




Channel Flow Analysis: A RANS Perspective

Author: Rosario Donnarumma  **Student ID:** M53/1488

Supervisor: Prof. Gianluca Iaccarino

Department of Mechanical Engineering, Stanford University, Stanford, CA, USA

 Correspondence: rosar.donnarumma@studenti.unina.it

January 8, 2023

Abstract In this report, the implementation of a Reynolds-Averaged Navier-Stokes (RANS) model for simulating turbulent flow is presented. The main objective of the work is to calculate the mean velocity profile using a mixing length model. The specific case of a channel flow problem is discussed, in which the flow is confined between two parallel plates placed a distance h apart, driven by a uniform pressure gradient, and assumed to be fully developed in the streamwise (x) and spanwise (z) directions (i.e. L_x and L_z are much greater than $L_y = 2\delta$). The boundary conditions used are periodic, and a no-slip condition is applied at the walls.

Keywords: RANS, channel flow, turbulent flow, simulation.

1 Problem overview

The Reynolds-Averaged Navier-Stokes equations (RANS) are widely used to study wall-bounded flows, such as channel flows, by providing a means to obtain the averaged velocity profile. However, one of the main challenges in solving RANS equations is the closure problem, which arises from the presence of *Reynolds stresses*. To overcome this challenge, models are often introduced to account for these stresses. In the case of a fully-developed channel flow as the one shown in Fig. 1, the flow is assumed to be statistically stationary, homogeneous in the x and z directions, and with dimensions in these directions much larger than in the normal direction to the walls. By applying the gradient diffusion hypothesis and simplifying the equations given the nature of the problem, we can derive the equation of conservation of momentum along the x direction:

$$\frac{\partial}{\partial y} \left[(\nu + \nu_t) \frac{\partial \langle u \rangle}{\partial y} \right] = \frac{1}{\rho} \frac{d \langle p \rangle}{dx} = -\frac{1}{\rho} \frac{\tau_{\text{wall}}}{\delta} \quad (1)$$

where τ_{wall} is the normal stress at the wall, u is the x -components of the velocity, ρ the fluid density, p the pressure and lastly ν and ν_t are, respectively, the cinematic and turbulent viscosity. In the equation Eq. 1, the Reynolds stress $\langle u'v' \rangle$ has been represented using the *Van Driest model* that states that:

$$-\langle u'v' \rangle = \nu_t \frac{\partial \langle u \rangle}{\partial y} \quad (2)$$

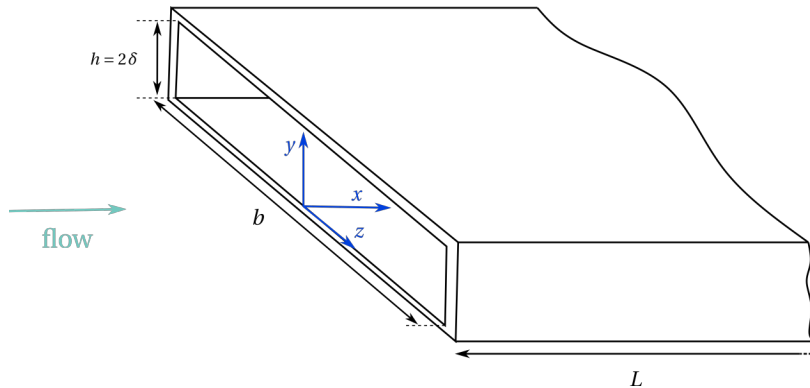


Figure 1: Representation of the channel under study.

In this context the following quantities (Eq. 3a, 3b, 3c) and dimensionless variables (Eq. 3d, 3e) can be defined:

$$u_\tau = \sqrt{\frac{\tau_{\text{wall}}}{\rho}} \quad \leftarrow \text{Friction velocity} \quad (3a)$$

$$\delta_\nu = \frac{\nu}{u_\tau} \quad \leftarrow \text{Viscous length scale} \quad (3b)$$

$$Re_\tau = \frac{u_\tau \delta}{\nu} \quad \leftarrow \text{Friction Reynolds number} \quad (3c)$$

$$y^+ = \frac{y}{\delta_\nu} \quad (3d)$$

$$\langle u \rangle^+ = \frac{\langle u \rangle}{u_\tau} \quad (3e)$$

and the equation of conservation of momentum in dimensionless form is written as follows:

$$\frac{\partial}{\partial y^+} \left(\left(1 + \frac{\nu_t}{\nu} \right) \frac{\partial \langle u \rangle^+}{\partial y^+} \right) = -\frac{1}{Re_\tau} \quad (4)$$

Considering an algebraic model with "eddy viscosity" as the turbulence model:

$$\nu_t = l_m^2 \frac{\partial \langle u \rangle}{\partial y} \quad l_m = \kappa y \left[1 - \exp\left(-\frac{y}{A}\right) \right] \quad (5)$$

In this formulation, l_m represents the *mixing length*, calculated as follows:

$$l_m = \kappa y [1 - \exp(-y/A)] \quad (6)$$

where κ and A^+ are the *Von Karman* and *Van Driest constants* and y is the distance from the wall. The exponential law assigned to l_m is such as to satisfy the condition that, near the wall, there is a viscous sublayer. A viscous length scale is considered: it allows to introduce wall units and further simplifies the equations of interest which take the following form.

$$\frac{d}{dy^+} \left[(\nu^+ + \nu^{t+}) \frac{d\langle u \rangle^+}{dy^+} \right] = -1 \quad (7)$$

It is possible to note that the Eq. 7 is a non-linear second-order differential equation.

2 Numerical simulation

In order to obtain a numerical simulation with higher resolution at the wall, a variable mesh was chosen: in fact, as can be seen from the illustration Fig. 2, the mesh was constructed in such a way as to be denser at the wall and less dense toward the center of the channel.

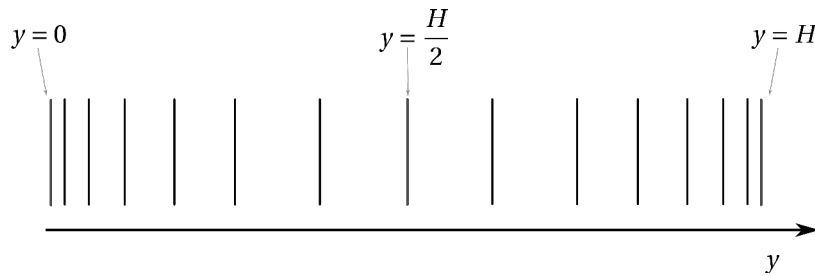


Figure 2: Schematic illustration of a nonuniform mesh for a channel flow problem.

Another challenge is solving the equation in which the eddy viscosity is still a function of the velocity gradient. The acquired answer is, in reality, the product of an iterative procedure based on the following criteria:

1. Initialize the solution by assigning the average velocity u_0 and the starting turbulent quantities;
2. Calculate the eddy viscosity using the Van Driest model;
3. Determine the velocity gradient by solving the velocity field.

This procedure yields a residual of the same order as the specified tolerance value.

3 Results and conclusion

In this section, the accuracy of Van Driest's model will be evaluated by comparing it with theoretical results. A Python code will be used for this purpose, which has been written in a way that takes into account the input parameters (Tab. 1). Two main cases will be considered:

- The stretching factor is set while the Reynolds number varies;
- The Reynolds number is set while the stretching factor varies.

Input Parameters	
Mesh points n	100
Channel height H	2
Tolerance tol	10^{-6}
Von Karman constant κ	0.4
Van Driest constant A	26

Table 1: Values used within the Python code.

