Dr. Richard Stevens (r.j.a.m.stevens@utwente.nl)
Yang Liu (y.liu-4@utwente.nl)
Davide Selvatici (d.selvatici@utwente.nl)
Physics of Fluids
Department of Applied Physics
University of Twente

Turbulence
Assignment 3

VERY IMPORTANT: Submit this assignment digitally on canvas before **January 19th, 2024, 23:59**. You are allowed to work in pairs: 2 people submit 1 solution. **Show all of your steps in the derivations, state the assumptions you make, and provide source code of numerically solved tasks!** Ensure that your solutions are clear and legible. Unfortunately, if the grader cannot understand your process, you will not receive any points. Reminder: Physical quantities must have units and figures must have appropriate labels and axes (i.e. logarithmic, if applicable). You may also find it useful to refer to the online cheatsheet.

1 Turbulent wall flow with wind turbines (2.5 points)

Here you will develop a simple model for a wind farm applying what we have learned about turbulent, wall-bounded flow. The turbines involved have a hub-height, $H_T = 100$m, a rotor diameter $D = 100$m, and thrust coefficient $C_T = 0.75$. The boundary layer has a height $\delta = 1000$m. The spacing between the turbines in the x and y direction respectively is S_x, S_y (Figure 1).

Figure 1: An image of wind farm indicating the turbine spacings S_x and S_y. Image source: `https://www.climate.gov/news-features/featured-images/wind-turbines-churn-air-over-north-sea`.

Consider three layers (Figure 2): below and above the turbine rotor blades (1,3) and in between (2). We parametrize the effect of the turbines by considering an effective roughness height z_0 and a friction velocity u^* in each region. In other words, the presence of the wind turbine results in a velocity profile similar to that of a flow with a friction velocity of u^* over a flat surface at a distance z_0 above the ground. For $z < H_T$ (region 1 and part of region 2 below the rotor hub) we assume $z_0 = z_{0,lo}$ and $u^* = u^*_{lo}$ and for $z > H_T$ (region 3 and part of region 2 above the rotor hub) we assume $z_0 = z_{0,hi}$ and $u^* = u^*_{hi}$. The velocity profile resulting from our assumptions is a piecewise continuous function of height.

Note: $z_{0,\{lo,hi\}} \neq H_T \pm D/2$ *and* $u^* \neq u(z_0)$*!*

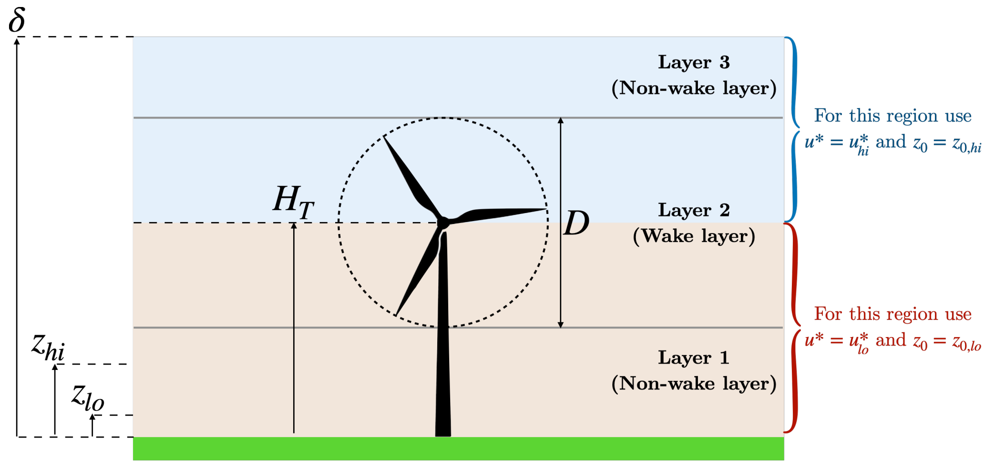

Figure 2: Schematic (not scaled) of the boundary layer: one wake layer (2) and two non-wake layers (1,3). When $z \leq H_T$, $z_0 = z_{0,\text{lo}}, u^* = u^*_{\text{lo}}$, and when $z > H_T$, $z_0 = z_{0,\text{hi}}, u^* = u^*_{\text{hi}}$. ($z_{0,\{lo,hi\}} \neq H_T \pm D/2$ and $u^* \neq u(z_0)$)

a) Begin by determining the velocity profile below and above the turbines (layers 1 and 3) using the stream-wise velocity gradient for the flow. Integrate vertically with respect to z considering that the below equation applies above the roughness height ($z > z_0$).

$$\frac{du(z)}{dz} = \frac{u^*}{\kappa z} \tag{1}$$

Why is it not realistic to characterize the entire region above the turbines ($H_T + D/2 < z < \delta$) using $z_{0,\text{hi}}$ and u^*_{hi}?

