Krister Wiklund
Dept. of Physics
Umeå University

TURBULENT CHANNEL FLOW

Version 1.1

1. INTRODUCTION

This lab concerns the analysis of a turbulent 2D-channel flow. We use the CFD-software Comsol and start from RANS-equations and use the Boussinesq hypothesis approach where we apply a *k*-*ε* model for the turbulent kinetic energy and dissipation. We compare our simulation results with high accuracy Direct Numerical Simulation (DNS) data to learn how to perform the process of benchmarking a simulation model.

2. THEORY

The *k*-*ε* model describes the turbulent kinetic energy *k* and the dissipation *ε* and is used in addition to the incompressible RANS-momentum equations. The model equations we use are

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\frac{\mu+\mu_t}{\rho}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right], \quad (1)$$

$$\frac{\partial U_i}{\partial x_i} = 0, \quad (2)$$

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = 2\frac{\mu_t}{\rho} S_{ij} S_{ij} - \varepsilon + \frac{\partial}{\partial x_j}\left[\frac{1}{\rho}\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right], \quad (3)$$

$$\frac{\partial \varepsilon}{\partial t} + U_j \frac{\partial \varepsilon}{\partial x_j} = 2\frac{\mu_t}{\rho} S_{ij} S_{ij}\left(C_{\varepsilon 1}\frac{\varepsilon}{k}\right) - \varepsilon\left(C_{\varepsilon 1}\frac{\varepsilon}{k}\right) + \frac{\partial}{\partial x_j}\left[\frac{1}{\rho}\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right], \quad (4)$$

where ρ is the fluid density, μ is the molecular viscosity, μ_t is the turbulent viscosity, U is the mean flow velocity and S_{ij} is the strain rate. For the model coefficients Comsol use the values $\sigma_k = 1.4$, $\sigma_\varepsilon = 1.5$, $C_{\varepsilon 1} = 1.5$ and $C_{\varepsilon 2} = 1.9$, and the turbulent viscosity is defined by

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon}, \quad (5)$$

where $C_\mu = 0.09$.

In a channel flow the turbulence is strongly modified near the channel walls, and one way to handle this is to adjust model coefficients $C_{\varepsilon 1}, C_{\varepsilon 2}, C_\mu$ with correction functions (often called a damping functions) that are wall-distance dependent and are designed to have value one far from walls. There are many such functions available in the literature, but in this lab we use the AKN-model since this is the damping function implemented in Comsol CFD-module.

3. SIMULATION AND EVALUATION

The general aims of this lab are:

- to learn how to analyze turbulent flow data and draw conclusions from it
- to understand the setting structure in the Comsol setup
- to understand how to import data and create an interpolation functions from it
- to investigate how simulation results may depend on flow domain and mesh settings

3.1 STEP1

Start Comsol and load the mph-file distributed through the course Canvas webpage. Go through the structure under Parameters and Variables, make sure that you understand what the parameters and variables means. You are also expected to explore the different geometry and flow settings in the file. However, do not change anything yet, make sure that you run the original file when starting with the questions below.

In the mph-file, under Definitions, you find an *Interpolation function* where DNS-data has been imported and fitted into a function which is used in the plots under Results. In this way we can compare our simulation results with a benchmark data. The DNS-data represents the velocity profile in a channel flow at $\mathrm{Re}_\tau = 395$, a Reynolds number based on the friction velocity which is often used to describes the flow situation near walls.

Under Results you find a plot called “Wall resolution”. This built-in plot that help us to determine if the mesh is fine enough along the walls. With the setting we use in the mph-file (see main node “Turbulent Flow, Low Re k-ε” and Wall treatment = Low Re), the Wall resolution should be around one. If the Wall resolution is much higher than one, the mesh is not good enough for our model.

3.2 STEP 2

Run the simulation and work through and answer the questions below. The simulation time should be around 3 minutes for a decent computer. It is advisable run each question task in separate mph-files, even if it should be possible to run multiple Studies in one mph-file.

Q1: The *k-ε* simulation data for the velocity, U+, fits very good with the DNS-data. What can we say about the fit for P_k, ε and k data? To answer this you should import the DNS-data files for P_k, ε and k (can be found at the course Canvas) and plot them in the figure. What is your conclusion after you made the plot?

> Note: Pk means production of turbulent kinetic energy and corresponds to the first term on the RHS of the kinetic energy equation (3).

Q2: Decrease the length of the channel from 4m to 2m, and run the simulation. What can you conclude by comparing the 2m and 4m channel results?

