Assignment 1

ME412 - Introduction to Computational Fluid Dynamics

Fully Developed Turbulent Channel Flows

Meliha Delalić – Department of Mechanical Engineering

Prof.Dr.Muhamed Hadžiabdić

Sarajevo, December 2024.

Table of Contents

1. About T-Flows CFD Program

T-Flows is a simulation program used for analyzing turbulent, single and multiphase flows. It contains various turbulent models such as different RANS (Reynolds Averaged Navier-Stokes) models, LES (Large Eddy Simulations) models, as well as hybrid RANS-LES approach.

T-Flows is a fluid solver program that does not include any graphical user interface and is navigated through from a terminal. It is written almost entirely in Fortran 2008 and its natural habitat is Linux operating system, however there are no restrictions on which operating system to use.

The main subprograms of T-Flows include: *Generate*, *Convert*, *Divide*, and *Process* where the first three are responsible for preprocessing stage and *Process* handles solving of the governing equations and processing of the input data. More about T-Flows and its build can be read on the Github platform for which the link will be provided at the end of this paper.

2. Introduction

In this paper, we are going to demonstrate and run fully developed turbulent channel flows. The first three test cases include simple channel geometry with periodic boundary conditions and two different approaches to simulations near the wall; integration to the wall, and wall function approach. The last case is backste channel where we will see effects of expansion on the fluid flow. Each of these will be explained in more detail in the separate chapters.

Turbulent model used for all the four test cases is RANS, which will then be compared to DNS results. More about DNS and turbulent modelling is written in the following chapter.

3. DNS and RANS

Direct Numerical Simulation (DNS) numerically solves Navier-Stokes equations for turbulent flow where all turbulent fluctuations are resolved directly without any modelling. This approach requires very fine grids that capture turbulent eddies on Kolmogorov scale, which results in a highly expensive computation method that is not used in industry for practical problems. DNS remains as the method used for studying flows with lower Reynolds numbers for scientific research and for comparison with turbulence models results.

Reynolds Averaged Navier-Stokes Equations model (RANS) is a model that draws the attention to the mean flow and the effect of turbulence on the mean flow properties.

Before applying numerical methods to the Navier-Stokes equations, the velocity terms are being split into the mean velocity and fluctuating velocity terms. Due to the interactions of these fluctuation terms, after time averaging the equations some extra uknown terms arise which require modelling in order to achieve full closure of the governing Navier-Stokes equations. Most of the turbulence methods out there use the Boussinesq hypothesis which states that the Reynolds stress tensor is direcly related to the mean rate of deformation:

$$\tau_{ij} = -\rho \overline{u_i u_j} = \mu_T \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)$$

The new term that needs to be modeled is turbulent dynamic viscosity μ_T $\left[\frac{kg}{m*S} \right]$ which is non-homogeneous and varies in space. One of the most widely used turbulence models is $k - \varepsilon$ model which introduces two main parameters for defining the turbulent viscosity.

k – turbulent kinetic energy $\left[\frac{m^2}{s^2} \right]$

ε – energy dissipation rate $\left[\frac{m^2}{s^3} \right]$

In order to solve for turbulent viscosity, $k - \varepsilon$ model solves Navier-Stokes equations for both of its parameters in order to calculate turbulent viscosity.

$$v_T = c_\mu \frac{k^2}{\varepsilon}$$

Here, c_μ stands for the coefficient of proportionality.

4. CASE 1: Stretched Mesh

The Stretched Mesh test case has a purpose to demonstrate integration to the wall approach that implements stretched cells near the wall. This method integrates the governing equations and solves for the pressure and velocity terms all the way to the walls, accurately capturing the boundary layer effects.

4.1.   Integration to the wall

At low Reynolds numbers, Navier-Stokes equations can easily resolve the entire boundary layer offering highly accurate simultions. However, this process requires extremely fine grids in the near wall region and can be very computationally expensive in both time and memory resources. In cases when Reynolds number is above the treshold value, this approach becomes impossible to conduct.

Therefore, integration to the wall remains as a tool for laminar flow research, and similar scientific purposes.
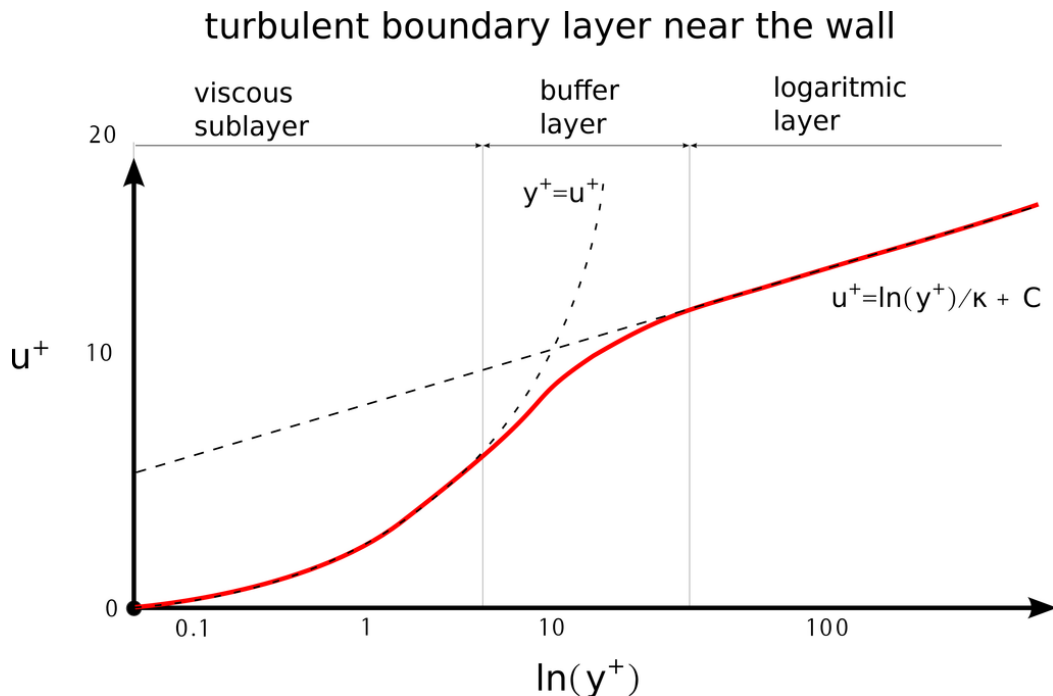

Figure 1 – Boundary layer sublayers, demonstrating high variations in the velocity gradients at the flow inlet.

4.2. PREPROCESSING

The first step in any CFD simualtion is to define the domain of interest, and specify its main spatial parameters, as well as the critical points for observation. This information is found in the .dat extension files and for this specifis case, the domain is defined as follows:

```
#------------------------------------------#
#
#     7----------8
#    /|         /|
#   5----------6 |
#   | |        | |
#   | |        | |
#   | |        | |
#   | 3 - - - -| 4
#   |/         |/
#   1----------2
#
#------------------------------------------------#
#  Nodes (cells), boundary cells and sides  #
#------------------------------------------------#
  37000 10000 100000

#----------#
#  Points  #
#----------#
8
  1   0.0   0.0   0.0
  2   1.0   0.0   0.0
  3   0.0   1.0   0.0
  4   1.0   1.0   0.0
  5   0.0   0.0   1.0
  6   1.0   0.0   1.0
  7   0.0   1.0   1.0
  8   1.0   1.0   1.0

#----------#
#  Blocks  #
#----------#
1 # line1 - number of nodes in x, y and z directions, 1 less for cells
  # line2 - cluster coefficient -0.51, to create narrow cells in the negative       direction
  1   6       6        31
      1.0    1.0      -0.51
      1  2  3  4  5  6  7  8
```

Figure 2 – Domain specifications for the Stretched Mesh case.

The domain is defined by the number of its points with all of their coordinates specified in x,y and z direction. The blocks part means that the entire domain is observed as one block as this is a very simple geometry case.

Next step is to generate and divide the mesh for this case. Using the *Generate* and *Divide* submodules, we create a fine mesh that will capture all significant changes in the flow. The newly created cells will serve as separate control volumes for solving the governing equations. Then we divide the mesh for the processing step which will run on two different processors due to the high CPU performance requirements.

Mesh properties are:

Minimum cell volume: $7.75 * 10^{-5}$

Maximum cell volume: $3.55 * 10^{-3}$

Total domain volume: 1.0

Significant mesh volume variations indicate the stretching or refienement that aims to capture the boundary layer phenomena.
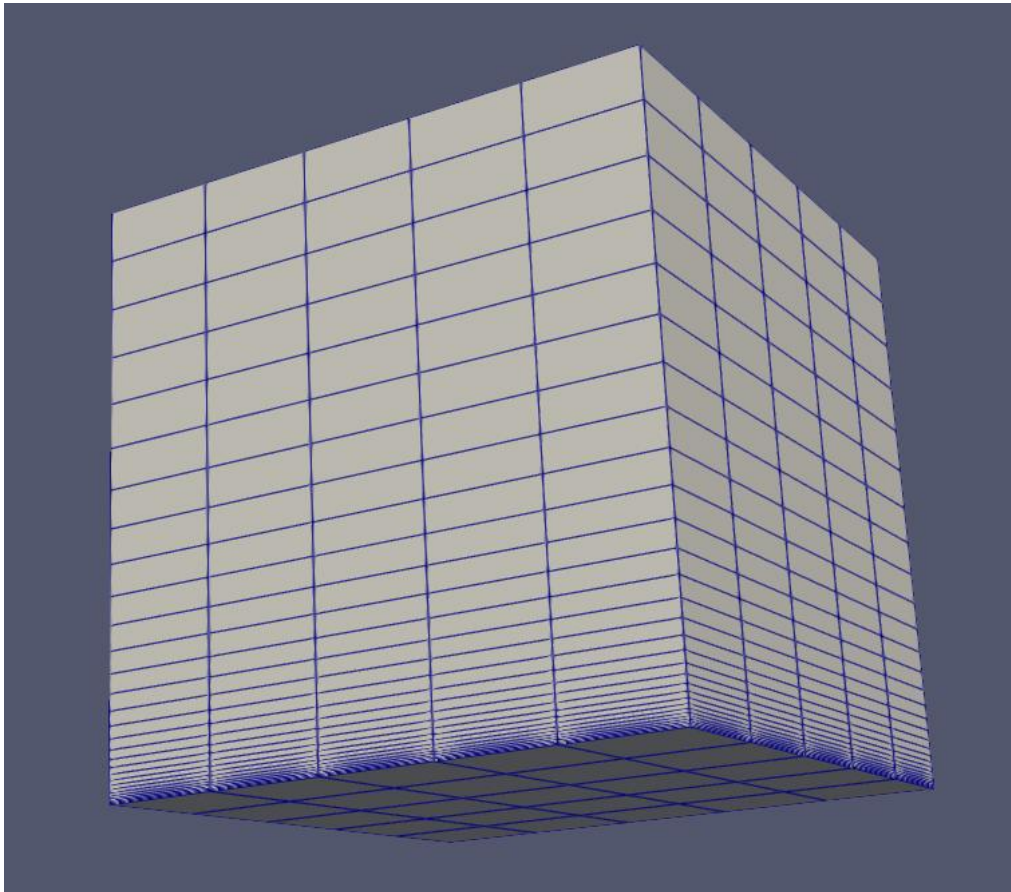

Figure 3 - Generated stretched mesh in Paraview. The number of cells gradually increases going vertically towards the bottom of the mesh to ensure boundary layer effects are captured by finer mesh cells.

The critical part is setting the right boundary conditions.

Two main boundary conditions are LOWER_FACE and UPPER_FACE, which stand for the bottom and the top of the channel domain. At the bottom, we have the wall boundary condition where the primary concern is studying the changes in velocity. The top wall is set to the symmetry boundary condition since the channel is symmetric and we save on computational resources by only simulating the flow in half of the channel.