The first case being dealt with is one in which the Reynolds number $Re_\tau = 2000$ and stretching factor $fact = 7$ are imposed, and solutions of different models are compared: the mixing length model, the linear model within the viscous sublayer, and the log-law in the log-layer. As can be observed in Fig. 3, the selected model is quite accurate in both the viscous sublayer, where the velocity profile is linear, and the log layer. Looking at the graph for eddy viscosity, we can see how, towards the wall, the distribution produced is similar to that for $(y^+)^4$. So, we can conclude that the eddy viscosity becomes zero towards the wall, when turbulence is no longer effective and only viscous dissipation occurs.

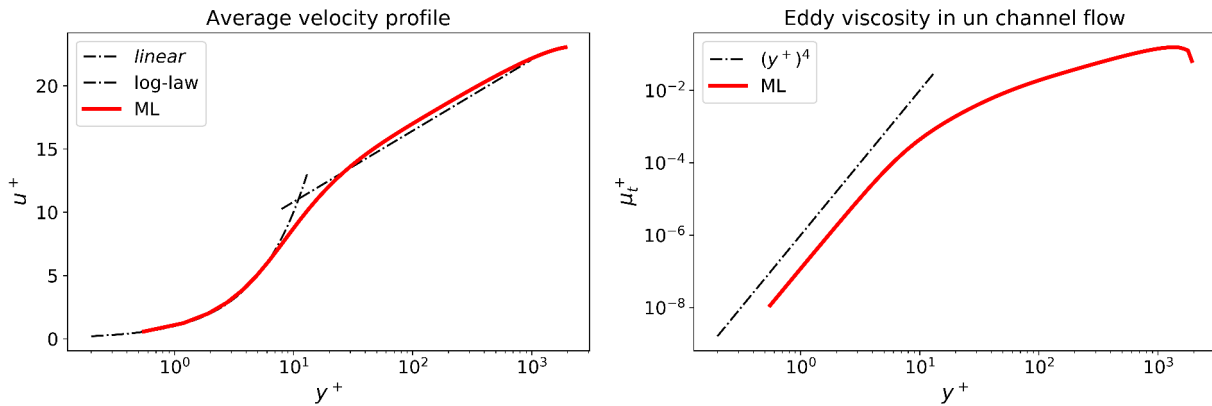


Figure 3: Average velocity profile (on the left) and eddy viscosity (on the right) in a fully developed channel flow. $Re_\tau = 2000$ and $fact = 7$.

At this point, it will be evaluated how the solutions change as the Reynolds number changes (set $fact = 7$), with specific focus on the cases $Re_\tau = 700$ (Fig. 4) and $Re_\tau = 3500$ (Fig. 5). As the Reynolds number changes near the wall, the curves converge. This can be explained by the fact that variations in turbulent Reynolds number have no effect on the average velocity profile in this area. The dominating viscous effect near the wall is responsible for this behaviour. The linear and logarithmic rules found in the viscous sublayer and in the log-layer exhibit universal behaviour, but in the "wake region", the findings are very sensitive to the Reynolds number. This is because, as we go away from the wall and closer to the channel centerline, viscous stresses become less important and Reynolds stresses take over, the behaviour of which is affected by both Reynolds number and channel height. Therefore, the out-layer region cannot be assumed as characterised by a universal behaviour.

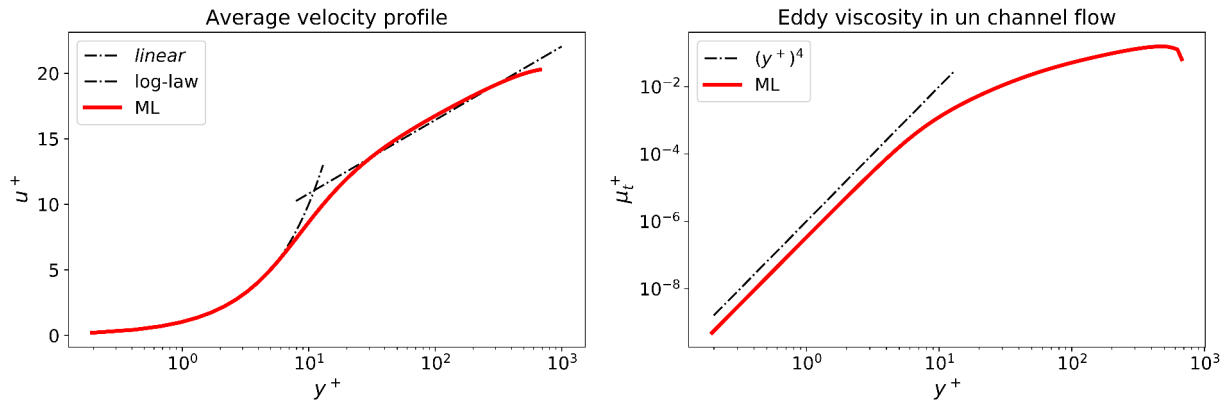


Figure 4: Average velocity profile (on the left) and eddy viscosity (on the right) in a fully developed channel flow. $Re_\tau = 700$ and $fact = 7$.

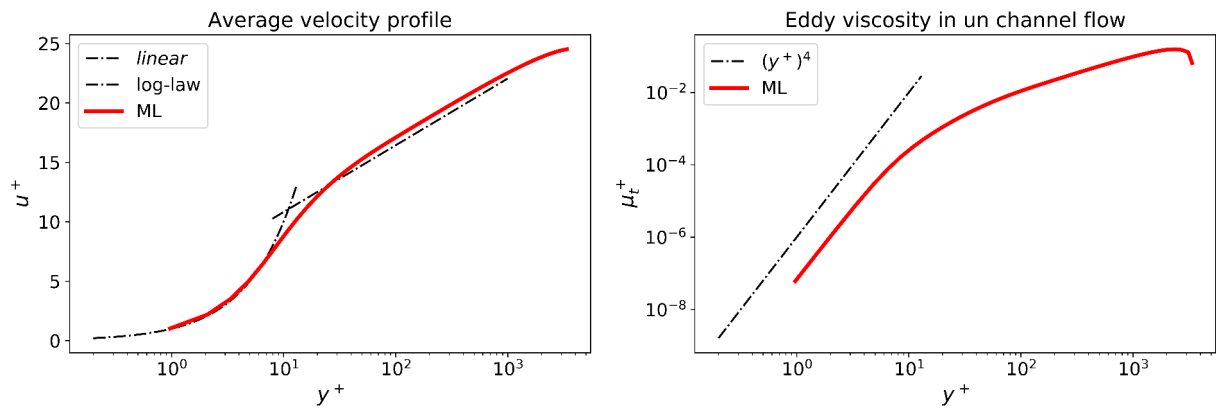


Figure 5: Average velocity profile (on the left) and eddy viscosity (on the right) in a fully developed channel flow. $Re_\tau = 3500$ and $fact = 7$.

As stated in the previous section, the mesh is a key element that must be properly constructed in order to achieve the greatest outcomes. It is discovered that raising the stretching factor (from 4 to 9) results in much greater near-wall resolution. This is due to a smaller (thus denser) computing grid along the walls, which allows one to record a larger quantity of motion information in such locations.

