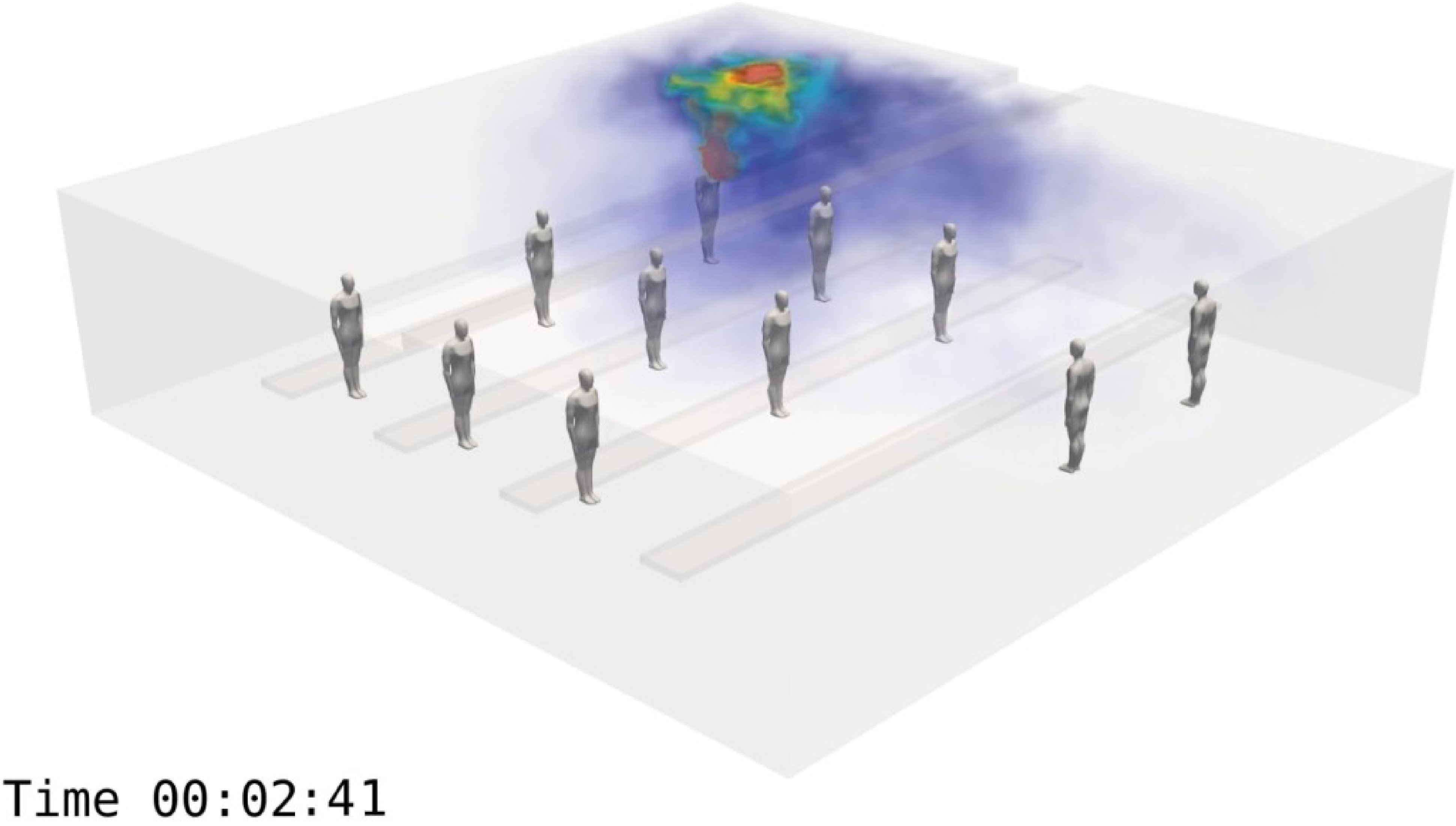
7) Reynolds averaged Navier-Stokes simulation (RANS turbulence model): Offers time-averaged solution.

Large-eddy simulation of buoyant airflow in an airborne pathogen transmission scenario publication