Periodic boundary conditions are imposed on all the remaining sides of the channel, as this is the case where the domain is assumed to be infinitive.

```
#----------------------#
#  Boundary conditions  #
# (it will use default) #
#----------------------#
 2
   1 Kmin
      1  lower_face
   2  Kmax
      1  upper_face
#----------------------#
#  Periodic boundaries  #
#----------------------#
 2
   1  1  2  6  5
      3  4  8  7
   2  1  3  7  5
      2  4  8  6
```

Figure 4 – Defining the top and bottom faces of the domain and the periodic boundary conditions .

Based on the chosen turbulence model, we also specify the intial conditions for the turbulence parameters such as k and ε, as well as the inlet velocity in x-direction.

```
#--------------------
# Initial conditions
#--------------------
  INITIAL_CONDITION
    VARIABLES       u     v     w     t      kin      eps      zeta      f22
    VALUES         1.0   0.0   0.0  20.0   1.0e-2   1.0e-3   1.0e-1   6.6e-3

#--------------------
# Boundary conditions
#--------------------
  BOUNDARY_CONDITION    lower_face
    TYPE                wall_flux
    VARIABLES       u     v     w     q       kin    eps      zeta   f22
    VALUES         0.0   0.0   0.0   1.0     0.0   1.0e-3    0.0    0.0

  BOUNDARY_CONDITION    upper_face
    TYPE                symmetry
    VARIABLES       u     v     w     t
    VALUES         0.0   0.0   0.0   20.0
```

Figure 5 – Initial and boundary condition for the Stretched Mesh case.

After completing all of the steps above, the preprocessing phase is done and ready for the processing of data and analyzing the simulation results.

4.3. RESULTS AND DISCUSSION – Stretched Mesh Case

In this section we will demonstrate the visuals of the Stretched Mesh case in Paraview, more specifically the velocity and turbulent kinetic energy fields. Also, the simulation results are plotted over the DNS results for a direct comparison with the real data in XMGrace.
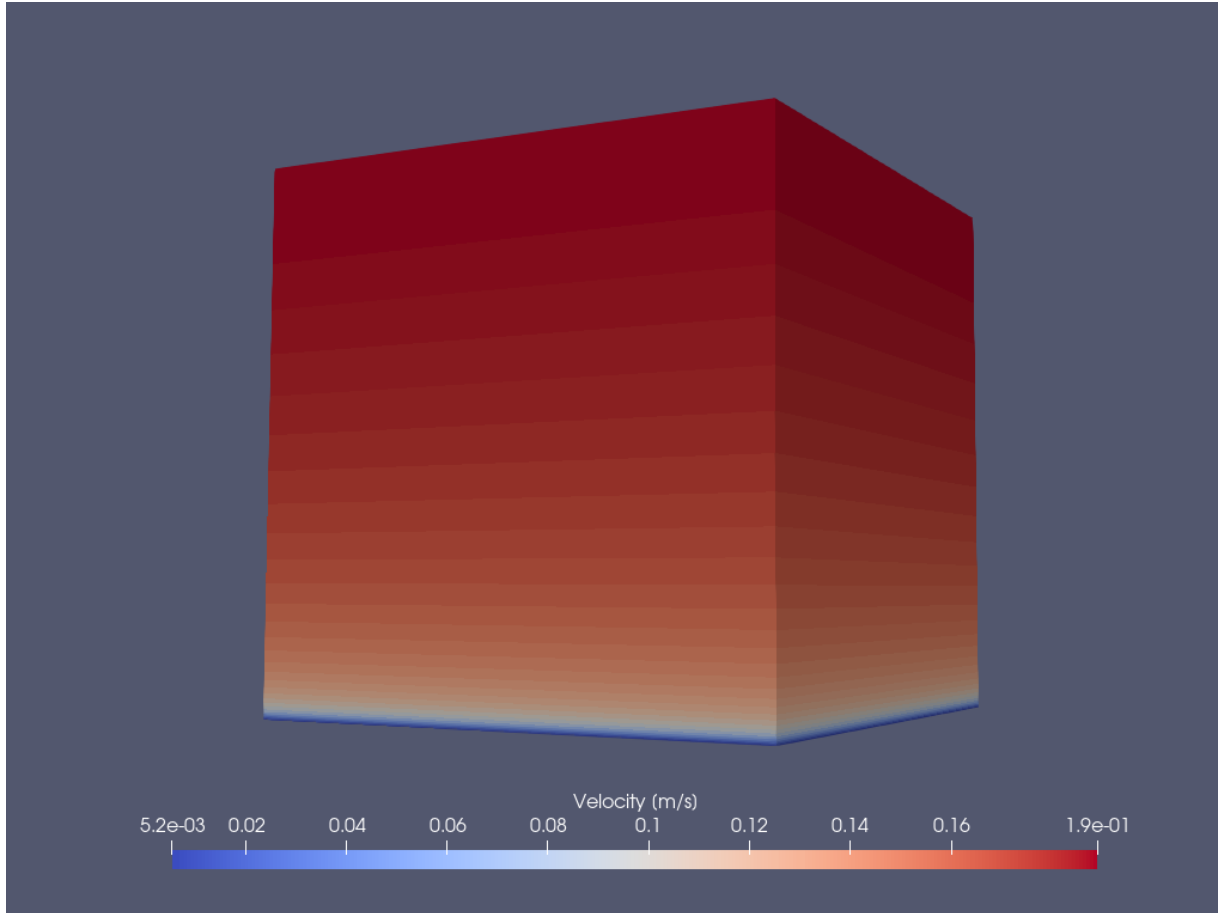

Picture 1 – Velocity field indicating lower velocities at the bottom of the channel due to the No-Slip Condition.

We are already familiar with the no-slip condition where fluid elements right at the wall have zero velocity with respect to the wall due to the viscous forces preventing the flow. This phenomena indicates that at the entrance the velocity of the fluid will be zero and then increase going up as the velocity profile continues to stabilize. The blue layer represent the viscous sublayer where shear stress dominates the flow. Moving up we reach the buffer layer where flow is transitioning from laminar to turbulent, and finally the red area which is the largest portion of the domain represents fully developed turbulent flow region.
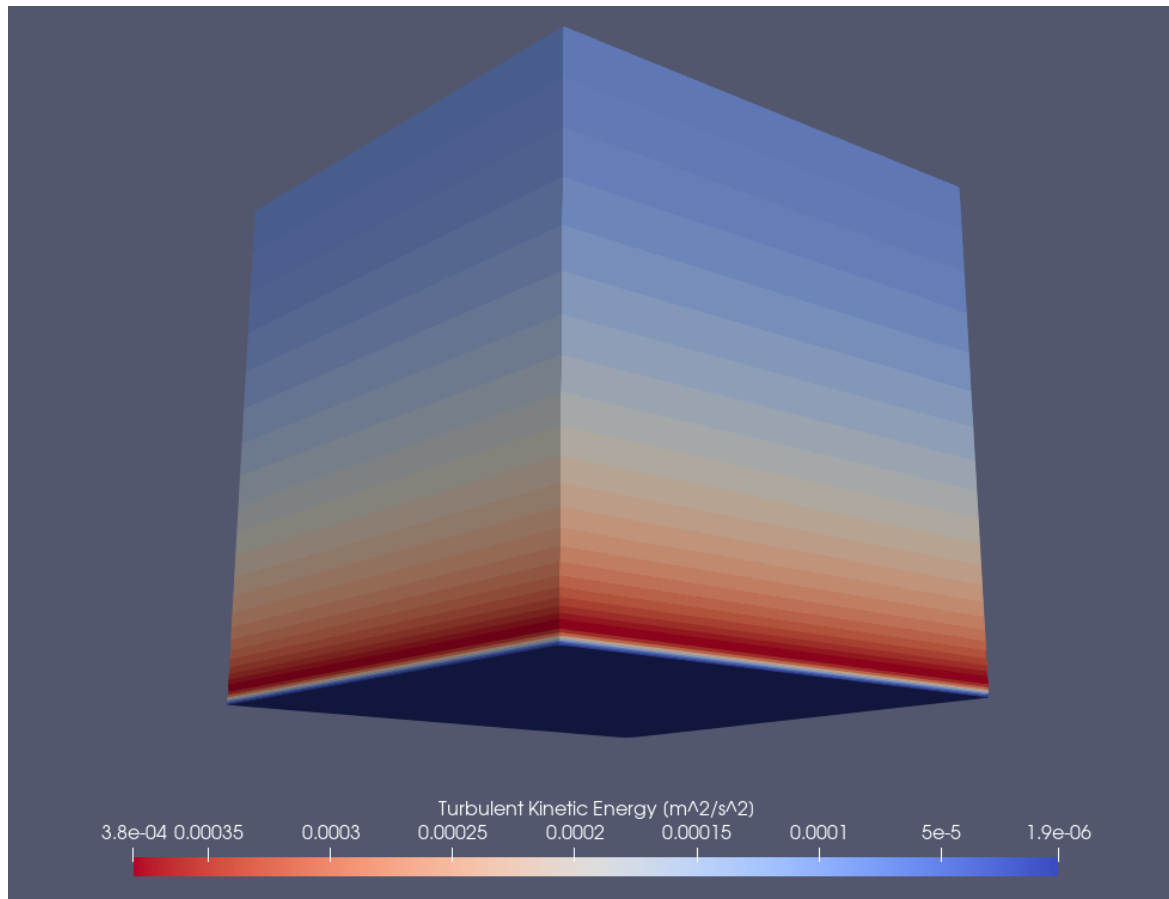
Picture 2 – Turbulent kinetic energy visualisation
demonstrating the decrease of turbulent kinetic energy as we
move further away from the wall.