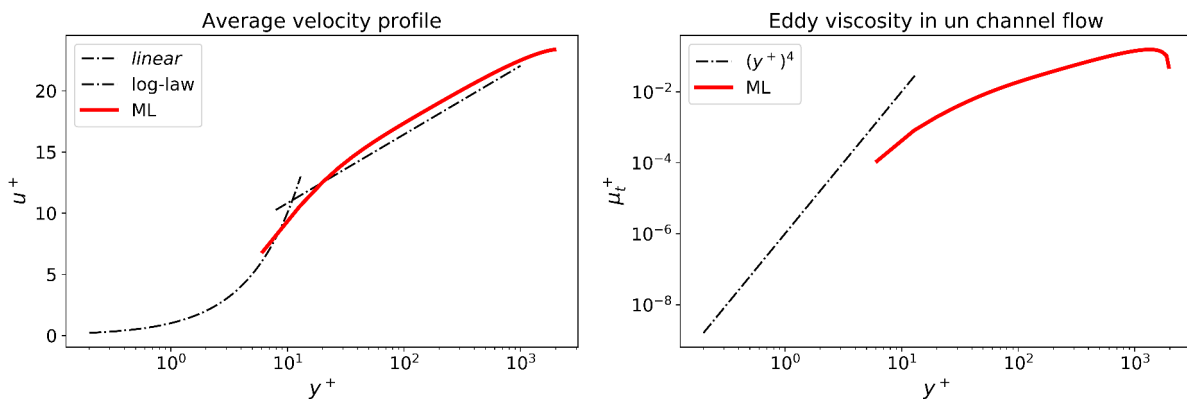


Figure 6: Average velocity profile (on the left) and eddy viscosity (on the right) in a fully developed channel flow. $Re_\tau = 2000$ and $fact = 4$.

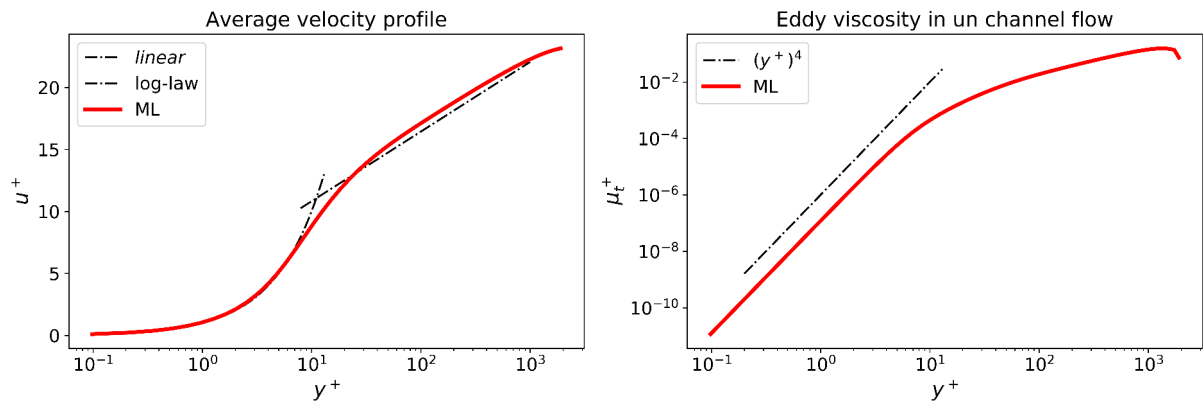
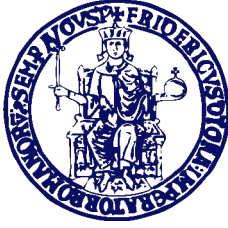



Figure 7: Average velocity profile (on the left) and eddy viscosity (on the right) in a fully developed channel flow. $Re_\tau = 2000$ and $fact = 9$.



JHTDB-Based Analysis of Channel Flow Turbulence

Author: Rosario Donnarumma  **Student ID:** M53/1488

Supervisor: Prof. Gianluca Iaccarino

Department of Mechanical Engineering, Stanford University, Stanford, CA, USA

 Correspondence: rosar.donnarumma@studenti.unina.it

January 17, 2023

Abstract This report presents an analysis of turbulent channel flow using data from the *Johns Hopkins Turbulence Databases* (JHTDB). The simulation was performed using a Direct Numerical Simulation (DNS) approach, with a wall-normal, velocity-vorticity formulation, and with periodic boundary conditions in the longitudinal and transverse directions and no-slip conditions at the top and bottom walls. The simulation was run using a MATLAB code and the results were studied in a sub-area of the $x - z$ plane. The results show the presence of structures typical of turbulent flow called "streaks". The 3D isosurface representation of u and the Reynolds stress normal plots are also presented. Additionally, a comparison of the average velocity profile for a channel flow in linear scale and logarithmic scale is provided, with the latter allowing for the identification of three different zones: the viscous substrate, the log-layer and the wake layer.

Keywords: Turbulent channel flow, Johns Hopkins Turbulence Databases (JHTDB), Reynolds stress, velocity profile analysis.

1 Problem overview

The *Johns Hopkins Turbulence Databases* (JHTDB) provide a wealth of information on turbulent channel flow, which is a type of wall-bounded flow that is commonly encountered in industrial settings. The database is generated using a *Direct Numerical Simulation* (DNS) approach, which allows for the accurate modelling of the flow dynamics. In this simulation, the Navier-Stokes equations are solved using a *wall-normal, velocity-vorticity* formulation. The simulation is performed with periodic boundary conditions in the longitudinal and transverse directions, and no-slip conditions are applied at the top and bottom walls. These conditions are chosen to closely mimic the flow conditions that are typically found in practical applications. By utilising the data from the JHTDB, it is possible to gain a deeper understanding of the complex phenomena that occur in turbulent channel flow, and this information can be used to develop more efficient and effective control strategies.

2 Simulation

In order to be able to develop MATLAB code to solve a turbulent channel flow problem, routines developed by researchers at Johns Hopkins University were used. Those routine use MATLAB web service functions to call JHTDB through the *TurbulenceService Matlab class* which uses the MATLAB intrinsic web service functions to create SOAP messages¹, query the *Turbulence Database*, and parse the results. To properly implement the numerical simulation, the dimensions of the channel and the computational grid need to be defined in detail. Using the documentation on the JHTDB site, a channel with $L_x = 8\pi h$, $L_y = 2h$ and $L_z = 3\pi h$ where h is the half-channel height ($h = 1$ in dimensionless units) is considered and the grid is discretized with $N_x = 2048$, $N_y = 512$ and $N_z = 1536$. In order to carry out the simulation, a MATLAB script based on the one downloadable from the mentioned platform was used. In particular, it was decided to study a sub-area of the $x - z$ plane ($17 < x < 21$, $y = -0.985$ and $1 < z < 2$) at $t = 0$. By default, the MATLAB script uses the following input parameters:

¹SOAP (*Simple Object Access Protocol*) provides a way to communicate between applications running on different operating systems, with different technologies and programming languages.



Simulation parameters	
Viscosity ν	5×10^{-5}
Mean pressure gradient dP/dx	0.0025
DNS time step Δt	0.001
Database time step δt	0.0065
Time stored t	[0, 25.9935]

Table 1: Simulation parameters used by the JHTDB MATLAB code for the solution of the channel flow problem.

Flow statistics averaged over $t = [0, 26]$	
Bulk velocity U_b	0.99994
Centerline velocity U_c	1.1312
Friction velocity u_τ	4.9968×10^{-2}
Viscous length scale δ_ν	$\nu/u_\tau = 1.0006 \times 10^{-3}$
Reynolds number based on bulk velocity and full channel height Re_b	$\frac{U_b 2h}{\nu} = 3.9998 \times 10^4$
Centerline Reynolds number Re_c	$U_c h/\nu = 2.2625 \times 10^4$
Friction velocity Reynolds number: Re_τ	$u_\tau h/\nu = 9.9935 \times 10^2$

Table 2: Simulation parameters used by the JHTDB MATLAB code for the solution of the channel flow problem.

