

Additional Reading Material

Turbulence Modelling in OpenFOAM

Calculation of y_p and y^+

Turbulence models use wall functions to model near wall regions in a turbulent flow. In order to satisfy the physics near the wall, the first cell centre y_p should be within the log-law region from the wall.

The dimensionless wall distance y^+ is given by

$$y^+ = \frac{y_p \sqrt{0.5 C_f U_\infty^2}}{\nu} \quad (1)$$

Where, y is the distance from the wall,
 U_∞ is the free stream velocity, and
 ν is the kinematic viscosity.

The skin friction coefficient C_f is given by the $-1/4$ power law,

$$C_f = 0.078 Re^{-\frac{1}{4}} \quad (2)$$

Where, Re is the Reynolds number.

For the first cell centre y_p to be within the log-law region from the wall, the dimensionless wall distance y^+ should be

$$30 < y^+ < 300$$

In this tutorial, we have considered $y^+ = 200$. Therefore, from (1)

$$y^+ = \frac{y_p \sqrt{0.5 C_f U_\infty^2}}{\nu} = 200 \quad (3)$$

Since the $Re = 2 \times 10^6$ the skin friction coefficient is

$$C_f = 0.078 (2 \times 10^6)^{-\frac{1}{4}} = 2.074 \times 10^{-3} \quad (4)$$

Also, in this tutorial, $\nu = 1 \times 10^{-5} \text{ m}^2/\text{s}$ and $U_\infty = 20 \text{ m/s}$.
Therefore, from (3)

$$y_p = \frac{200 \times (1 \times 10^{-5})}{\sqrt{0.5 \times (2.074 \times 10^{-3}) \times 20^2}} = 0.0031 \text{ m}$$

Therefore, the cell centre of the first cell should be **0.0031 m** away from the wall.
The cell width along y -axis is therefore double this distance, i.e. **0.0062 m**.

Expansion Ratio

The channel geometry in our tutorial is divided into two blocks as shown in fig. 1.