Turbulent kinetic energy is highest in the logarithmic layer due to large amount of eddie dissipation that results in turbulent kinetic energy. Since turbulent energy is transferred through energy cascade where the larger eddies transfer energy to the smaller ones, as we move away from the wall the energy supplied by the mean flow weakens. It is in the region away from the wall, (wake region), that the turbulent kinetic energy becomes less intense compared to the rest of the domain. In this area, large scale eddies dominate the flow meaning turbulent kinetic energy is less due to lack of small eddies dissipation responsible for transfer of energy through cascade.
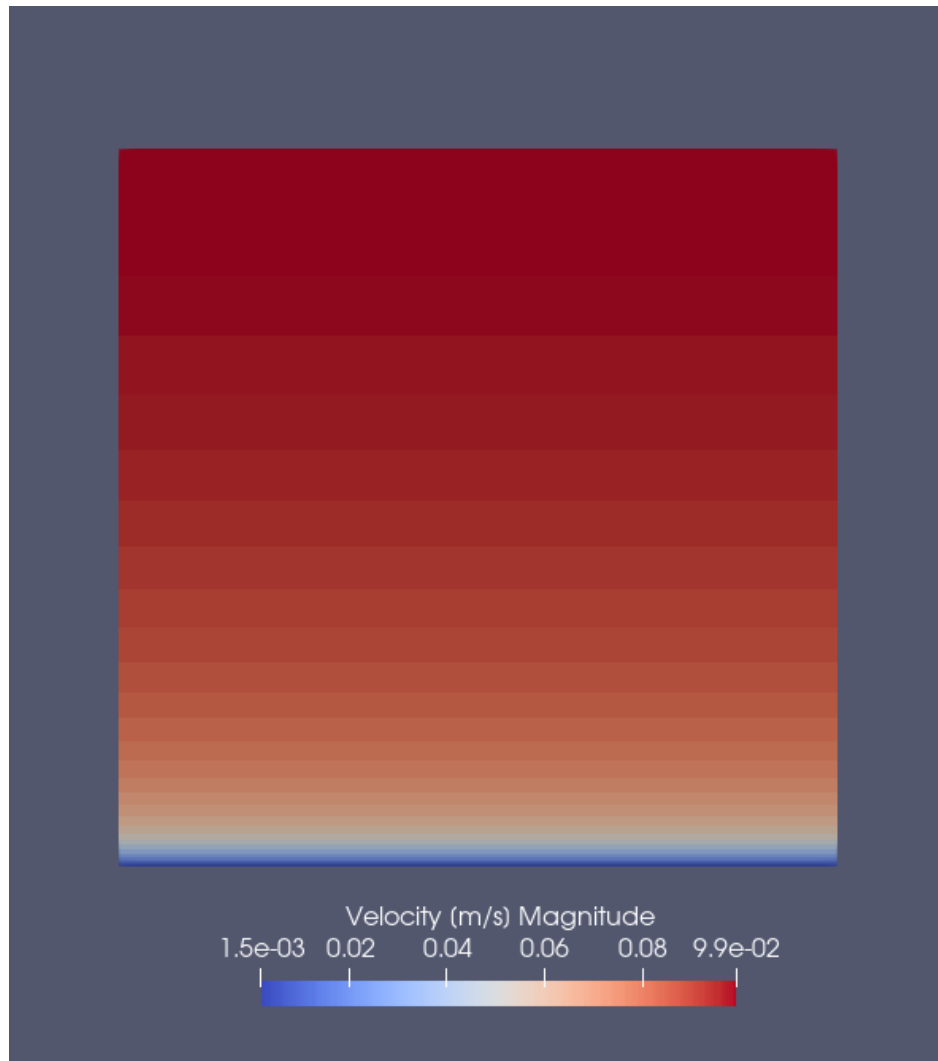
Picture 3 – Central slice through the stretched mesh channel showing effects of boundary layer on the velocity profile.
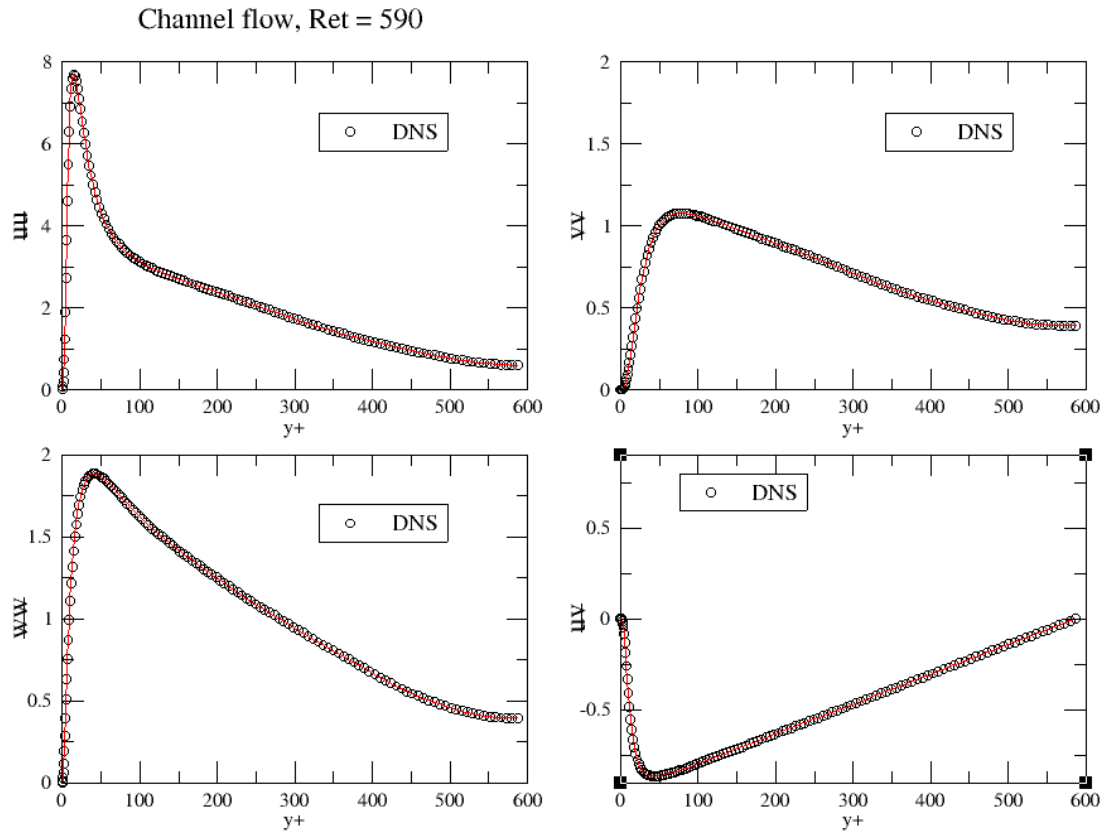
Figure 6 – Comparison of DNS results with the simulation data for velocity x,y, and z components, as well as Reynolds stress component responsible for kinetic energy. X-axis is represented by normalized wall height y^+.

In the figure above, it is clearly seen that our simulation results match very well with the DNS values meaning that this computational method can be used as a good alternative for actual results that are often very hard or impossible to obtain.

5. CASE 2: Uniform Mesh

This case will help us understand the importance of wall functions, and how they can mean a great simplification for problems involving more complex geometry or high Reynolds numbers where it gets difficult or in many cases impossible, to conduct integration to the walls. This case also involves periodic boundary conditions due to the nature of its domain and similar features as Stretched Mesh case.
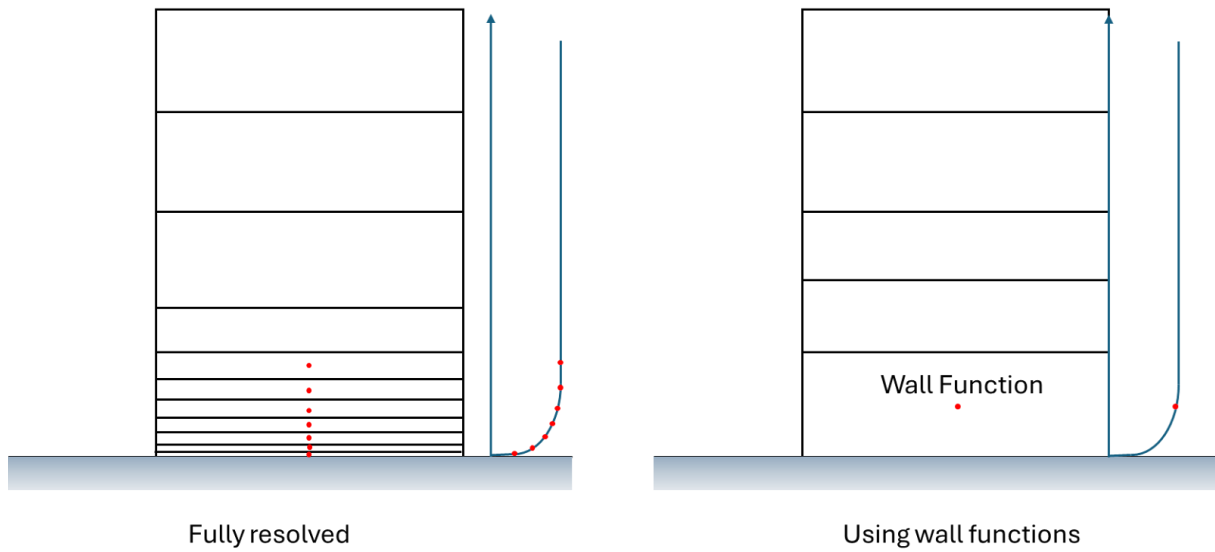

Figure 7 – Boundary layers with fine mesh for intergation to the wall on the left, and using wall function to the right.

5.1. Wall Function

The $k - \varepsilon$ model avoids integration to the wall at high Reynolds numbers using log-law which requires that the rate of turbulence production equals the rate of dissipation. When this assumption is satisfied, we can use the wall function which relates local shear stress to the mean velocity, turbulent kinetic energy and rate of dissipation as:

$$u^+ = \frac{U}{u_\tau} = \frac{1}{\kappa} \ln(E y_P^+) \quad k = \frac{u_\tau^2}{\sqrt{C_\mu}} \quad \varepsilon = \frac{u_\tau^3}{\kappa y}$$

Figure 8 – Wall function describing normalized velocity component u^+, kinetic turbulent energy, and rate of energy dissipation.

This approach is extremely useful in turbulent flows where it is impossible to excplicitly resolve the smallest turbulence scales. Wall function models these turbulence effects and serve as a good practical method where accuracy and cost are balanced.

5.2.   PREPROCESSING

We start by defining the domain for this case, pointing out some of the most important features for generating the mesh.

```
#-------------------------------------------------#
#
#      7-----------8
#     /|          /|
#    5-----------6 |
#    | |         | |
#    | |         | |
#    | |         | |
#    | 3 - - - - | 4
#    |/          |/
#    1-----------2
#
#------------------------------------------------#
#  Nodes (cells), boundary cells and sides  #
#------------------------------------------------#
  37000 10000 100000
```

Figure 9 – Defining the domain for Uniform Mesh case.
Number of nodes: 37000, number of boundary cells: 10000
and sides: 100000.

The information above concerning the domain is found in .dom files, and general information about fluid properties, flow and boundary conditions is found in the control file.

```
#------------------
# Problem definition
#------------------

 PROBLEM_NAME           chan

 HEAT_TRANSFER          yes

 PRESSURE_DROPS         0.0036  0.0  0.0
 MASS_FLOW_RATES        0.2     0.0  0.0


#--------------
# Time stepping
#--------------

 TIME_STEP                 0.6
 NUMBER_OF_TIME_STEPS    1800
 RESULTS_SAVE_INTERVAL    600
 BACKUP_SAVE_INTERVAL     600

# LOAD_BACKUP_NAME    chan-ts001800.backup
```

```
#---------#
#  Blocks #
#---------#
1
  1   6       6        11
      1.0    1.0     1.0
      1  2  3  4  5  6  7  8
#--------#
# Lines  #
#--------#
   0
#----------#
# Surfaces #
#----------#
   0
#---------------------#
#  Boundary conditions #
# (it will use default) #
#---------------------#
 2
   1  Kmin
      1  lower_face
   2  Kmax
```

Figure 10 – Control file information.

Figure 11 – Defining the domain as a block with first row representing the number of nodes in x,y and z directions. Second row tells us about simple connection between the nodes and the third row are points of the domain.

Time stepping in control file allows us to define two important files; saving results and backup save in case of program crashing. The results in both cases will be saved after 600 time steps. The final number of time steps is set to 1800, though this number can vary due to accuracy requirements.

Next our domain is divided on two processors for computation, and finally we can see how both of the the two subdomains contain five cells in z direction. The mesh is visualized in Paraview by applying the chan.pvtu files.
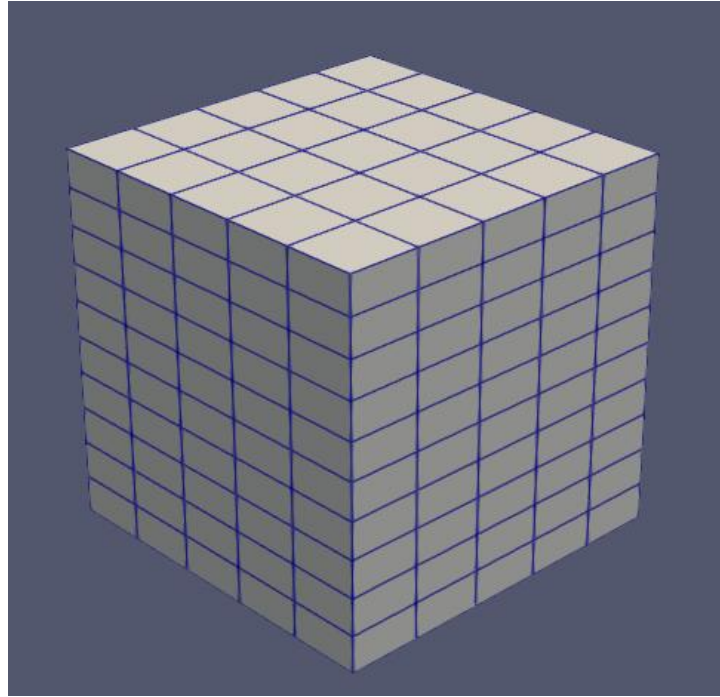
Picture 4 – Uniform mesh in Paraview with both subdomains.