Grid spacing in viscous units	
x direction	$\Delta y_1^+ = 1.65199 \times 10^{-2}$
y direction at first point	$\Delta y_c^+ = 6.15507$
y direction at center	$\Delta y_c^+ = 6.15507$
z direction	0.0065

Table 3: Simulation parameters used by the JHTDB MATLAB code for the solution of the channel flow problem.

3 Results

Observation of the flow shows a combination of low and high velocities (Fig. 1), with the presence of structures typical of turbulent flow (especially for the u and w component) called "*streaks*". These structures are important because they are responsible for a significant part of the momentum transport in the flow. However, in order to represent them correctly, a sufficiently large domain must be used. If too small a domain were chosen, in fact, the streaks would be limited in their formation and it would not be possible to obtain an accurate and complete representation of the flow. Therefore, in order to properly study and understand turbulent flow, it is essential to use a domain of adequate size.

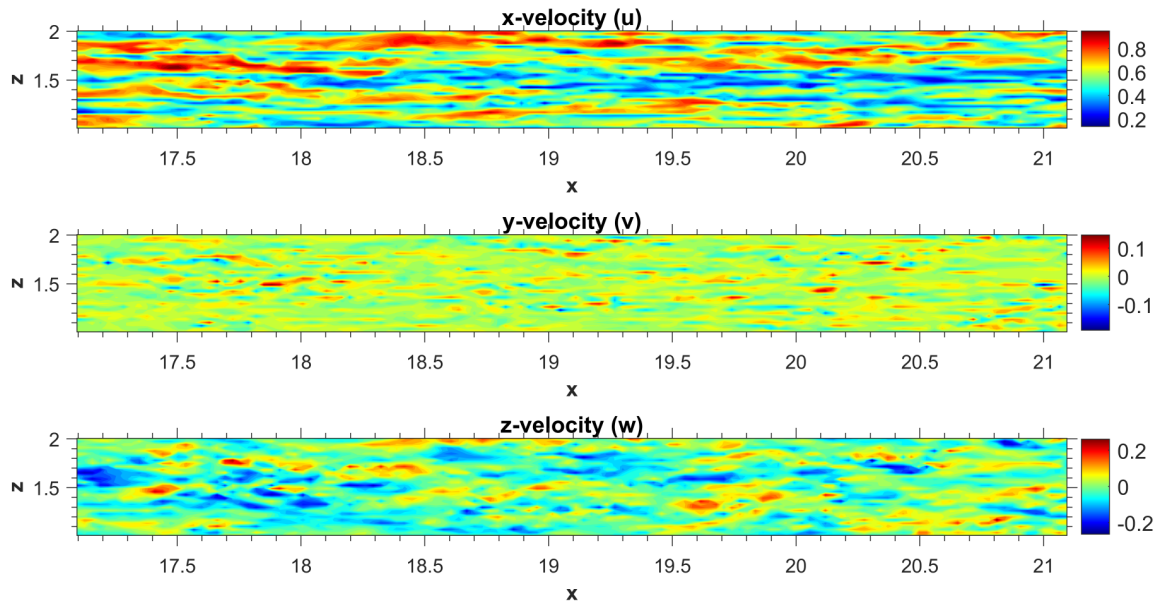


Figure 1: Representation of velocity components in turbulent flow in the plane $x - z$

For the understanding of turbulence, the 3D isosurface representation of u can be useful. To obtain this representation (Fig. 2), data were extracted in a subregion of the domain defined by $x \in [0, 8\pi]$ (64 points), $y \in [-1, 0]$ (48 points) and $z \in [0, 3\pi]$ (32 points) at $t = 0$.

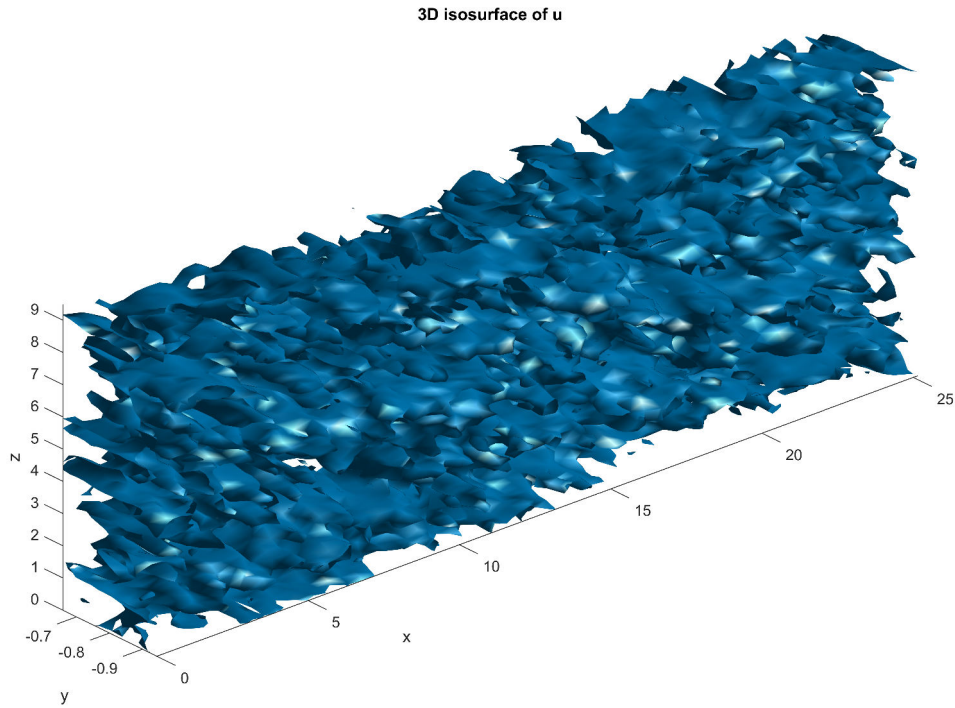


Figure 2: Representation of a velocity isosurface.

Another interesting analysis that can be carried out from the JHTDB is on the evolution of Reynolds stresses. The normal stresses of the tensor have a great importance as they allow for determining the turbulent kinetic energy that represents its strength. The plots in Fig. 3 reveal that at the wall the normal stress is non-existent and this is due to the fact that fluctuations at the wall are absent. Close to the wall, normal fluctuations decrease much faster than fluctuations that are parallel to the wall. As a result, the expected Reynolds stress has a disk-like appearance at the wall.