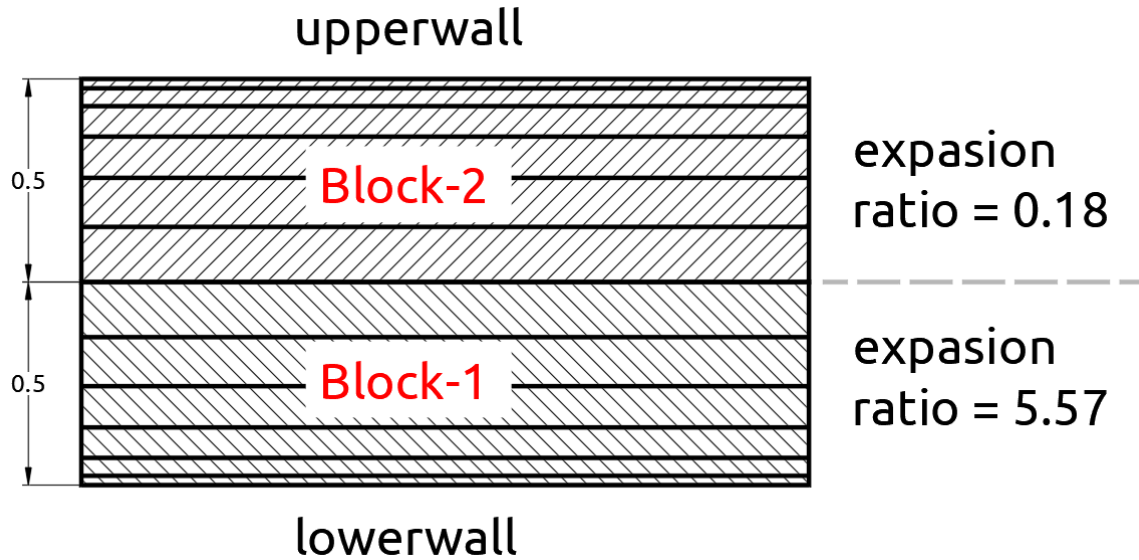


Figure 1. Blocks in the Channel Geometry

The bottom face of block 1 is the wall. Therefore, the cells near the bottom of block 1 needs to be refined. We do this using expansion ratio along y -axis. Let this expansion ratio be e . Since we have prescribed uniform cell expansion using the keyword `simpleGrading` in the `blockMeshDict` file, the cell widths, along y -axis, forms a geometric progression. If the cell width along y -axis of the cell closest to the wall is a , then the cell width of cell above it would be ar and the one above it ar^2 and so on. We have 30 cells running along the y -direction, therefore, the cells width of the topmost cell of block 1 is ar^{29} .

From the previous section we know that cell closest to the wall has a cell width of 0.0062 m.

$$a = 0.0062 \text{ m} \quad (5)$$

Since the channel width of block 1 along the y -axis is 0.5 m,

$$a + ar + ar^2 + \dots + ar^{29} = 0.5 \text{ m} \quad (6)$$

Solving (5) and (6), we get two real solutions for r . Since one of them is $r = 1$ and this means the cell width is uniform, we neglect this solution. The other real solution is

$$r = 1.0611$$

The expansion ratio is defined as the ratio of the width of the last cell to that of the first cell. Therefore,

$$e = \frac{ar^{29}}{a} = r^{29} = 5.57 \quad (7)$$

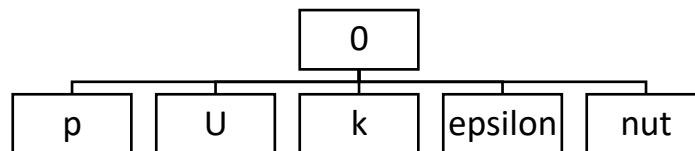
The expansion ratio along y -axis for block 1 is 5.57.

And the same way it is 0.18 for block 2.

$k - \varepsilon$ model

$k - \varepsilon$ model solves two additional equations, for turbulent kinematic energy k and rate of dissipation of turbulence energy ε . It performs poorly for complex flows involving severe pressure gradient, separation, strong streamline curvature. Suitable for initial iterations, initial screening of alternative designs, and parametric studies. Can be only used with wall functions.

To use this model following boundary conditions are required:



Where,

k is turbulent kinetic energy,

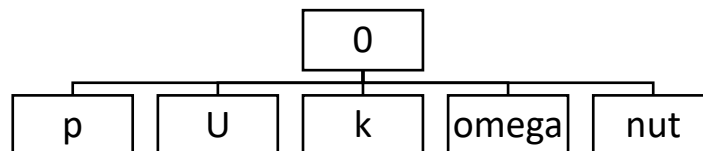
epsilon is turbulent dissipation rate,

nut is turbulent kinematic viscosity

$k - \omega$ model

$k - \omega$ model solves two additional equations, for turbulent kinematic energy k and specific dissipation rate ω . It performs better for complex flows involving severe pressure gradient, separation, strong streamline curvature. For the external flow it works better and near to the walls its performance is not that good.

To use this model following boundary conditions are required:



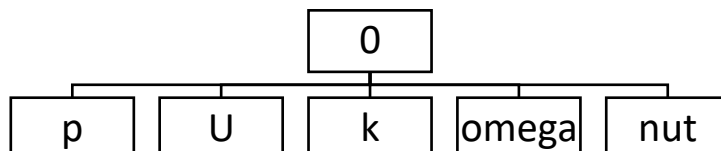
Where,

omega is specific dissipation rate,

$k - \omega$ SST model

This model solves two additional equations, for turbulent kinematic energy k and specific dissipation rate. It gives best performance for wall-bounded boundary layer, free shear, and low Reynolds number flows compared to the $k - \varepsilon$ model. It is a hybrid model of k - ω and $k - \varepsilon$ models. It uses $k - \varepsilon$ formulas in the centre region and k - ω formulas near to the wall. Suitable for complex boundary layer flows. Provides more accurate prediction of flow separation than other RANS models. Can be used with and without wall functions. This is the most widely used RANS turbulence model.