Hint 1: Assume $u(z_{0,lo}) = 0$ for layer 1 and $u(z_{0,hi}) = 0$ for layer 3
Hint 2: Use the appropriate friction velocity and roughness height parameters for each region.

b) Next, assume that in the wake layer (layer 2), where the wind turbines are present, an extra mixing term is introduced using the effective eddy viscosity ν^*, such that:

$$(1 + \nu^*)\frac{du(z)}{dz} = \frac{u^*}{\kappa z} \tag{2}$$

Determine the velocity profile in the upper and lower part of this layer by integrating vertically. State the velocity at hub-height in terms of both parametrizations.

Hint 1: Again, use the appropriate u^* and z_0 values in the regions in which they are applicable.
Hint 2: Assume that the velocity is continuous at $z = H_T \pm D/2$.

c) The following expression relates the two friction velocities ($u^*_{\text{lo}}, u^*_{\text{hi}}$) considering the drag introduced by the turbines.

$$u^2_{*,hi} = u^2_{*,lo} + \frac{1}{2}c^{ft}u(H_T)^2 \tag{3}$$

where the loading coefficient, $c^{ft} = \frac{C_T A}{S_x S_y}$, depends on the characteristics of the wind farm such as the rotor area, $A \equiv \pi D^2/4$, and the spacing between turbines. Use this expression and the velocity at hub-height to determine an expression for the roughness height above the turbines, $z_{0,\text{hi}}$.

d) Using the results thus far, plot and describe the velocity profile (in units of the unknown velocity u^*_{lo}) of a wind farm where $S_x = 785\text{m}, S_y =, 524\text{m}$, $z_{0,\text{lo}} = 0.1\text{m}$, $\nu^* = 28\sqrt{\frac{1}{2}c^{ft}}$, and $\kappa = 0.4$ (von Kármán constant).

2 Data from a DNS of channel flow (2.5 points)

In this exercise we will use a freely available database for channel flow to study some basic statistical properties of turbulent boundary layers ourselves.

The Johns Hopkins Turbulence database (`http://turbulence.pha.jhu.edu`) provides scientists access to the full space and time history of large scale simulations of forced isotropic turbulence, forced MHD turbulence, channel flow, and homogeneous buoyancy driven turbulence.

For this problem, we have already downloaded a part of the channel flow data from the Johns Hopkins Turbulence database and we provide this HDF5 file along with the assignment through Canvas (channel_data_JHU.h5). We provide the instantaneous snapshot of the streamwise (x-component) velocity field 'u' which is a dataset of size $512 \times 512 \times 384$ covering a grid in (x, y, z). We also provide the wall-normal coordinates (in the y direction) as a dataset 'y' of size 512.

Note: Most relevant formulae and explanation can be found in the relevant chapter of Pope's book. You may also refer to the online cheatsheet for a reminder of how to read these HDF5 files. Normalise the velocities and length with the appropriate quantities.

a) Make a visualization of the streamwise (x-component) velocity at the wall-parallel ($x - z$) plane at a distance $\Delta y^+ = 10$ from the bottom wall of the channel. Describe the flow structures that you see in the visualization.

Hint 1: Note that the y-coordinates provided in the HDF5 file range between $y = -1$ for the bottom wall and $y = 1$ for the top wall of the channel.
Hint 2: You may recall the scaling of distances in wall units is given by $\Delta y^+ = \Delta y u_\tau / \nu$, where Δy is the distance to the wall, u_τ is the friction velocity and ν is the viscosity.
Hint 3: For the values of Re_τ, u_τ and the grid spacing in the x and z directions, please refer to the channel flow database (`http://turbulence.pha.jhu.edu/Channel_Flow.aspx`).
Hint 4: For more information on the flow structures, you may refer to [6].

b) Calculate the mean velocity profile as function of the wall normal distance. Describe your working procedure. Make two plots of the results, using both linear and logarithmic scales for the x-axis, but keep the y-axis linear. Identify the viscous sub-layer, and the logarithmic sub-layer. Do they begin and end at the locations you expect?