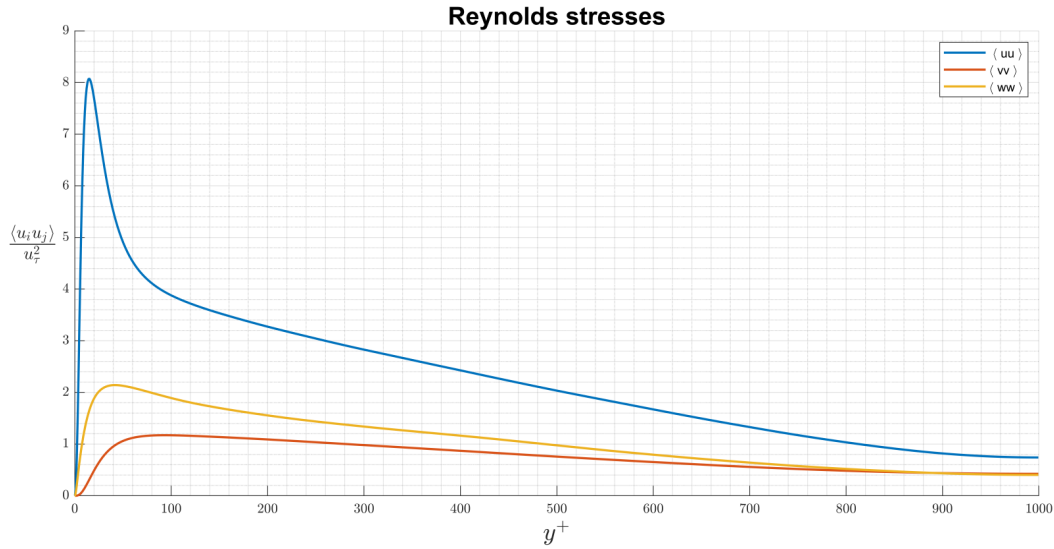


Figure 3: Representation of the normal stress profiles.

In order to complete the discussion, it was decided to also include within this report a comparison of the average velocity profile for a channel flow in linear scale (Fig. 4 - a) and logarithmic scale (Fig. 4 - b). By analysing the average velocity profile on a logarithmic scale (Fig. 4 - b), three different zones can be identified: the *viscous substrate* zone, in which viscous effects are prevalent and the greatest pressure drop occurs; the *log-layer* zone, in which the effect of viscosity is limited and the velocity follows a logarithmic trend; this region is often used as a reference for measuring Reynolds number; and the *wake layer* zone, near the center of the channel, in which viscous effects become less and less significant and greater sensitivity to changes in Reynolds number occurs. This zone is characterised by high turbulence and significant velocity fluctuations. In addition, it is important to note that the transition between different zones depends on the Reynolds number and the properties of the fluid used, so the distinction between different zones may vary depending on the specific study context.

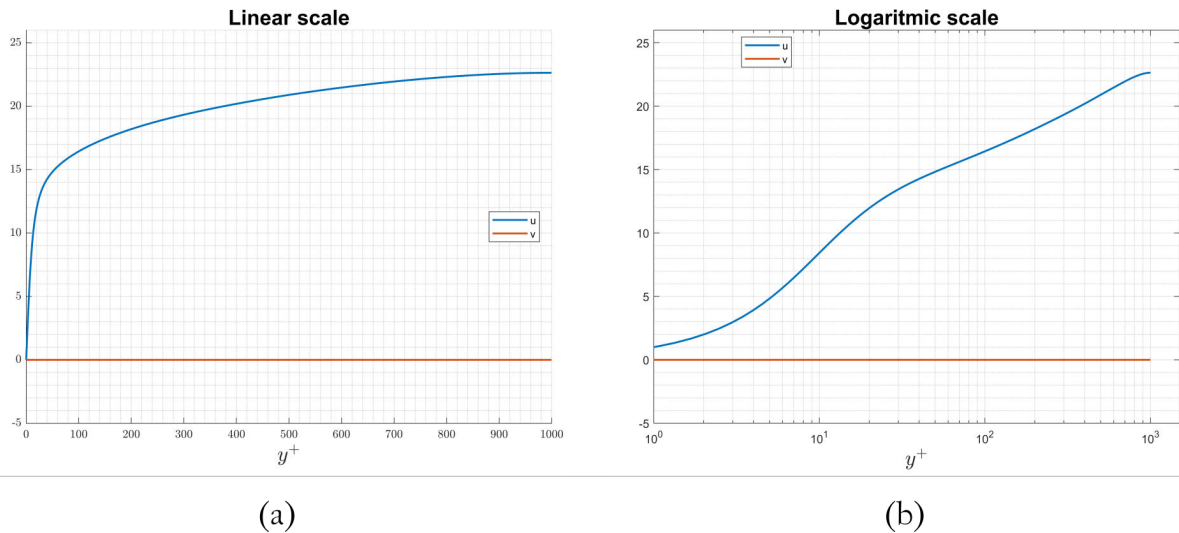


Figure 4: Average velocity profile for a channel flow in linear scale (a) and logarithmic scale (b).



RANS Turbulence Modeling in Ansys Fluent

Author: Rosario Donnarumma ✉ **Student ID:** M53/1488

Supervisor: Prof. Gianluca Iaccarino

Department of Mechanical Engineering, Stanford University, Stanford, CA, USA

✉ Correspondence: rosar.donnarumma@studenti.unina.it

January 24, 2023

Abstract This report discusses a channel flow problem where the flow is fully developed, statistically steady, homogeneous in planes parallel to the wall, incompressible, and studied in a two-dimensional plane. The problem is solved using the Ansys Fluent software, and different turbulence models are implemented, specifically the $\kappa - \varepsilon$ turbulence models and a user-defined function (UDF) model. The report also includes a description of the computational grid used and how changes in mesh density can affect the quality of the results.

Keywords: Turbulent channel flow, UDF, Ansys Fluent, $\kappa - \varepsilon$ turbulence model.

1 Problem overview

In this report, a channel flow problem is discussed and for which the following assumptions are made:

- $Re_\tau = 395$;
- The flow is fully developed away from the inlet section;
- The flow is statistically steady and homogeneous in planes parallel to the wall;
- The flow is incompressible;
- The flow is studied in a two-dimensional plane $x - y$ (2D problem).

The problem that is considered in this case is equivalent to the one studied in Report 1 where the channel flow was solved using a Python script. The substantial difference between the two cases is that in the first one the problem was one-dimensional and the solution for the streamwise velocity in the x direction was found, while in Ansys a 2D problem will be discussed. Just like in the case of Report 1, the following parameters are set: $Re_\tau = 1/\nu$, $u_\tau = 1$, $h = 2\delta = 2$, and that $y^+ = y/\nu$. Therefore the equation to be solved is:

$$\frac{\partial}{\partial y} \left[(\nu + \nu^+) \frac{\partial \langle u \rangle}{\partial y} \right] = -1 \quad (1)$$

The computational grid (Fig. 1) that is used is clearly two-dimensional and it is constructed in such a way that it is denser near the wall. The boundary conditions that are applied are as follows:

- The *no-slip condition* is applied at the wall;
- The solution is assumed to be *periodic* in the x direction, avoiding the variability of the problem;
- At the center of the channel, we have a *symmetry condition*.

Since the RANS equations are involved, clearly there are no variations in time or fluctuation and what is calculated is nothing more than the average solution.