To use this model following boundary conditions are required:



Calculations for k

Turbulent kinetic energy k can be represent by this equation,

$$k = \frac{1}{2} \overline{u' \cdot u'} = \frac{1}{2} \overline{u'^2 + v'^2 + w'^2}$$

If we assume isotropic turbulence $\overline{u'^2} = \overline{v'^2} = \overline{w'^2}$ we will get

$$k = \frac{3}{2} \overline{u'^2}$$

Isotropic turbulence assumption is valid in core region only. Which is far from walls where wall effects are negligible.

Units of k is $\frac{m^2}{s^2} = \frac{J}{kg}$

$$k = \frac{3}{2} (U_\infty I)^2$$

Where,

U_∞ is a characteristic velocity scale,

I is the turbulent intensity

$$I = \frac{u'}{\overline{u}} = \frac{\sqrt{u'^2 + v'^2 + w'^2}}{\sqrt{\overline{u^2} + \overline{v^2} + \overline{w^2}}} = \frac{\sqrt{u'^2 + v'^2 + w'^2}}{U_\infty}$$

Note, that I is a dimensionless quantity.

I typically has a value ranging from [0, 20] %, and [0, 1] % is considered low intensity, [1, 5] % is medium intensity, and [5, 20] % is considered high turbulent intensity.

Turbulent intensity can be predicted from experimental data. For fully developed pipe flow we can use this relation,

$$I = 0.16 Re^{-\frac{1}{8}}$$

<p>In our case,</p> <p>$Re = 20,00,000$</p> <p>$U_\infty = 20 \frac{m}{s}$</p> <p>Turbulent Intensity: $I = 0.16 Re^{-\frac{1}{8}}$</p> <p>$I = 0.16(20,00,000)^{-\frac{1}{8}}$</p> <p>$I = 0.0261$</p>	<p>Turbulent kinetic energy: $k = \frac{3}{2} (U_\infty I)^2$</p> <p>$k = \frac{3}{2} (20 \cdot 0.0261)^2$</p> <p>$k = \frac{3}{2} \cdot 0.272484$</p> <p>$k = 0.40875 \frac{m^2}{s^2}$</p>
---	---

Calculations for ε

Calculation of turbulent dissipation rate ε is very straight forward. Put data of k and characteristic length of the channel in this formula,

$$\varepsilon = \frac{C_\mu^{3/4} \cdot k^{1.5}}{0.07 \cdot L}$$

Where, C_μ is an empirical constant. Value of C_μ is 0.09.

In our case,

$$k = 0.40875 \frac{m^2}{s^2} \text{ \& } L = 1 m$$

$$\text{Turbulent dissipation rate: } \varepsilon = \frac{0.164 \cdot k^{1.5}}{0.07 \cdot L}$$

$$\varepsilon = \frac{0.164 \cdot (0.40875 \frac{m^2}{s^2})^{1.5}}{0.07 \cdot 1}$$

$$\varepsilon = 0.61$$

Calculations for ω

Calculation of specific dissipation rate ω is very straight forward. Put data of k and characteristic length of the channel in this formula,

$$\omega = \frac{\varepsilon}{C_\mu \cdot \kappa}$$

$$\omega = C_\mu^{-1/4} \frac{\kappa^{1/2}}{0.07 \cdot L}$$

Where, C_μ is an empirical constant. Value of C_μ is 0.09.

In our case,

$$k = 0.40875 \frac{m^2}{s^2} \text{ \& } L = 1 m$$

$$\text{Turbulent dissipation rate: } \omega = C_\mu^{-1/4} \frac{\kappa^{1/2}}{0.07 \cdot L}$$

$$\omega = 0.09^{-1/4} \frac{(0.40875)^{1/2}}{0.07 \cdot 1}$$

$$\omega = 16.67$$

Calculations for *nut*

Turbulent viscosity ν_t will automatically be calculated by OpenFoam.