c) The largest channel flow simulation to date has been performed by Lee and Moser [7]. Data from this simulation can be found on `https://turbulence.oden.utexas.edu/channel2015/data/LM_Channel_5200_mean_prof.dat`. Download the results and compare with the results you have obtained in the previous exercise.

d) By fitting the appropriate function at the appropriate range, give an estimate for κ, the von Karman constant from this DNS data. Does it meet your (read: our) expectation?

e) Now go back to the to your 3D dataset and calculate the variance ${u'}^2 = (u - \langle u \rangle)^2$ as function of wall distance and plot the results as function of the inner and outer length scale. Describe the procedure you used. Compare your results with the results for higher Reynolds number simulation by Lee and Moser (`https://turbulence.oden.utexas.edu/channel2015/data/LM_Channel_5200_vel_fluc_prof.dat`).

f) Use the 3D dataset to calculate the streamwise energy spectrum of the streamwise velocity $E_{uu}(k_x)$ for three different heights

- mid-plane i.e. $y^+ = 1000$
- in the log-law region at $y^+ = 100$
- in the viscous sublayer at $y^+ = 0.5$

Compare the spectra you computed in the log-law region and at the mid-plane with the spectra provided in the JHU dataset (`http://turbulence.pha.jhu.edu/docs/channel/spectra-kx-yplus-99.75.txt` and `http://turbulence.pha.jhu.edu/docs/channel/spectra-kx-yplus-999.7.txt`). Plot the Kolmogorov scaling $-5/3$ in each of the three graphs. Which differences do you see, and why?

As we will see more quantitatively in the next problem, DNS simulations require a lot of computational resources, and this translates into money. Aside from the Johns Hopkins Turbulence Database, other initiatives to share DNS data include `https://torroja.dmt.upm.es` and `http://newton.dma.uniroma1.it/database/`.

3 DNS and resolution requirements for turbulence (2.5 points)

For over a quarter of a century numerical simulations are contributing to the understanding of turbulence. A major advantage of Direct Numerical Simulations (DNS) compared to experiments is that the complete velocity is available for analysis. However, in order to obtain a correct solution one must resolve all scales of the turbulent field, leading to a problem with a large number of degrees of freedom.

a) Explain why smaller scales are two-way coupled to larger scales in three-dimensional (3D) turbulence, even though there is a direct energy cascade.

b) In a DNS, the grid spacing must not be much larger than the smallest physical scale in the flow. The computational domain (grid size) must of course also be large enough to fit the integral scale of the turbulence L. Find how the amount of required numerical grid points $\mathcal{D}$ scale with the integral Reynolds number Re for homogeneous 3D turbulence, i.e. find α in $\mathcal{D} \sim Re^{\alpha}$.

c) In this problem let's estimate the computational resources required for weather prediction. We need a three dimensional numerical simulation of Earth's atmosphere. Let's make the following assumptions for simplicity.

i) The curvature of earth's surface can be neglected and laid out on a flat surface.

ii) Take 100 slices in the wall-normal direction and assume uniform grid discretisation with a cell size of 1 kilometre in each direction (essentially its a box simulation with the bottom surface representing earth's surface).

iii) Simulating one time-step requires 30 seconds while each computational cell requires 10^3 floating-point operations per time-step.

Estimate the time it will take to make a 24 hour forecast on a 1 Tflop/s machine.

d) The amount of grid points is not the only contributing factor to the computational requirements of simulating turbulence. One must also consider the timestep. Here, the timestep is the advancement in physical time between two consecutive iterations over the complete grid. Why is this important for turbulence simulations?

e) Consider a DNS simulation of a Rayleigh-Bénard flow in a box where the top and bottom are two walls and the other directions are periodic boundaries. In an unconfined domain, large-scale flow structures frequently emerge. If the computational box size is made too small, these bigger turbulent structures might not be detectable. Explain how the energy spectrum of the flow field can be a tool to detect turbulent structures and indicate how the minimum size of a sufficient simulation covering all turbulent scales can be determined.

f) Explain whether the computational work would increase, decrease or remain identical for non-homogeneous (e.g. with no-slip walls) turbulence compared to homogeneous turbulence at the same Reynolds number.

g) By looking at the answer to part (b), you can see why DNS is impossible in practice for very large Reynolds numbers. Briefly (2-3 short paragraphs) explain why RANS and LES require less computational resources than DNS, focusing on what parts of turbulence they model and how this reduces the resolution required. Remember that the approaches LES and RANS take to model turbulence are different, so explain each separately.