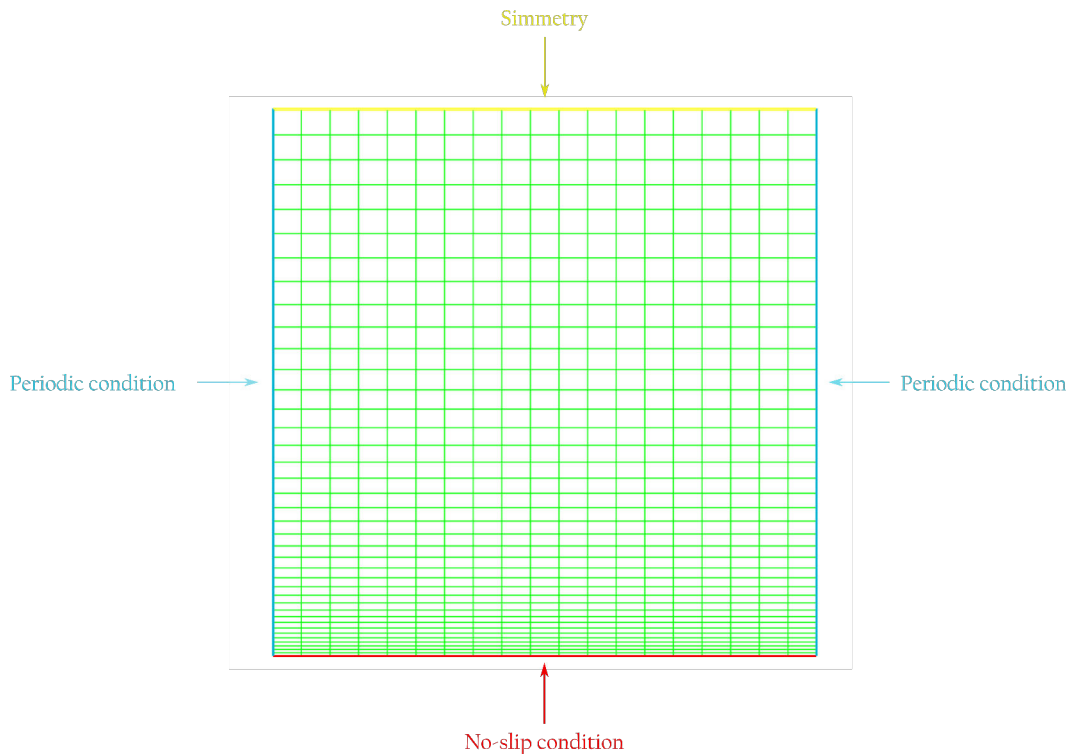


Figure 1: Mesh considered and boundary conditions applied.

2 The turbulence models

The $\kappa - \varepsilon$ turbulence model is a commonly used model in computational fluid dynamic to simulate mean flow characteristics for turbulent flow conditions. It is a two-equation model that provides a general description of turbulence through two transport equations. In this model the first transported variable is the *turbulent kinetic energy* (κ) while the second is the *rate of dissipation of turbulent kinetic energy* (ε). Ansys Fluent uses various types of models to account for turbulent fluctuations and scalar quantities such as:

- The $\kappa - \varepsilon$ models like the *standard*, the *realizable*, and the *RNG*;
- The $\kappa - \omega$ models, including the *standard* and the *SST*.

The $\kappa - \varepsilon$ models are based on the *Gradient Diffusion Hypothesis* and involve equations for turbulent kinetic energy and dissipation rate to determine local turbulent velocity and time-scale. Different near-wall treatment conditions can be applied to these models, and choosing the right one is crucial in predicting the behaviour of the flow near the wall. In the $\kappa - \varepsilon$ models, an *enhanced wall treatment* with damping functions is applied to account for behaviour at the wall.

2.1 The realizable $\kappa - \varepsilon$ model

In this section, a case of a channel flow is being evaluated and an analysis has been performed using the Ansys Fluent software, applying the *realizable $\kappa - \varepsilon$ model*. Air is being chosen as the material, but its properties have been appropriately modified to take into account the operating conditions that were described in the previous section. For this reason, a unit density (ρ) has been assigned, a viscosity of $\nu = 0.0025316$ ¹ and a pressure jump of $\Delta P = -1$ have been assigned. The problem was dealt using a pressure based solver with 1000 iterations. As it is possible observe, the contour generated (Fig. 2-b) reflects the theoretical initial predictions: due to the no-slip condition, the velocity is zero at the wall while due to the periodic conditions, no velocity variations can be observed in the x-direction but only in the y-direction. When the Turbulent Kinetic Energy contour (Fig. 2-a) is examined, it can be observed that, due to the no-slip condition, the energy is zero at the wall. As one moves away from the wall, the energy increases significantly, reaches its peak (which is located between the buffer layer and the logarithmic layer), and then decreases. The same result can also be seen the following plot (Fig. 3) in which the x-axis is $y^+{}^2$.

¹The viscosity is equal to $1/Re_{\tau}$.

²In this case, based on the previous assumptions, $y^+ = y/\nu$ ($\delta = 1$).

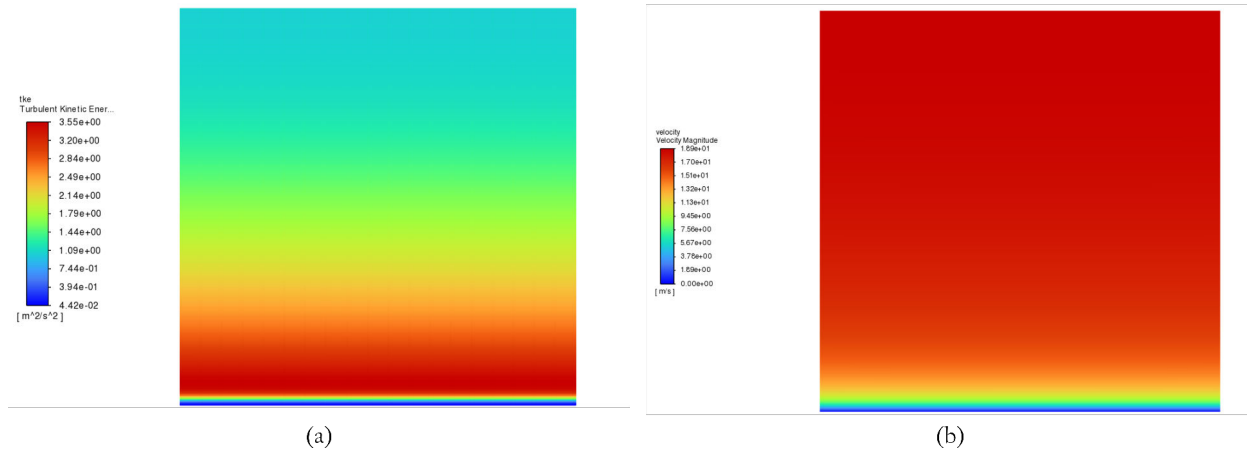


Figure 2: Velocity and Turbulent Kinetic Energy contour.