Q3: The P_k and ε results data from the simulation seems to be almost identical (at least on average), what does this mean?

Q4: The P_k-curve is very staggered. Why is it so, and how do we smooth the curve?

Three approaches to test:

a) Run a simulation with a finer mesh by under Mesh applying Element size =Coarser. What do you conclude from the result?

b) Run a second simulation. Change settings under Mesh/Sequence type to "User-controlled mesh". Use Size=Extra coarse, Size1=Coarse, but under the Boundary Layer setting use Number of BL=20 and Stretching factor =1.1. What do you conclude from the result?

c) Run a third simulation. For the mesh use "Physics-controlled mesh" with "Extremely coarse" element size. Under the main node "Turbulent Flow, Low Re k-ε" change Discretization from *P1+P1* to *P2+P1*. What do you conclude from the result?

Note: It is expected that the simulation time becomes 3-4 times longer for P2+P1 compared to P1+P1.

4. REPORT INSTRUCTION

In this lab we focus on the results and the analysis of the results. The report that you individually hand in, according to the instructions below, is therefore a very brief report.

4.1 DEFINITION OF BRIEF REPORT IN THIS COURSE

Do not use bullet lists, discuss instead using a running text. Use max 4 figures and about a half page discussion for each figure. In the report the following parts should be included:

Alternative 1

- Front page (with the same data as in a full report, no Abstract)
- Result section (Clear figures and tables in which interesting parts are pointed at and very shortly commented)
- Discussion section (Each figure and table in the Result section are referred to and analyzed)

Alternative 2

- Front page (with the same data as in a full report, no Abstract)
- Result and Discussion are combined into one section (Clear figures and tables in which interesting parts are pointed at and discussed)
- Conclusion

4.2 Instructions for report submission

To make the organization of report submission easy for the examiner and to decrease the grading time the following instructions should be obeyed:

1. The report must be in a PDF-format
2. The name of the PDF must have the following form:

   TurbFlow_FIRSTNAME_SECONDNAME_DATE.pdf

   FIRSTNAME= Your first name
   SECONDNAME= Your second name
   DATE = The date you hand in the report and it should be on the format 211125

3. The report is submitted through Canvas / Assignments
4. **Deadline for submission is 25 November kl. 23:56**

APPENDIX

To compare flow between different experimental setups one usually construct plots with dimensionless quantities. In Figure A1 such plot is shown for a turbulent channel flow, where the velocity and distance to the wall is mad dimensionless by friction velocity and viscosity.

The friction velocity is defined by

$$u_\tau = \sqrt{\frac{\tau_w}{\rho}} ,$$

where τ_w is the wall shear stress and ρ is the density.

By using friction velocity, density and viscosity relevant quantities can be made dimensionless in the following way:

$$U^+ = \frac{u}{u_\tau}, \; k^+ = \frac{k}{u_\tau^2}, \; P_k^+ = \frac{P_k \mu}{u_\tau^2 \rho}$$

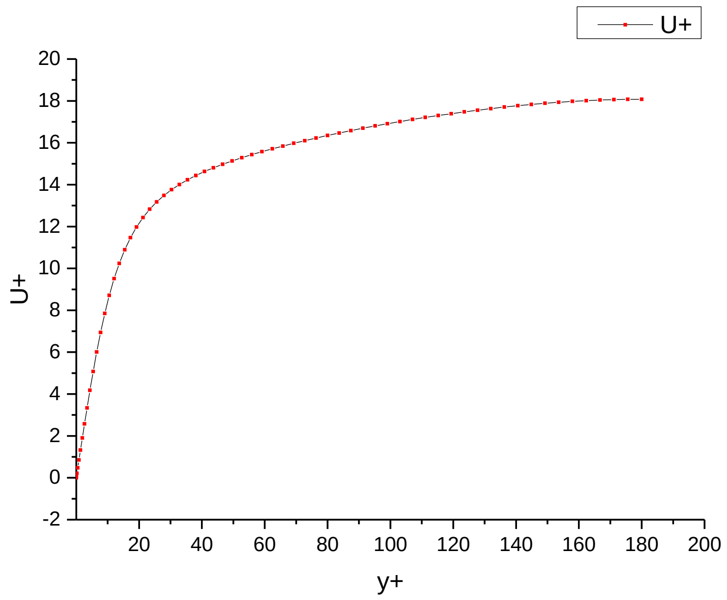

Figure A1. The velocity profile for a turbulent channel flow vs the perpendicular distance from a wall. The velocity is made dimensionless by friction velocity and the distance is made dimensionless by friction velocity and viscosity.