Next, boundary conditions are applied. In this case, the top face is subjected to symmetry boundary condition and the lower face to the wall boundary condition. The main difference between this case and Stretched Mesh case is that now on the bottom side,

```
#---------------------
# Initial conditions
#---------------------
  INITIAL_CONDITION
    VARIABLES      u    v    w    t      kin     eps      zeta    f22
    VALUES        1.0  0.0  0.0  20.0   1.0e-2  1.0e-3   1.0e-1  6.6e-3

#---------------------
# Boundary conditions
#---------------------
  BOUNDARY_CONDITION   lower_face
    TYPE               wall_flux
    VARIABLES      u    v    w    q      kin   eps      zeta  f22
    VALUES        0.0  0.0  0.0  1.0    0.0   1.0e-3   0.0   0.0

  BOUNDARY_CONDITION   upper_face
    TYPE               symmetry
    VARIABLES      u    v    w    t
    VALUES        0.0  0.0  0.0  20.0
```
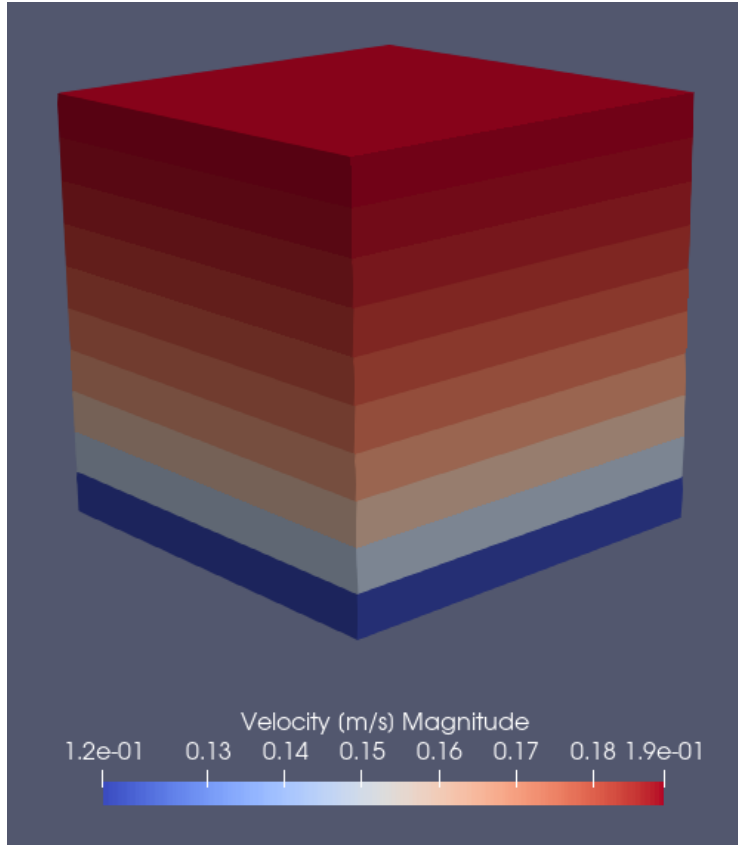
Figure 12 – Boundary conditions for Uniform Mesh case. The same turbulence model is used.

wall function is applied.

5.3.   RESULTS AND DISCUSSION – Uniform Mesh Case

The resulting velocity and turbulent kinetic fields are presented below:

Picture 5 – Velocity field for uniform mesh case, showing structured transition from low to higher velocities starting at viscous sublayer up to the wake region of the flow.

Velocity field behaviour is according to the no-slip condition in the viscous sublayer, and moving away frmom the wall the shear stress influence is reduced hence leading to the increase in velocity. The field is quite similar to that of Stretched Mesh case only now the cells are much sharper due to presence of the wall function in the boundary layer. Applying the wall function no longer requires fine mesh, but a more coarse grid is now appropriate.

The similar scenario is found in the turbulent kinetic energy visual:

Picture 6 – Turbulent kinetic energy field for uniform mesh case. Maximum energy is observed in the buffer layer where there is maximum dissipation of the turbulent eddies.
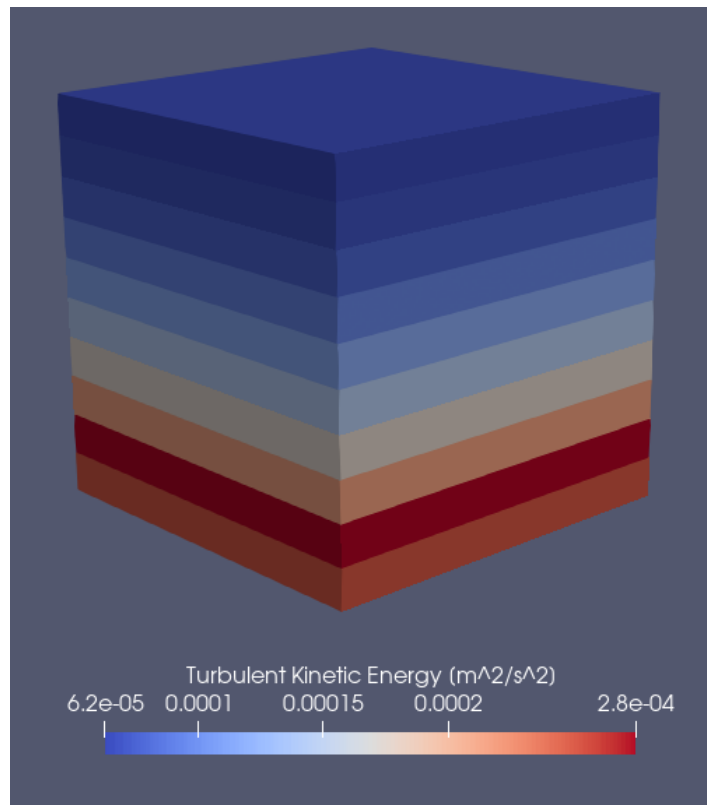
As we can see, wall function assumes no initial layer where there are small values of turbulent kinetic energies. Since it models the behaviour of eddies in the boundary layer, it only provides a rough approximation of turbulent kinetic energy, and therefore maximum values of energy are found just about at the very bottom of the domain.

Moving away from the wall we have again, reduced dissipation of eddies where large scale eddies take place and overpower the domain, resulting in low energy values in the wake region.

Next, we compare the simulation results with DNS values.
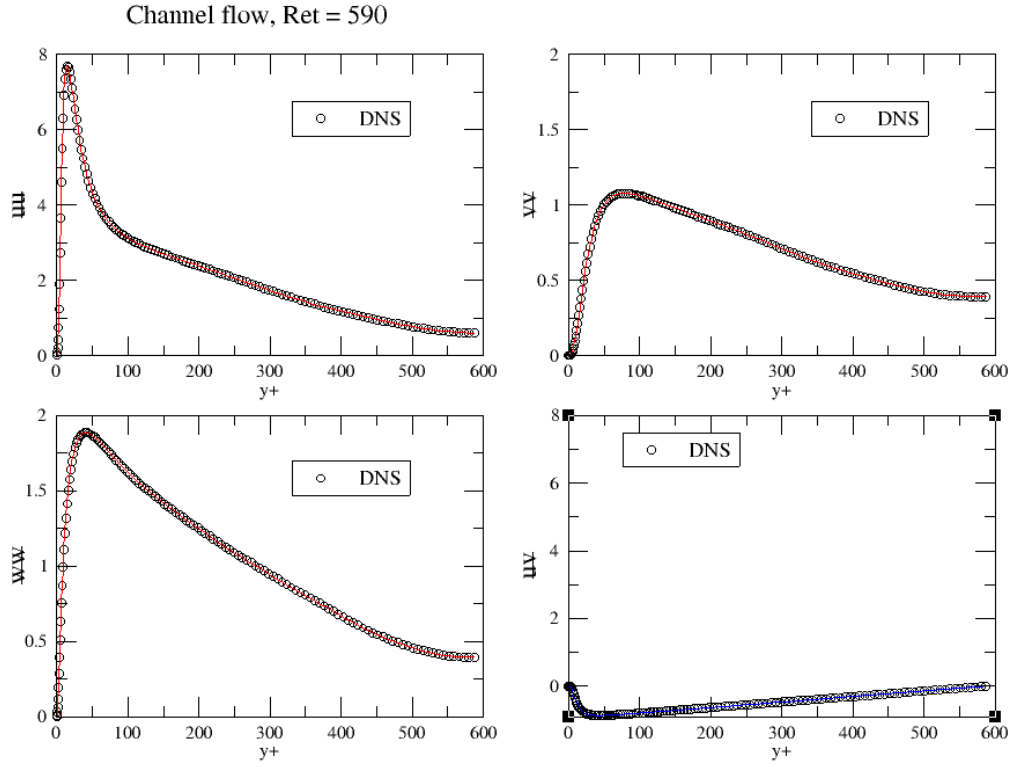
Channel flow, Ret = 590

*Figure 13 – Comparison of DNS values
with uniform mesh simulation results.*
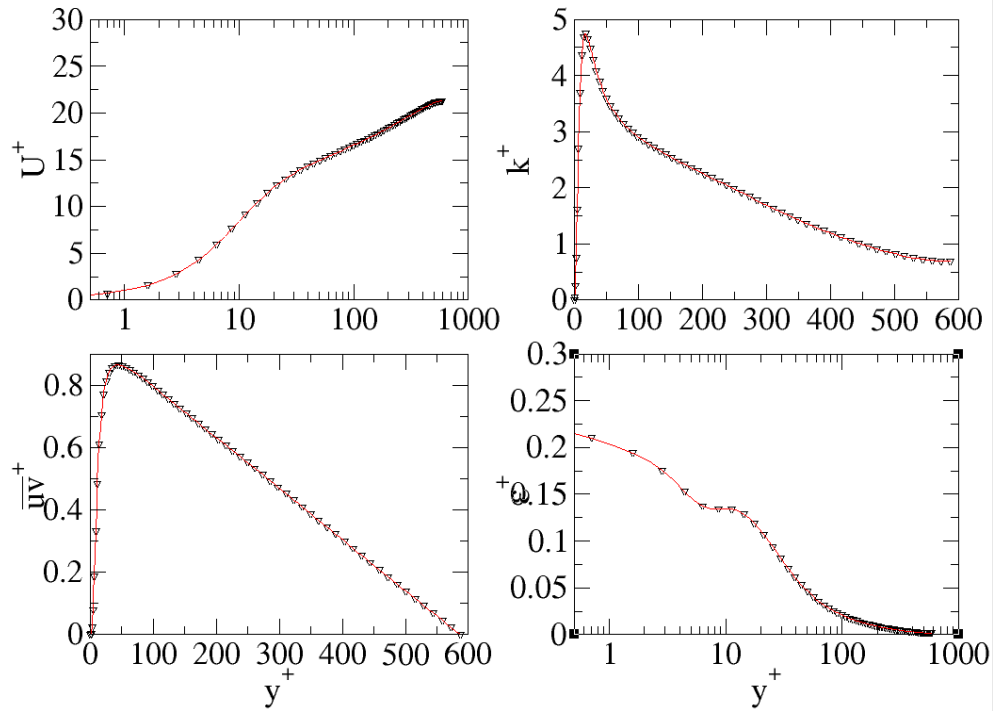
*Figure 14 – Comparison of DNS values with normalized velocity, kinetic energy and
dissipation rate with simulation results with uniform mesh simulation results.*

As for unifom mesh case, the results are primarily compared to the DNS normalized values since this case relies on using wall function. Therefore, comparing the normalized values of velocity, turbulent kinetic energy and dissipation rate with our results show how wall function produces good approximation for real data.