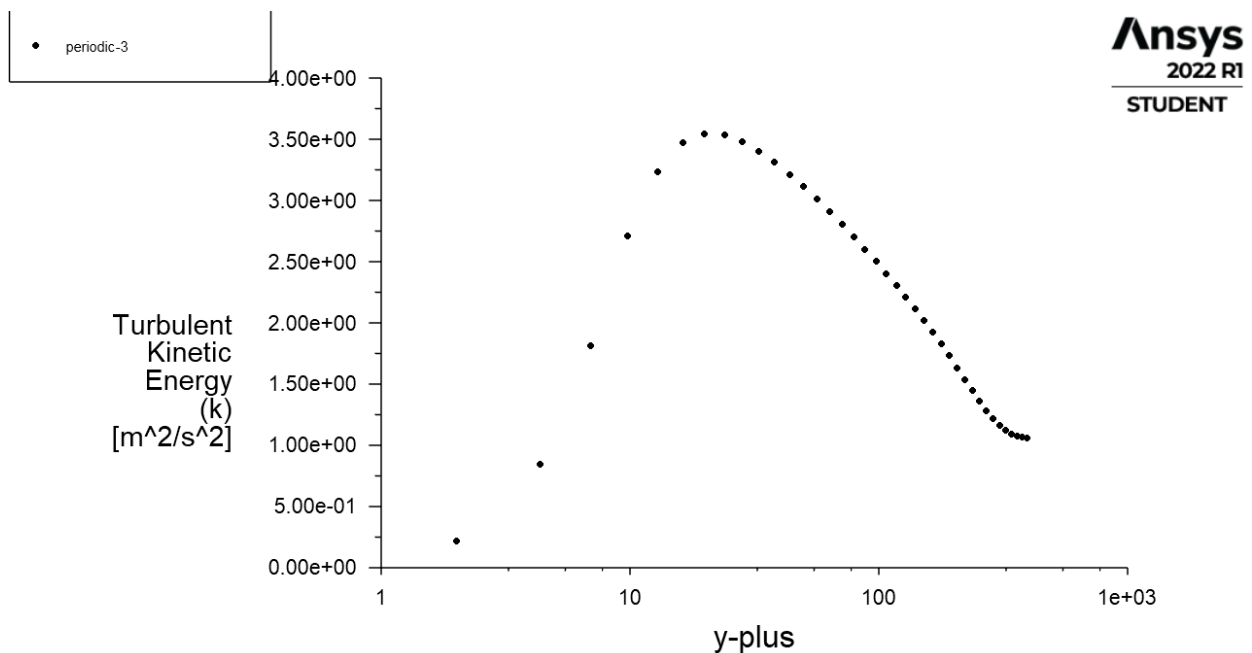


Figure 3: Turbulent Kinetic Energy *semilogx* plot.

2.2 Comparison between different $\kappa - \epsilon$ turbulence model

The $\kappa - \epsilon$ turbulence models that will be discussed are the *standard*, the *realizable* and the *RNG model*. The *RNG $\kappa - \epsilon$ model* is a variant of the standard $\kappa - \epsilon$ model, which introduces the concept of *invariant theory* to ensure universal behaviour of the equations. Unlike the *standard model*, the RNG model introduces a destruction term in the equation related to ϵ . On the other hand, the *realizable $\kappa - \epsilon$ model* derives an expression for ϵ from the dynamic equation of "*mean-square vorticity fluctuation*", which allows for modification of the standard model through the production term. This model has been developed to improve the accuracy of the standard model in describing turbulent flows and simulating phenomena such as flow separation. By observing the plot in Fig. 4, it is possible to see that all the three model give almost the same average velocity profile. But an important point to note is that the models that have been discussed up until now have certain constants that act as adjustable parameters. These constants are included to provide more adaptability to the models and make up for any uncertainty in the assumptions used for modelling. One of these constants is related to the concept of *turbulent viscosity*. It is possible to see a huge discrepancy between the RNG and the realizable model (Fig. 4) because in the first the turbulent viscosity coefficient C_μ is supposed to be constant while in the other it varies.

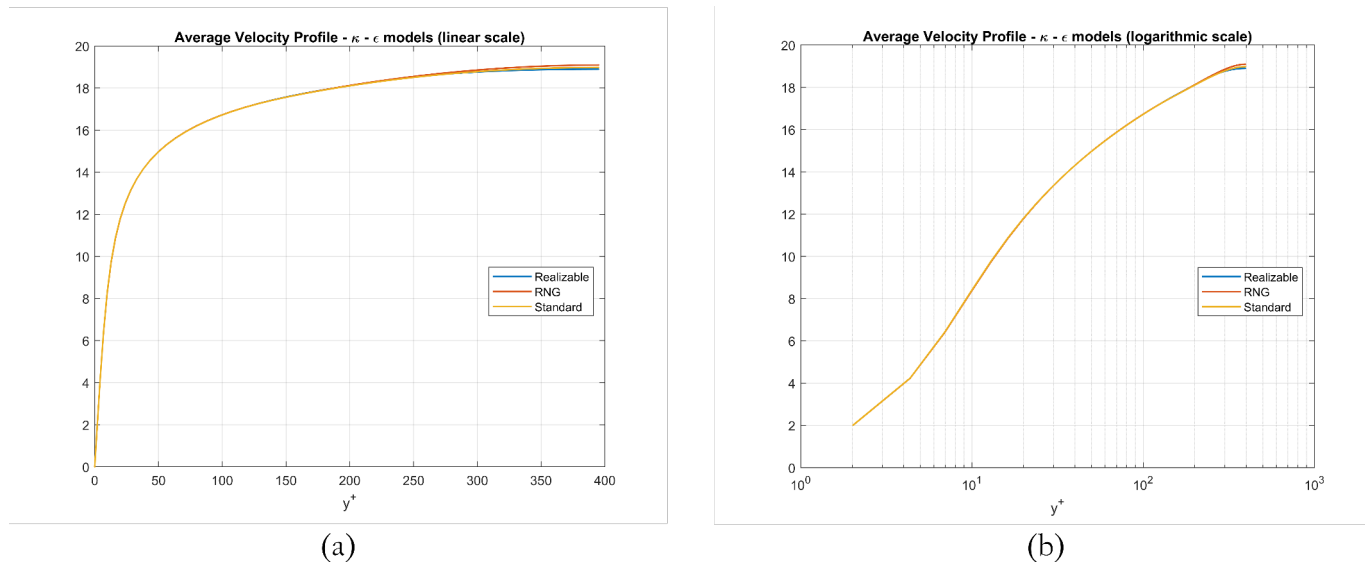


Figure 4: Average velocity profile. A) Linear scale B) logarithmic scale.

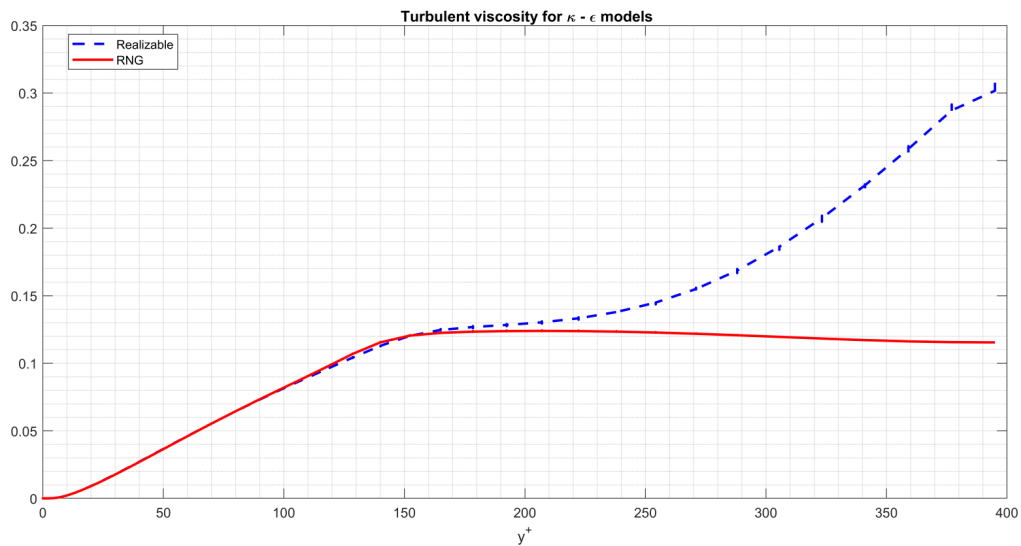


Figure 5: Turbulent viscosity.

3 Grid Sensitivity

In order to improve the results obtained, it is possible to work on the mesh used: in fact, the use of a denser mesh allows an increase in the quality of the results. To appreciate the quality of a numerical simulation carried out through the use of the mentioned models, it is convenient to make a comparison with the DNS results produced in the same assumptions. Taking advantage of the JHTDB, it is possible to verify the results obtained so far. It is important to note that the results stored in JHTDB were obtained at a different Reynolds number. For this reason, the simulation in Ansys will be carried out by going to suitably modify both the material properties³ and the mesh, which will be denser this time.