(Laitinen, Vuorinen et al. 2023)

https://www.sciencedirect.com/science/article/pii/S0360132323004894

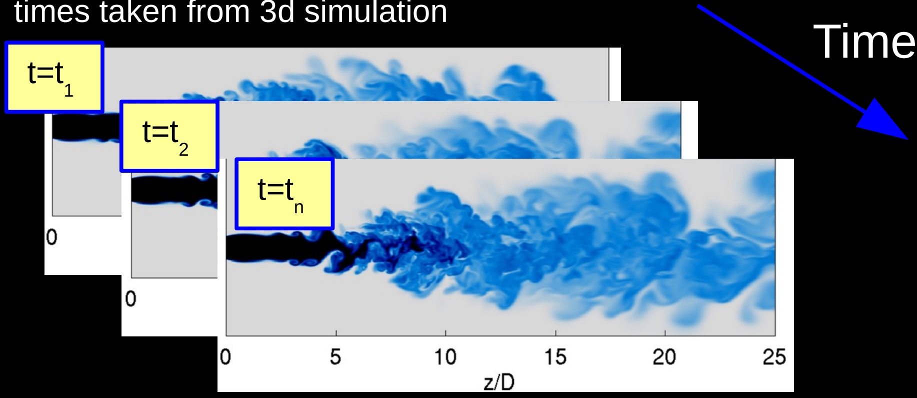
https://www.youtube.com/watch?v=f7MLFW_QJLo



Turbulent jets

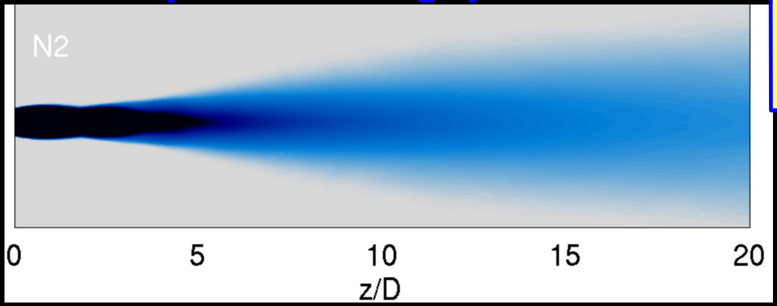
Below: we inject a concentration field in the jet flow to study mixing.

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



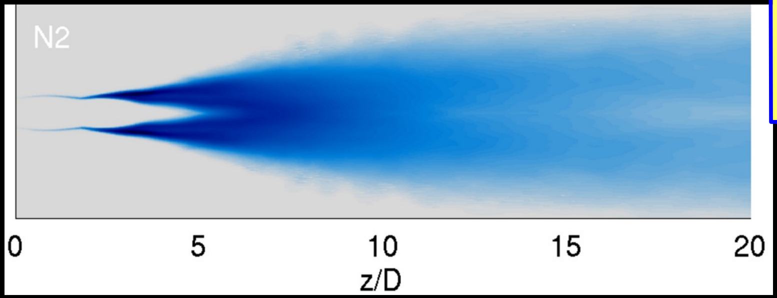
Here: passive scalar field averaged in time over several thousand cutplanes (time snapshots)

Mean (time average)



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

Standard Deviation



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

Turbulent channel flows

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

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

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

Domain size

$2\pi\times2\times\pi$

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

$H=2$

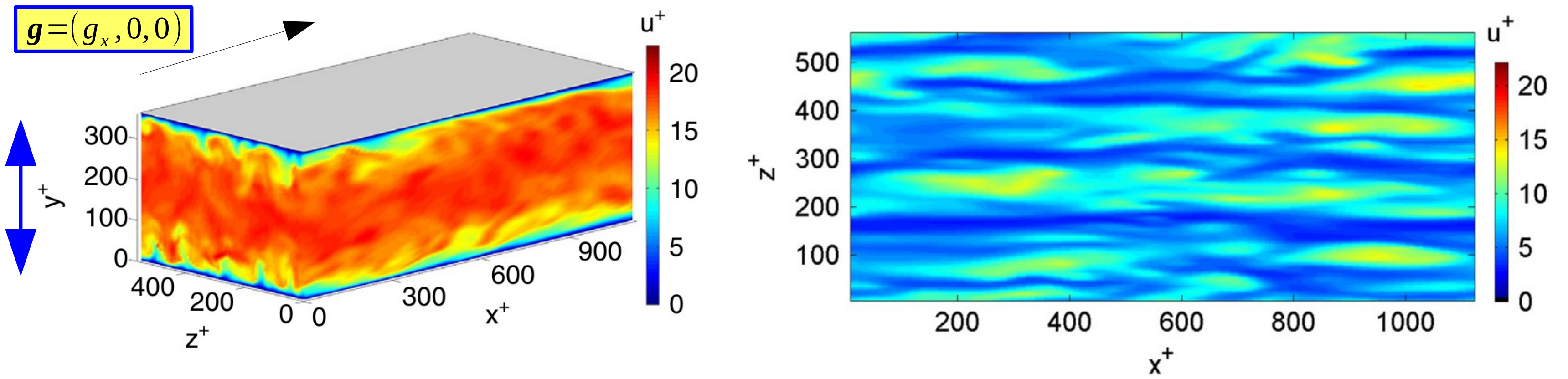


Fig. 5. Matlab simulations of instantaneous velocity in the channel (left), and visualization of the near-wall streaks at $y^+ \approx 10$ (right).

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