6. CASE 3: Long Domain

In this test case, we will observe the effects of imposed inflow and outflow boundary conditions.

6.1. Inflow and Outflow Boundary Conditions

Inflow boundary conditions are much more challenging than outflow boundary condition since they are convected downstream and have a large impact on the entire flow. It is important to pick the right geometry at the inlet and extend the computational domain further upstream with turbulence-free inflow conditions to reduce any distrotions in the flow. Inflow boundary conditions require having outflow conditions which are way less problematic, as they are easily mirrored from the last computed cells.

These conditions can be either pressure, velocity, mass flow rate, and similar properties depending on the type of the flow.

6.2. PREPROCESSING

As before, we start with defining the domain of the problem. The general information is again found in the .dom file for Long Domain.

```
#
#                                    ^ z
#                                    |
#                                    |    7-------------8
#                                    | /|            /|
#                                    |/ |           / |
#  delta = 1.0                       5------------6  |
#                                    |  3---------|--4
#  L = 2 PI delta = 6.4              | /          | /
#  W = 1 PI delta = 3.2              |/           |/
#  H = 2 delta    = 2.0              1------------2---> x
#
#  Nx = 32 => dx = 0.2
#  Ny = 32 => dy = 0.1
#  Nz min = ?  <=  parametar za tango: 0.94
#
#-----------------------------------------------------------
```
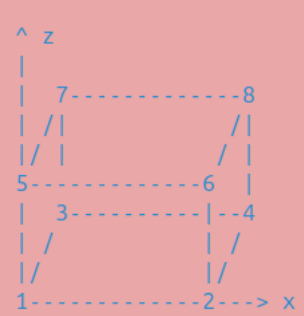
Figure 15 – Long domain description

```
#------------------------------------------#
#  Nodes (cells), boundary cells and sides  #
#------------------------------------------#
 500000 500000 500000

#----------#
#  Points  #
#----------#
8
  1    0.0    0.0    0.0
  2   60.0    0.0    0.0
  3    0.0    1.0    0.0
  4   60.0    1.0    0.0
  5    0.0    0.0    1.0
  6   60.0    0.0    1.0
  7    0.0    1.0    1.0
  8   60.0    1.0    1.0

#----------#
#  Blocks  #
#----------#
1
  1   65   6   31
      1.0  1.0  -0.515
      1 2 3 4 5 6 7 8
```
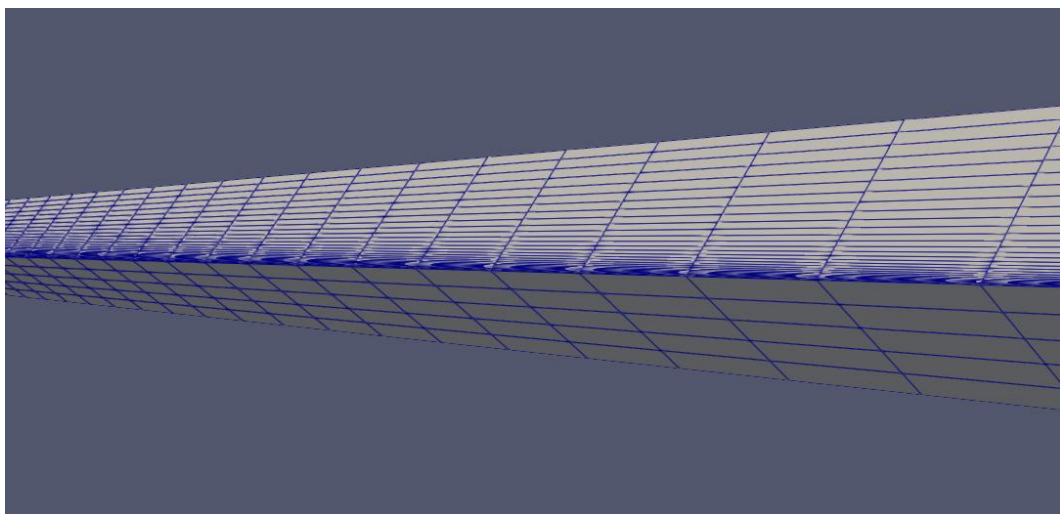
Figure 16 – Long domain – information on the number of
nodes, cells and sides. The domain is defined as one block
with its points and coordinates.

The domain is then divided on four subdomains for generation of mesh. Each of these has a specific number of cells and nodes which are documented during mesh generation.

The resulting mesh is then visualized in Paraview:

Picture 7 – Long domain mesh. Finer cells are found in the bottom
of the channel tor capture boundary layer effects.

Boundary conditions information is found in the control file:

```
#------------------------------------------#
#  Boundary conditions                      #
#------------------------------------------#
   4
   1  Kmin
      1    wall
   2  Kmax
      1    symmetry
   3  Imin
      1    inflow
   4  Imax
      1    outflow

#----------------------#
#  Periodic boundaries  #
#----------------------#
   1
   1  1  2  6  5
      3  4  8  7
```

Figure 17 – Description of each of the four sides and their boundary conditions type. At the bottom of the channel we have wall, the top boundary condition is under symmtery, and then in the x-direction there are inflow and outflow boundary conditions.

```
#--------------------
# Initial conditions
#--------------------
  INITIAL_CONDITION
    VARIABLES        u     v     w     t      kin      eps      zeta     f22
    VALUES           1.0   0.0   0.0   20.0   1.0e-2   1.0e-3   1.0e-1   6.6e-3

#--------------------
# Boundary conditions
#--------------------
wall
wall_flux
symmetry
inflow
outflow
pressure

  BOUNDARY_CONDITION   wall
    TYPE               wall
    VARIABLES          u     v     w     q     kin    eps      zeta  f22
    VALUES             0.0   0.0   0.0   0.1   0.0    1.0e-3   0.0   0.0

  BOUNDARY_CONDITION   symmetry
    TYPE               symmetry
    VARIABLES          u     v     w     t      kin      eps      zeta     f22
    VALUES             0.0   0.0   0.0   20.0   1.0E-2   1.0E-3   6.6e-2   1.0e-3

  BOUNDARY_CONDITION   inflow # must match with the face name
    TYPE               inflow # must be from the b.c. list
    VARIABLES          u     v     w     t      kin      eps      zeta     f22
    VALUES             1.0   0.0   0.0   20.0   1.0E-2   1.0E-3   6.6e-2   1.0e-3

  BOUNDARY_CONDITION   outflow
    TYPE               outflow
    VARIABLES          u     v     w     t      kin      eps      zeta     f22
    VALUES             0.0   0.0   0.0   20.0   1.0E-2   1.0E-3   6.6e-2   1.0e-3
```
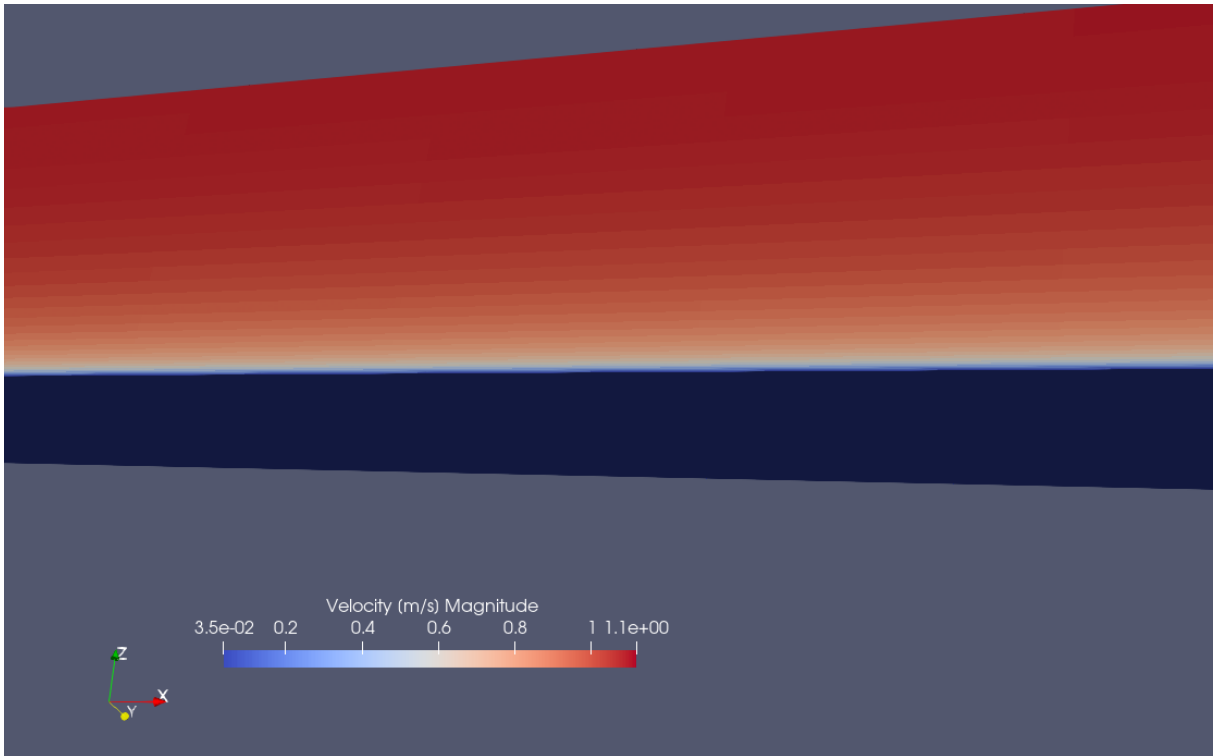
Figure 18 - Initial and Boundary condition details. At the inflow we have initial velocity, and also turbulence kinetic energy is set according to the model we use.

6.3. RESULTS AND DISCUSSION – Long Domain Case

Picture 8 – Velocity field for in long domain case. The velocity gradients are highest in the boundary layer where the viscous effects are present due to no-slip condition.
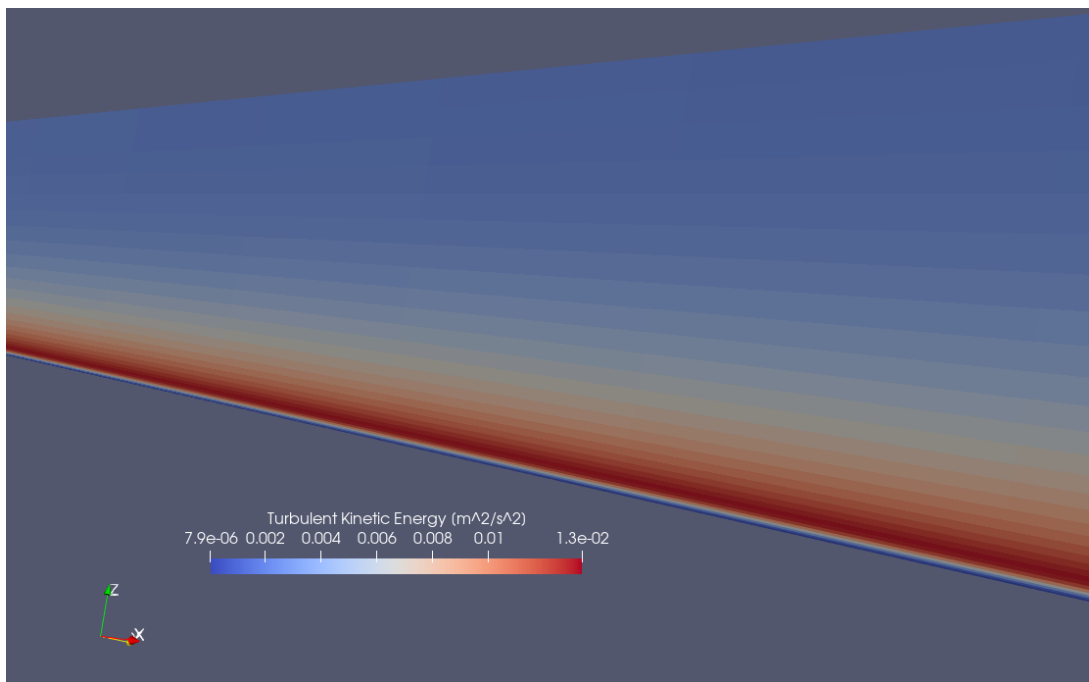
Picture 9 – Turbulent kinetic energy field in long domain case. The reduction in eddie dissipations significantly decreases towards the center of the channel, resulting in lower energy values.