³ $\nu = 10^{-3}$

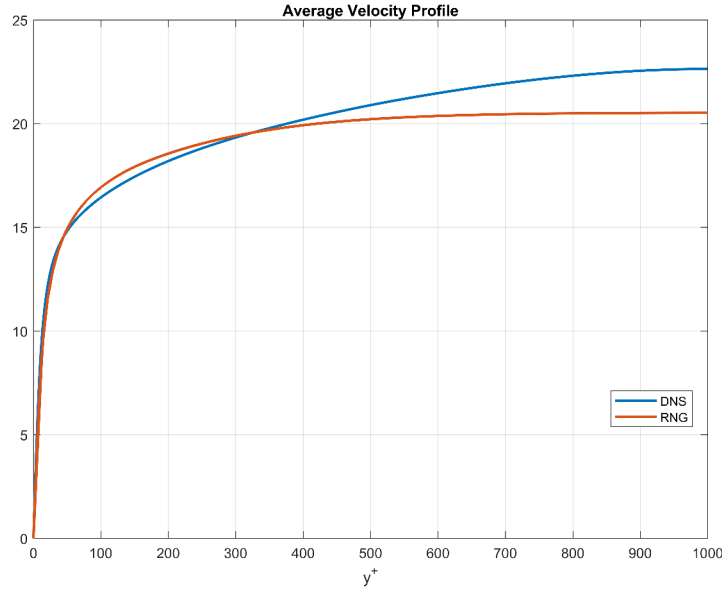


Figure 6: Average velocity profile. DNS vs RNG.

4 User-Defined Functions - UDF

User-defined functions (UDF) are a fundamental feature of programming and represent functions that are directly provided by the user to the software. The use of UDF inside Ansys Fluent allows to implement different turbulence models that are not present by default. It is possible to import UDF coded in C in Ansys Fluent in order to manually tune the simulation: in fact, it is possible to define boundary conditions, the material properties or to post-process the data. In this section, the Low-Reynolds $\kappa - \varepsilon$ turbulent model will be designed using a UDF.

$$\nu^t = f_\mu C_\mu \frac{k^2}{\varepsilon} \begin{cases} \frac{\partial k}{\partial t} + \langle u_j \rangle \frac{\partial k}{\partial x_j} = \nu^t S^2 - \varepsilon + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu^t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \\ \frac{\partial \varepsilon}{\partial t} + \langle u_j \rangle \frac{\partial \varepsilon}{\partial x_j} = \frac{\varepsilon}{k} (f_1 C_{\varepsilon_1} \nu^t S^2 - f_2 C_{\varepsilon_2} \varepsilon) + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu^t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] \end{cases} \quad (2)$$

where $C_\mu = 0.09$, $C_{\varepsilon_1} = 1.44$, $C_{\varepsilon_2} = 1.92$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$. The damping and auxiliary functions are:

$$\begin{cases} f_\mu = [\tanh(0.008 \cdot Re_y)] \left(1 + 4/Re_T^{3/4} \right) \\ f_1 = 1 \\ f_2 = [1 - 2/9 \exp(-Re_T^2/36)] [1 - \exp(-Re_y/12)] \end{cases} \quad (3)$$

where $Re_T = k^2/\nu\varepsilon$, $Re_y = \sqrt{k}y/\nu$ and the boundary conditions are $k_{\text{wall}} = 0$ and $\varepsilon = 2\nu \frac{\partial \sqrt{k}}{\partial y}$. It is now possible to compare the results obtained using the RNG model and the ones from the model with UDF. The plots in Fig. 7 and Fig. 8 show the comparison for the average velocity profile and the turbulent kinetic energy.

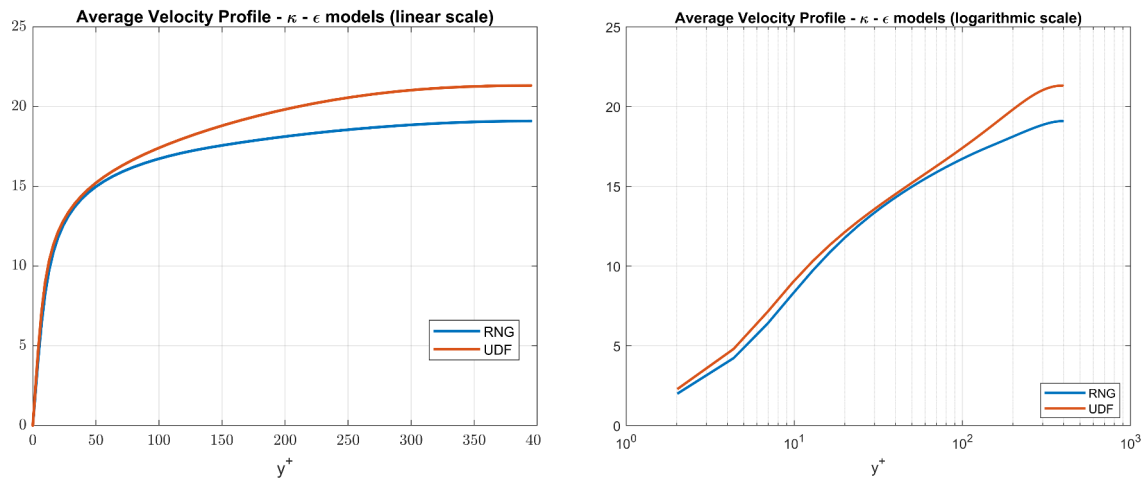


Figure 7: Average velocity profile. RNG vs UDF.

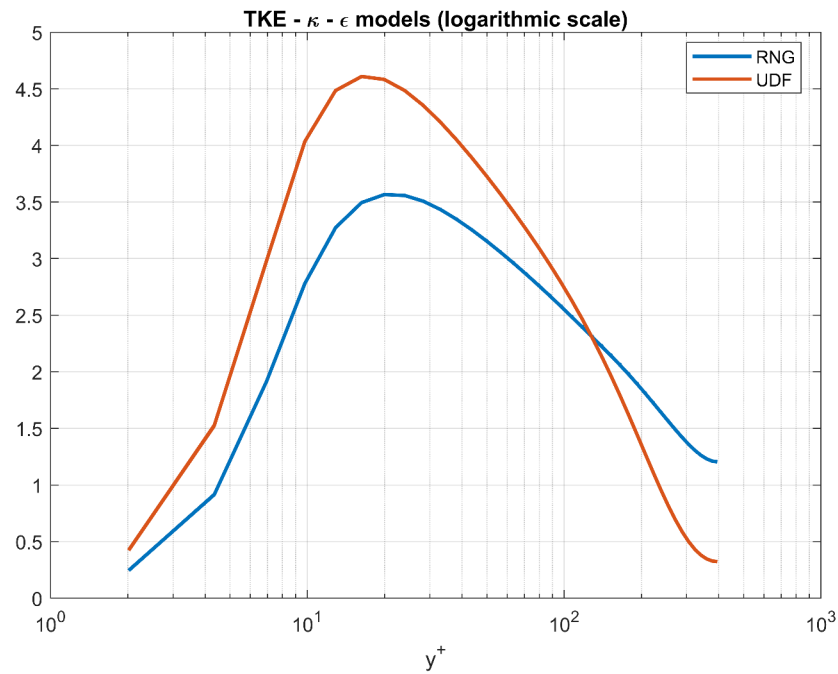


Figure 8: Turbulent kinetic energy. RNG vs UDF.