4 Structure functions from numerical data (2.5 points)

We will now calculate structure functions from a numerically obtained velocity field. The pth-order longitudinal structure function D_p for the radius $\boldsymbol{r}$ at position $\boldsymbol{x}$ is given by:

$$D_p(\boldsymbol{x}, \boldsymbol{r}) = \left\langle (u(\boldsymbol{x} + \boldsymbol{r}) - u(\boldsymbol{x}))^{2p} \right\rangle \tag{4}$$

On Canvas you will find a HDF5 file (`RB_vz_planes.h5`) containing slices of the z-component of the velocity field, u_z, taken at a specific time instant of the Rayleigh-Bénard set-up. This setup consists of a fluid layer which is heated from below and cooled from above, between horizontal plates at constant temperature T_{hot} and T_{cold} respectively. Gravity is pointing in the negative x-direction (wall normal). The velocities in y-direction and z-direction are periodic. Note that the domain is square with respect to y and z having widths of 32 times the wall-normal height.

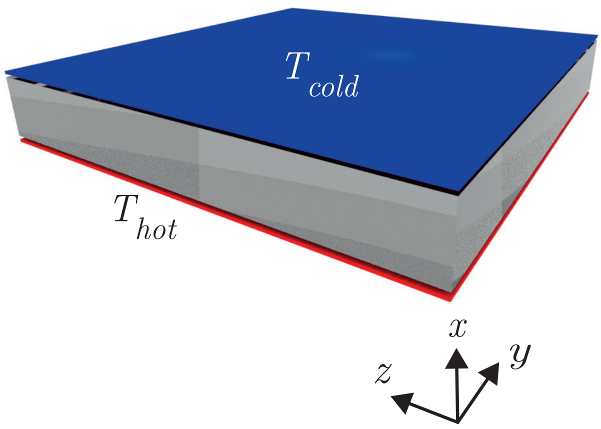

Figure 3: Sketch of the Rayleigh-Bénard problem. A fluid layer is heated from below and cooled from above.

The corresponding grid can be found in `coords.h5`, where three arrays are stored, namely `x[i]`, `y[j]` and `z[k]` with `i,j,k` integers. You will not need `x` since we are only providing you with three horizontal slices, at $x = 0.005$, $x = 0.25$, and $x = 0.5$. Refer to the online cheatsheet for a reminder of how to read these HDF5 files.

a) Calculate the RMS of the velocity perturbation for each of the plane slices. This provides a measure of turbulence intensity at each height. Do the results make sense? Explain.

b) In a single figure, plot the PDFs of u_z for each plane. Compare these with a Gaussian distribution and explain your results.

c) Calculate D_p , as given in (4), for each of the three heights and for $p = 1, 2, 3, 4$. For each p plot D_p as a function of $r = \|\boldsymbol{r}\|$, use log scales for both axes. Due to the large domains, you should only need to average D_p over a small number of sample values for $\boldsymbol{x} = (x, y, z)$, but it is important to consider a wide range of $\boldsymbol{r} = (y', z')$. We have provided you with a code snippet to help you get started in `structure_funs.py`.

d) What happens for large r? What does this mean physically? Relate this to your previous questions.

e) Calculate the exponents of the power-laws of D_p in the inertial range. Are they equal (or not) to the K41 predictions? If not, suggest a reason for the discrepancy.

References

[1] S. Hoyas and J. Jiménez Phys. Fluids, 18, 011702 (2006).

[2] E. B. Gledzer, Sov. Phys. Dokl. 18, 216 (1976).

[3] M. Yamada and K. Ohkitani, J. Phys. Soc. Jpn. 56, 4210, (1987).

[4] R. Benzi *et al.* Phys. Rev. E 48, R29 (1993)

[5] S. Grossmann, D. Lohse, A. Reeh, Phys. Rev. E **56** 5473 (1997)

[6] S. Kline, W. Reynolds, F. Schraub, P. Runstadler, J. Fluid Mech. 30(4), 741-773 (1967)

[7] M. Lee, R. D. Moser, J. Fluid Mech. 774, 395-415 (2015)