Similarly as in the previous cases, velocity profile depends greatly on the boundary layer in the flow. Moving away from the wall we notice how velocity gradient reduces, hence we obtain a stabilized velocity moving towards the right end of the channel. On the other hand, turbulent kinetic energy has the reverse effect. Highest energy values are found in the logarithmic layer where there is maximum dissipation rate. As the eddies grow and the dissipation effects fade, the energy levels are gradually reduced.

The inflow boundary conditions along with the outflow have produced similar velocity and energy profiles as in the previous cases, whereas in long domain, these periodically occur across the length of the domain.
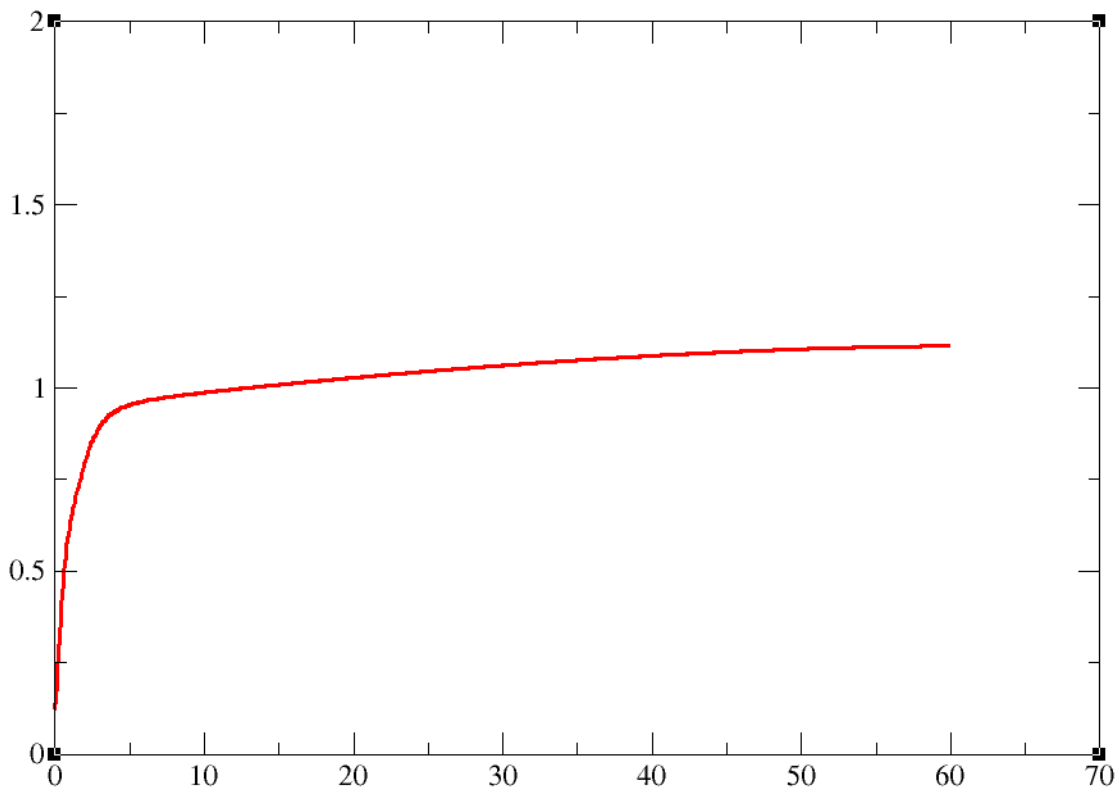

Figure 19 – Boundary layer in the long channel. x- axis represents the wall height and the y-axis is for velocity.

In the long domain case, it is very easy to see the boundary layer and understand its development across the length of the channel. Thanks to no-slip condition, the velocity at the inlet or beginning of the wall is 0. As we move away from the wall, the velocity profile develops into more stable profile where velocity gradients get smaller and smaller.
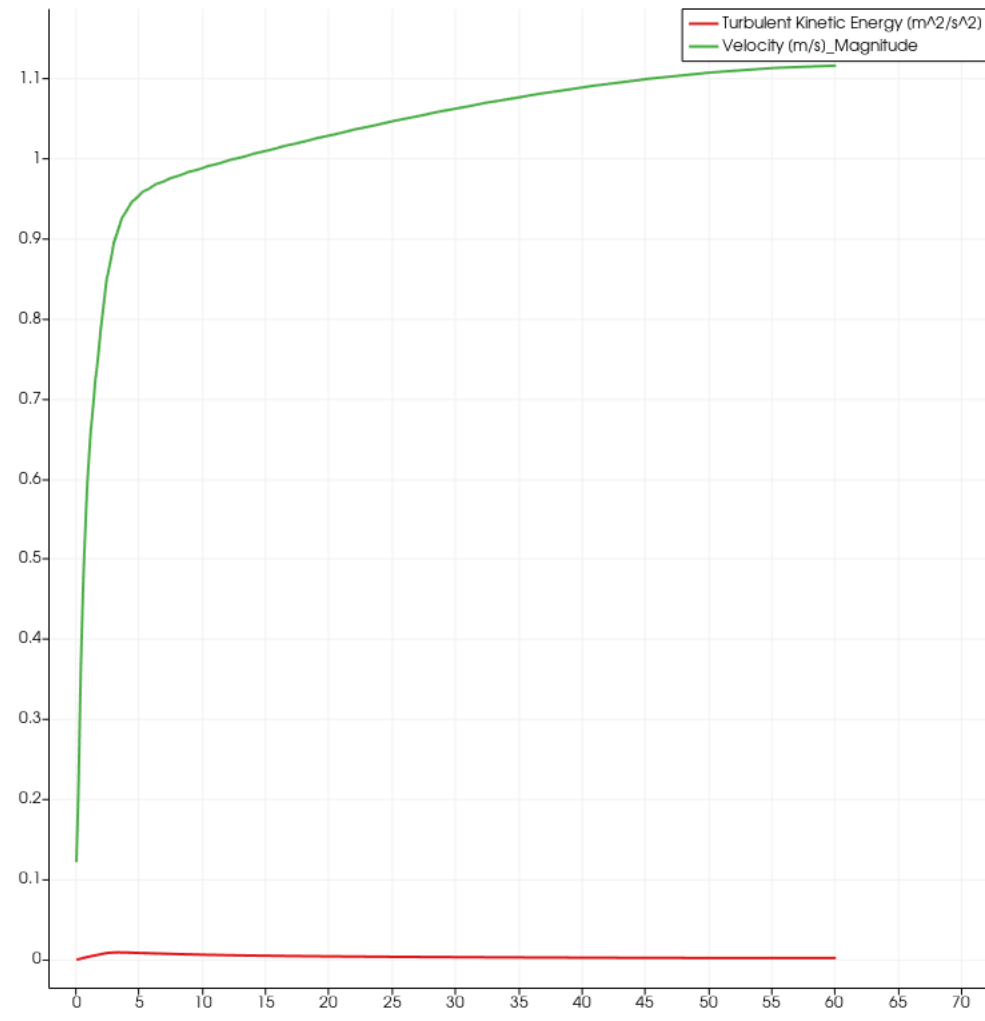
Figure 20 – Comparison of velocity and turbulent kinetic energy profiles in the long domain case. X-axis represents the wall distance in y direction.

Velocity profile for the long domain follows the expected path defined by no-slip condition. The fluid elements right at the wall have zero velocity which means they will shear with the incoming fluid elements creating drag force that slows down the flow near the wall. But as we move away from the wall, these effects lose their dominance and the velocity gradients are much smaller before velocity stops changing at all right at the center of the y-axis, and the velocity profile becomes fully developed.

Turbulent kinetic energy, however, only sees small changes in the buffer layer where there is maximum eddie dissipation. As this effect weakens, so does the turbulent kinetic energy.

7. CASE 4: Backstep Channel

The main issue discussed in this chapter is flow separation due to sharp corners of the channel. Flow separation occurs any time there is a sudden change in the geomettry of the channel, whether it be expansion or contraction, and these changes are most prominent when these channels have sharp edges. Region of separation also increases with increasing Reynolds number due to increase in inertial forces that accelerate the flow.
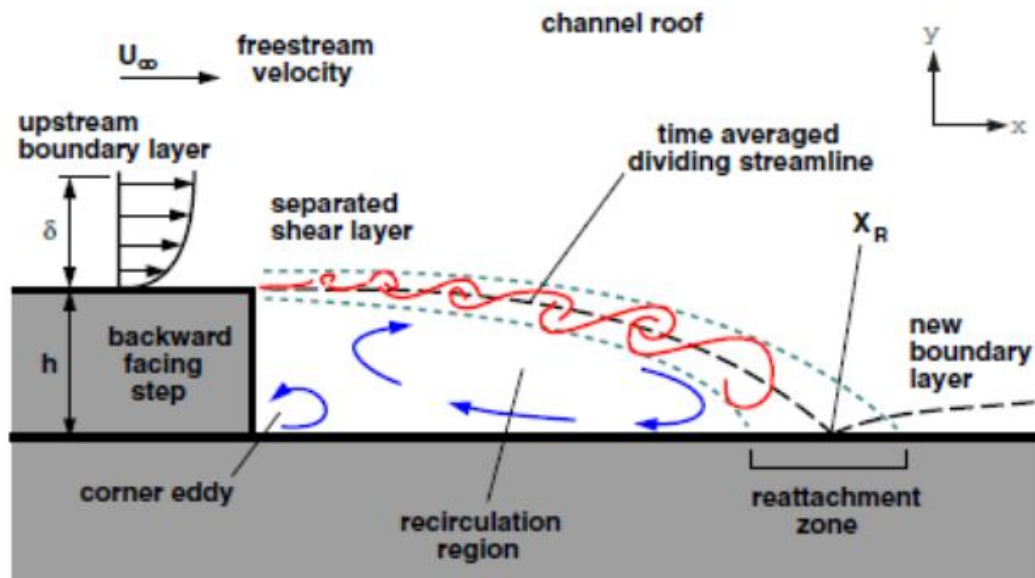
7.1.   Flow Separation

Figure 21 – Flow separation due to sharp edges.

In the figure 20, the flow starts with initial freestream velocity and boundary layer. As it gets to the sharp edge of the channel, the flow starts to separate and small eddies start to form right in the area bounded by sharp edges, also known as recirculation region. This is the area of low pressure where the velocity of the flow is decreased due to increase in pressure. This bubble only exists for as long as the fluid is not reattached to its original path in the reattachment zone where new boundary layer starts to form.

Therefore, we can expect that turbulent kinetic energy achieves its maximum right in the separation region, and velocity in this area should hit its lowest point.