For $k - \epsilon$,

Formula, OpenFoam calculates for ν_t is,

$$\nu_t = \frac{C_\mu k^2}{\epsilon}$$

Where, C_μ is an empirical constant. Value of C_μ is 0.09.

For k - Ω ,

Formula, OpenFoam calculates for ν_t is,

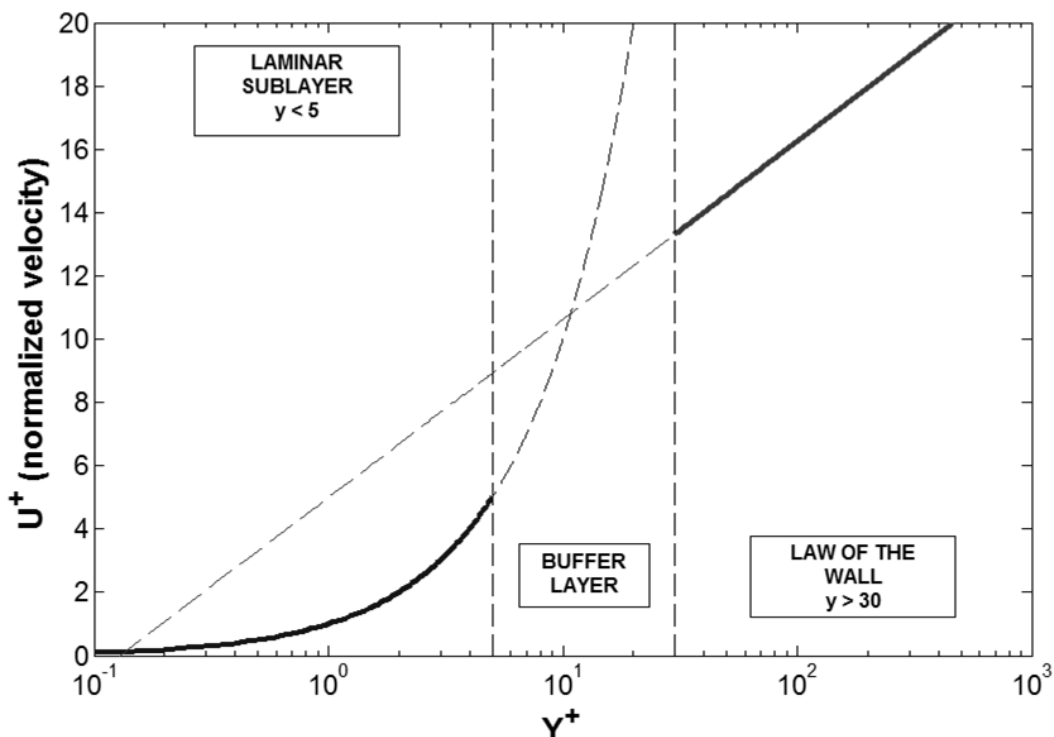
$$\nu_t = \frac{k}{\omega}$$

Effective viscosity, $\nu_{eff} = \nu + \nu_t$

Wall functions

The k - ϵ equations are only valid for fully turbulent high Reynolds number flow, and are thus not valid near the wall. Therefore, k - ϵ equations are not integrated all the way to the walls. We can divide near-wall region in three different categories:

- Viscous sub-layer $0 < y^+ < 5$
- Buffer layer $5 < y^+ < 30$
- Log-law layer $30 < y^+ < 300$



Source: <https://www.computationalfluidynamics.com.au/turbulence-part-3-selection-of-wall-functions-and-y-to-best-capture-the-turbulent-boundary-layer/>

If we keep first cell centre in the viscous sub-layer, all turbulence equations give us accurate results. But that is very difficult to simulate. This approach leads to requirement of abundant mesh number, which means a substantial computational resource is needed.

The viscous sub-layer: ($y^+ < 5$)

In the viscous layer the fluid is dominated by the viscous affect so it can be assumed that the shear stress of fluid is equal to the wall shear stress τ_ω . In viscous layer viscous stress decide the flow and the velocity profile is liner, given by

$$u^+ = y^+$$

Another way of getting accurate simulation results is applying some additional boundary condition at wall which calculates non-linearity of velocity near to the wall. For that we need to add a logarithmic velocity function at walls. This log functions are only valid in log-law region where y^+ value is between 30 & 300.