This phenomena is demonstarted in the backstep case which will be conducted in the following pages.

7.2. PREPROCESSING

We start this case by defining the domain and mesh generation.

```
%--|-------------------------------------%
%
%    14----------15--------------------16
%    /|           /|                   /|
%  11----------12--------------------13 |
%   | |   (1)   | |          (2)      | |
%   | 6 - - - - | 7- - - - - - - - - -|10
%   |/          |/|                   |/|
%    1----------2---------------------5 |
%           | |              (3)      | |
%           | 8--------------------|-9
%           |/                     |/
%            3--------------------4
%
%-------------------------------------------%
%  Nodes (cells), boundary cells and sides  %
%-------------------------------------------%
  180000 10000 290000

%----------%
%  Points  %
%----------%
16
   1  -0.1444        0.0           0.038
   2   0.0           0.0           0.038
   3   0.0           0.0           0.0
   4   2.28          0.0           0.0

   5   2.28          0.0           0.038
   6  -0.1444        0.1           0.038
   7   0.0           0.1           0.038
   8   0.0           0.1           0.0

   9   2.28          0.1           0.0
  10   2.28          0.1           0.038
  11  -0.1444        0.0           0.19
  12   0.0           0.0           0.19

  13   2.28          0.0           0.19
  14  -0.1444        0.1           0.19
  15   0.0           0.1           0.19
  16   2.28          0.1           0.19
```

As before, the first step is to define the domain of interest. In this case, we have a channel with an inlet block 1 where the fluid enters. Moving to the right, the fluid is facing a sudden expansion in geometry of the channel where the separation of the fluid is expected to occur.

The critical area for this case will be exactly the region at the transition between the block 1 and at the inlet of block 2 and 3.

Figure 22 – Backstep case domain, divided into three blocks 1,2 and 3. The coordinates of each block are also defined in the .dom file.
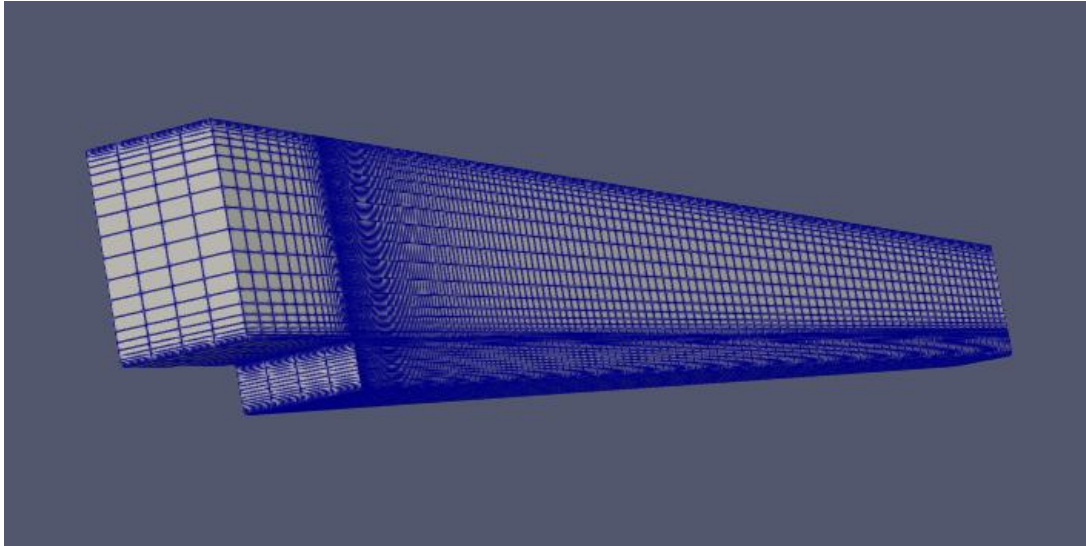
Picture 10 – Backstep channel mesh, inlet side.

Mesh is generated with 180 000 nodes and cells, with 10 000 boundary cells.

The most interesting changes expected to occur are right at the expansion of the channel which is where there are very fine cells needed to capture all the important changes.

As before, the boundary layer region is also of crucial interest for inspection of the flow and therefore finer cells are more concentrated here as well.

Besides the boundary layer in first block of the domain, we are also going to observe the formation of the second boundary layer that is developed after separated flow streams are reattached to the original streams.

The next step in computing the results is dividing the mesh to four subdomains for faster results. The process of saving the subdomains and creating the necessary files took 0.951 seconds which is quite efficient given the complexity of the grid.

We continue this case by defining its boundary conditions.

```
Initial values
  INITIAL_CONDITION
    VARIABLES     u      v      w      t      kin    eps     f22     zeta
    VALUES        8.0    0.0    0.0    20.0   0.01   0.001   0.08    0.01

Boundary conditions
  BOUNDARY_CONDITION     adiabatic_wall
    TYPE                 wall
    VARIABLES            u      v      w      q      kin    eps      zeta    f22
    VALUES               0.0    0.0    0.0    0.0    0.0    0.0004   0.0     0.0

  BOUNDARY_CONDITION     heated_wall
    TYPE                 wall
    VARIABLES            u      v      w      q      kin    eps      zeta    f22
    VALUES               0.0    0.0    0.0    0.1    0.0    0.0004   0.0     0.0


  BOUNDARY_CONDITION     outlet_face
    TYPE                 outflow
    VARIABLES            u      v      w      t      kin    eps      zeta      f22
    VALUES               0.0    0.0    0.0    0.0    0.6    0.0822   1.0004    0.00001

  BOUNDARY_CONDITION     inlet_face
    TYPE                 inflow
    VARIABLES            z  u  v  w  t  kin   eps   zeta   f22
    FILE                 dns_inflow.dat
```

Figure 24 – Assigning each side of the channel to a specific boundary condition.

```
8
  1  Kmax
     1     adiabatic_wall
  2  Kmin
     1     adiabatic_wall
  3  Kmax
     2     adiabatic_wall
  4  Imin
     3     adiabatic_wall
  5  Kmin
     3     heated_wall
  6  Imin
     1     inlet_face
  7  Imax
     3     outlet_face
  8  Imax
     2     outlet_face

%--------------------%
%  Periodic boundaries  %
%--------------------%
   3
   1  1  2  12 11
      6  7 15 14
   2  2  5 13 12
      7 10 16 15
   3  3  4  5  2
      8  9 10  7
```

The bottom face of the block three is subjected to heat, whereas the top sides of blocks three and one are isolated.

Also, the inlet is defined at the start of x-axis, and the outlet is right at the maximum x value.

Periodic boundaries are defined for the front sides of the channel meaning that values at these faces will be periodically calculated.

Some other important information on this

31

case include having 5000 time steps, and a backup file is being created at every 1000 steps. It also includes heat transfer and standard k-eps-zeta turbulence model.

## 7.3. RESULTS AND DISCUSSION: Backstep Channel Case

The resulting velocity and turbulent kinetic energy profiles are shown below:
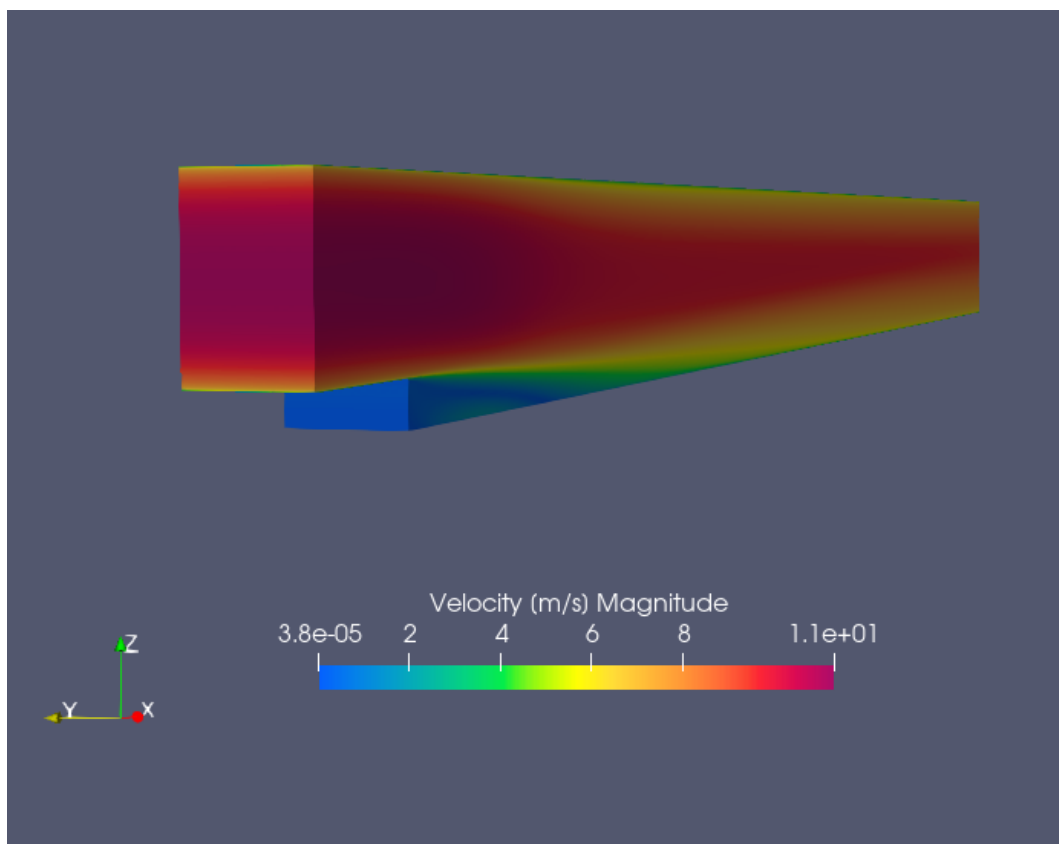
Picture 11 – Development of velocity profile in the backstep case.

At the inlet of the channel, we can see that a very thin boundary layer forms due to high velocity at the inlet. As the flow experiences an expansion in geometry, there is an expansion of the flow resulting in very low velocity area in the separation bubble.

Moving away from the inlet of the channel, the separated fluid again reattaches to the origianl fluid streamlines where velocity is now slightly higher than in the separation region (green area). Finally, as this newly formed fluid is formed, there is again creation of another boundary layer with low velocity near the wall and then increasing away from the wall where the fully developed profile is formed with maximum velocity values in the center of the channel.
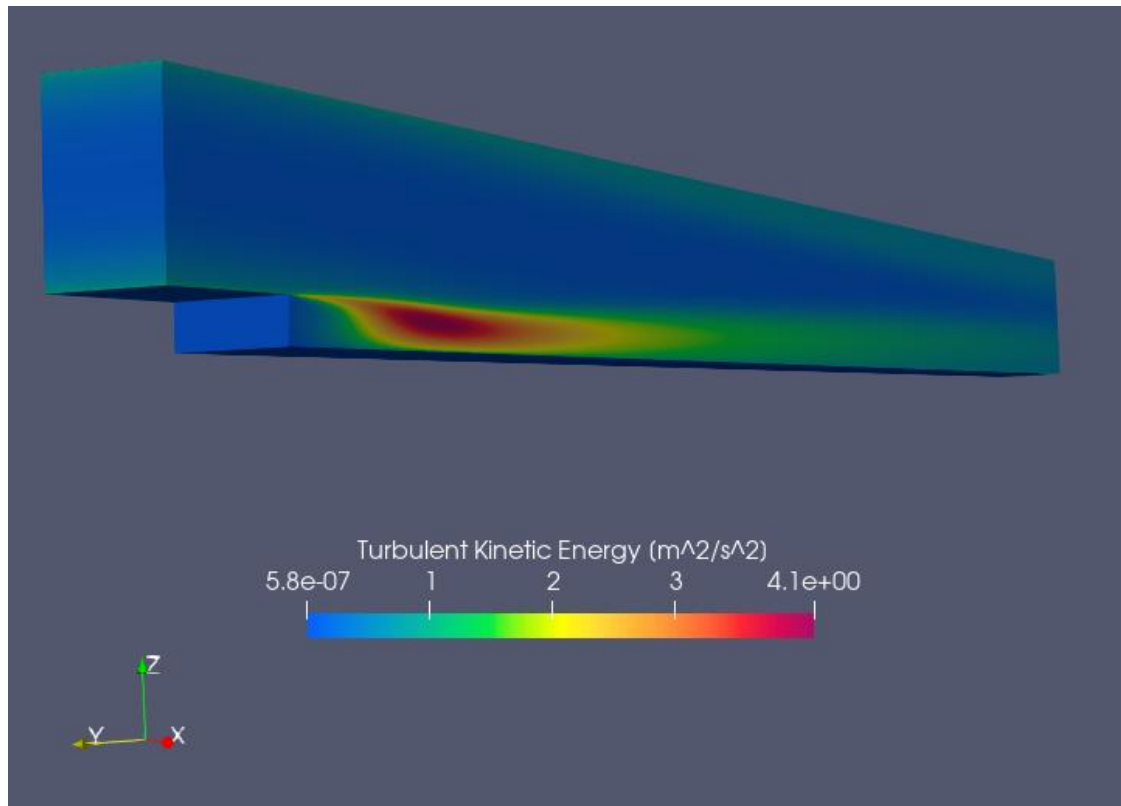
Picture 12 – Turbulent kinetic energy in backstep case.