The log-law layer: ($30 < y^+ < 300$)

In the logarithmic layer turbulence stress dominate the flow and velocity profile varies very slowly with a logarithmic function along the distance y , given by

$$u^+ = \frac{1}{\kappa} \times \ln(Ey^+)$$

Let \overline{U}_p be a velocity component at a node p closest to the wall. We then impose that the velocity at this node should follow the logarithmic wall law according to

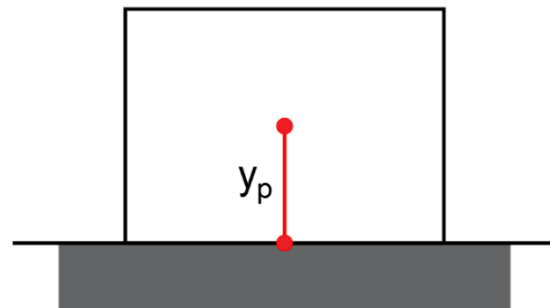
$$\frac{\overline{U}_p}{v^*} = \frac{1}{\kappa} \ln\left(\frac{v^* y_p}{\nu}\right) + B$$

Where, v^* is the wall friction velocity

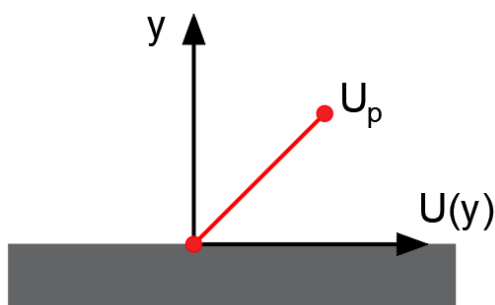
$$v^* = \left(\frac{\tau_\omega}{\rho}\right)^{1/2}$$

$\kappa \approx 0.40$ is the Von Karman constant

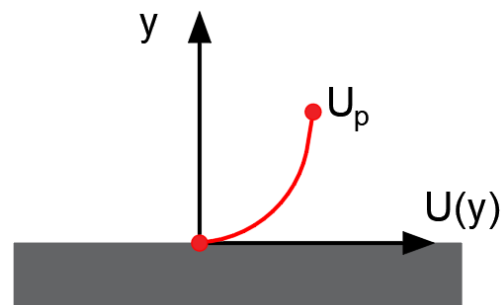
$B \approx 5.0$ is also a constant



Without Wall function



With Wall function



The Buffer layer: ($5 < y^+ < 30$)

Viscous and turbulent stresses are of similar magnitude and since it is complex velocity profile is not well defined and the original wall functions avoid the first cell centre located in this region. However, the improved wall functions allow the first cell centre is located in buffer layer. In OpenFOAM buffer layer is divided 2 parties. One is using the liner relation as that in viscous sublayer, the other is to use logarithmic function as that in logarithmic region. **Cells in this layer will not get accurate results. So always avoid to keep cells in this region.**

The wall functions used in spoken tutorial are:

1. Wall function for k
 - 1.1. kqRWallFunction
 - 1.2. kLowReWallFunction
2. Wall function for epsilon
 - 2.1. epsilonWallFunction
 - 2.2. epsilonLowReWallFunction
3. Wall function for omega
 - 3.1. OmegaWallFunction
4. Wall function for nut
 - 4.1. nutkWallFunction
 - 4.2. nutkRoughWallFunction
 - 4.3. nutkAtmRoughWallFunction
 - 4.4. nutkFilmWallFunction
 - 4.5. nutLowReWallFunction
 - 4.6. nutUWallFunction
 - 4.7. nutURoughWallFunction
 - 4.8. nutUSpaldingWallFunction
 - 4.9. nutUTabulatedWallFunction

To study wall functions in detail, please go through the link [here](#).