As expected, maximum turbulent energy is observed right at the separation region where fluid slows down. Just as the flow starts to separate, small eddies start to form underneath this separated stream of fluid and as we move more to the inner area of this region, they dissipate at their maximum. This chaotic movement of eddies in separation region is what creates the maximum values of turbulent kinetic energy.

As the fluid moves away from this region after reconnecting, we can see that the effects of eddie dissipation are dramatically reduced. Velocity experiences smaller gradients which leads to more stable velocity profiles moving to the right end of the channel.

The changes of velocity and turbulent kinetic energy are also visualized on the following plot.
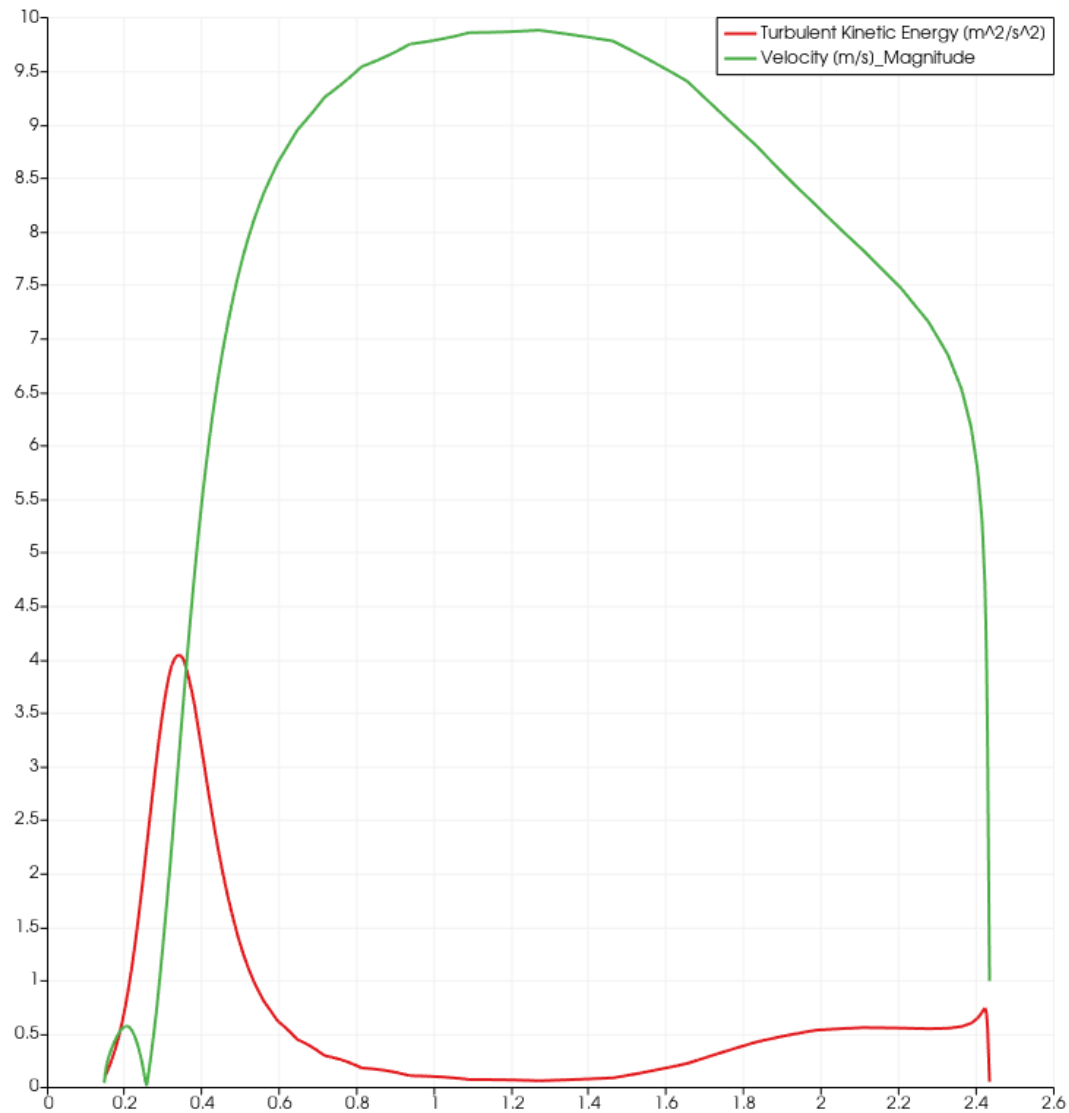
Figure 25 – Changes in velocity and turbulent kinetic energy in the backstep case.

The intial velocity peak is due to stabilization of the initial boundary layer at the inlet of the channel. The dramatic drop that comes afterwards is result of flow separation up until it hits zero at the bottom of the sharp corner whith low pressure vaues. Increase in velocity function is where the fluid is reattached and where new large boundary layer forms. This boundary layer hits its peak at the center of the wall height where the velocity prifle is fully developed.

As for the turbulent kinetic energy, it hits its peak right where the flow is fully separated and then drops rapidly as the fluid streams are reattached.

8. CONCLUSION

In this paper, we inspected the channel flow under different circumstances such as numerical methods used, certain geometries of channels and also boundary conditions.

We saw how simulation results often give a good approximation of the real DNS values which is the reason why CFD remains as an effective approach to solve turbulence flows by modelling. In almost all of the cases, changes in boundary layer are the most significant for study, as they provide us with a direct insight into no-slip condition and how it affects velocity boundary layer, as well as turbulent kinetic energy.

In backstep channel, we saw how a sudden expansion in the geometry of the flow produces a separation bubble, creating a whole new boundary layer after the flow is reattached to the original streams. Additionaly, assigning proper boundary conditions to the flow is crucial for obtaining resepctable results. The accuracy and time within which the computation gets done is dictated by the numerical scheme used. Often times, these include linear or upwind schemes depending on the flow type.

Though the cases in this paper were mostly straightforward and simple to run, they offer us a valuable insight into how more complex problems can be solved using CFD.

9. Table of Pictures and Figures

Picture 1 – Velocity field indicating lower velocities at the bottom of the channel due to the No-Slip Condition.

Picture 2 – Turbulent kinetic energy visualisation demonstrating the decrease of turbulent kinetic energy as we move further away from the wall.

Picture 3 – Central slice through the stretched mesh channel showing effects of boundary layer on the velocity profile.

Picture 4 – Uniform mesh in Paraview with both subdomains.

Picture 5 – Velocity field for uniform mesh case, showing structured transition from low to higher velocities starting at viscous sublayer up to the wake region of the flow.

Picture 6 – Turbulent kinetic energy field for uniform mesh case. Maximum energy is observed in the buffer layer where there is maximum dissipation of the turbulent eddies.

Picture 7 – Long domain mesh. Finer cells are found in the bottom of the channel to capture boundary layer effects.

Picture 8 – Velocity field for in long domain case. The velocity gradients are highest in the boundary layer where the viscous effects are present due to no-slip condition.

Picture 9 – Turbulent kinetic energy field in long domain case. The reduction in eddie dissipations significantly decreases towards the center of the channel, resulting in lower energy values.

Picture 10 – Backstep channel mesh, inlet side.

Picture 11 – Development of velocity profile in the backstep case.

Picture 12 – Turbulent kinetic energy in backstep case.

Figure 1 – Boundary layer sublayers, demonstrating high variations in the velocity gradients at the flow inlet.

Figure 2 – Domain specifications for the Stretched Mesh case.

Figure 3 - Generated stretched mesh in Paraview. The number of cells gradually increases going vertically towards the bottom of the mesh to ensure boundary layer effects are captured by finer mesh cells.

Figure 4 – Defining the top and bottom faces of the domain and the periodic boundary conditions .

Figure 5 – Initial and boundary condition for the Stretched Mesh case.

Figure 6 – Comparison of DNS results with the simulation data for velocity x,y, and z components, as well as Reynolds stress component responsible for kinetic energy. X-axis is represented by normalized wall height y^+.

Figure 7 – Boundary layers with fine mesh for intergation to the wall on the left, and using wall function to the right.

Figure 8 – Wall function describing normalized velocity component u^+, kinetic turbulent energy, and rate of energy dissipation.

Figure 9 – Defining the domain for Uniform Mesh case. Number of nodes: 37000, number of boundary cells: 10000 and sides: 100000.

Figure 10 – Control file information.

Figure 11 – Defining the domain as a block with first row representing the number of nodes in x,y and z directions. Second row tells us about simple connection between the nodes and the third row are points of the domain.

Figure 12 – Boundary conditions for Uniform Mesh case. The same turbulence model is used.

Figure 13 – Comparison of DNS values with uniform mesh simulation results.

Figure 14 – Comparison of DNS values with normalized velocity, kinetic energy and dissipation rate with simulation results with uniform mesh simulation results.

Figure 15 – Long domain description

Figure 16 – Long domain – information on the number of nodes, cells and sides. The domain is defined as one block with its points and coordinates.

Figure 17 – Description of each of the four sides and their boundary conditions type. At the bottom of the channel we have wall, the top boundary condition is under symmtery, and then in the x-direction there are inflow and outflow boundary conditions.

Figure 18 - Initial and Boundary condition details. At the inflow we have initial velocity, and also turbulence kinetic energy is set according to the model we use.

Figure 19 – Boundary layer in the long channel. x- axis represents the wall height and the y-axis is for velocity.

Figure 20 – Comparison of velocity and turbulent kinetic energy profiles in the long domain case. X-axis represents the wall distance in y direction.

10.   Literature and References

Fluid Mechanics Fundamentals and Applications by Yunus A.Çengel and John M.Cimbala, second edition.

An Introduction to Computational Fluid Dynamics by H.K. Versteeg and W.Malalasekera, second edition.

Introduction to CFD Basics by Rajesh Baskaran and Lance Collins

DNS/LES Simulations of Separated Flows at High Reynolds Numbers by P.Balakumar

https://ntrs.nasa.gov/api/citations/20160006024/downloads/20160006024.pdf

Flow over a backward facing step by Harsha Villuri Apr 17, 2021

https://skill-lync.com/student-projects/week-3-flow-over-a-backward-facing-step-13

T-Flows GitHub Repository

https://github.com/DelNov/T-Flows