An estimate of the Reynolds number (Re) is required to determine if the flow is turbulent:

$$Re = \frac{U * L}{\nu} = \frac{(44.2)(0.101)}{1.56e-05} \approx \mathbf{286166}$$

The Reynolds number yields so high that the flow is undoubtedly turbulent.

In internal flows, such as flows through pipes or channels, the choice of the reference length scale in turbulence models like the **k–ε** model is crucial for accurate and meaningful simulations. The reference length scale should be related to the characteristic size or geometry of the internal flow. In the case of the tutorial case: backwardFacingStep2D which resembles a rectangular or non-circular duct, the height or width of the duct can be used. Hence, the *inlet height* is taken as the *reference length scale (characteristic length)* $L = 0.101\ m$.

The turbulent kinetic energy was calculated with a medium turbulence intensity of $5\ \%$. Hence based on these two previous assumptions, the turbulent kinetic energy was estimated as follows:

$$k = \frac{3}{2}\left(I\left|u_{ref}\right|\right)^2 = 7.32615\ \frac{m^2}{s^2}$$

In which, I is the turbulence intensity and u_{ref} a reference flow speed[1] set as 44.2 m/s.

The ϵ (turbulence dissipation rate) for the model $k-\epsilon$ was obtained according to the user-guide[2] as follows:

$$\epsilon = \frac{C_\mu^{0.75} k^{1.5}}{l_m} \approx 322.6\ \frac{m^2}{s^3}$$

In which, $C_\mu = 0.09$ and $l_m = 10\% \times characteristic\ lenght\ (inlet\ height)$. Thus, $l_m = 0.0101\ m$.

The ω (turbulence dissipation rate) for the model $k-\omega$ was obtained as follows:

$$\omega = \frac{k^{0.5}}{C_\mu^{0.25} l_m} \approx 489.27\ \frac{1}{s}$$

Regarding the turbulence model: **realisable k-ε**, This is a two-equation model. Suitable for complex shear flows involving rapid strain, moderate swirl, vortices, and locally transitional flows (e.g., boundary layer separation, massive separation, and vortex shedding behind bluff bodies, stall in wide-angle diffusers, room ventilation). It overcome the limitations of the standard k-epsilon model, widely known for its poor performance of poorly complex flows involving severe pressure gradient, separation, strong streamline curvature. Furthermore, the **standard k-ε** can be only used with wall functions.

In order to simulate the boundary layer, a plot (see figure 1) of a non-dimensional velocity ($U+$) versus distance to the wall of the first cell ($\mathbf{Y}+$) is helpful to verify whether the turbulence model

[1] Turbulence length scale --CFD-Wiki, the free CFD reference (cfd-online.com)
[2] https://doc.cfd.direct/openfoam/user-guide-v11/backwardstep

used is correct. Two processes are known that is, the *wall resolving* process (or wall resolution) that is strongly suggested, otherwise *wall modelling* process can be used by using wall functions. In other words, when meshing, inflation layers are needed with a recommended distance to the wall of the first cell close to the wall $(\mathbf{Y+} < \mathbf{5})$. The recommended distance varies depending on the author, but it should yield within the viscous sublayer (called low Re) to simulate the full boundary layer. However, the distance to the wall of the first cell can be computed within the log-law region $(\mathbf{Y+} > \mathbf{30})$ to model the boundary layer partially with help of empirical correlations about the viscous sublayers defined as the *wall functions* to save meshing effort and computational time (called high Re).

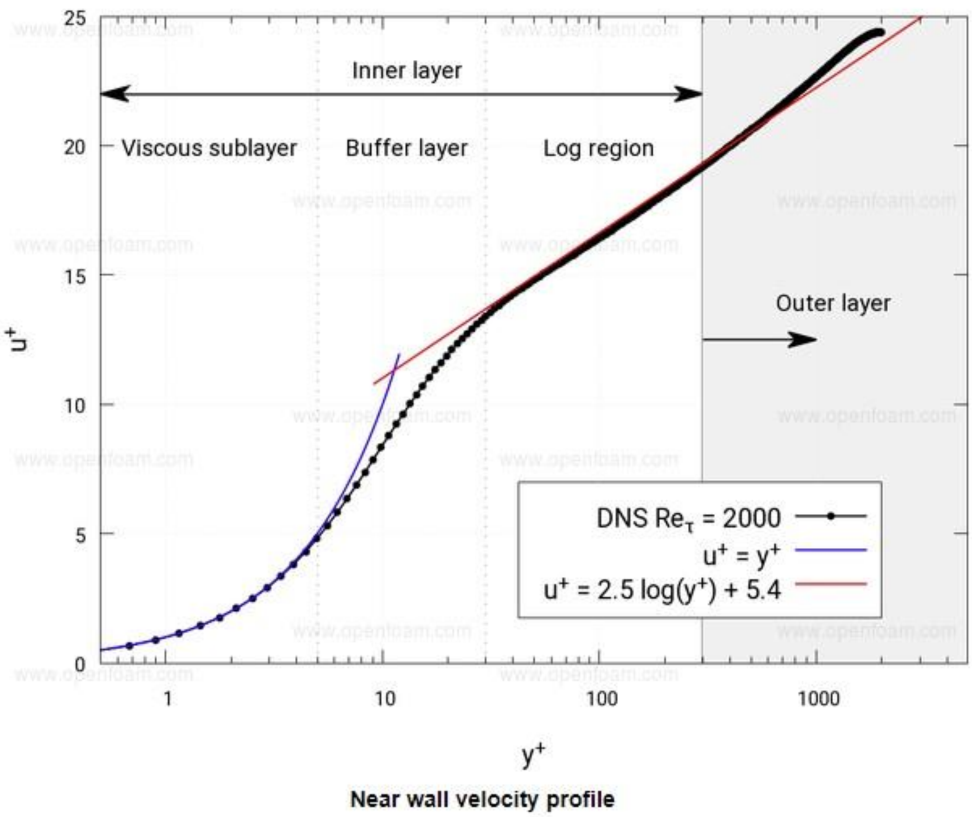

Figure 1: (OpenFOAM , 2017)

Another approach is by means of computing $y+$ as a postprocesing funcion in OpenFOAM in order to visualize qualitatively the contours.

For the model $k-\epsilon$:

The upper Wall yielded: min = 0.614175, max = 2.71758, average = **1.63254**
The lower Wall yielded: min = 0.000727426, max = 8.68193, average = **1.7255**

For the model $k-\omega$:

The upper Wall yielded: min = 0.500908, max = 2.6904, average = **1.39037**
The lower Wall yielded: min = 0.0279966, max = 8.76023, average = **1.68962**

For the model realizableKE:

The upper Wall yielded: min = 0.519134, max = 2.72148, average = **1.41757**
The lower Wall yielded: min = 0.0122769, max = 5.67568, average = **1.32923**

These results show good refinement level for proper wall resolving level. In all cases analysed, there is no need for wall functions since $y+$ is within the viscous sub-layer. For the purpose of this activity, it is worth focusing on the step of the domain, where a recirculation zone develops due to the abrupt change of plateau. Two criteria to observe are the recirculation zones and the reattachment point.

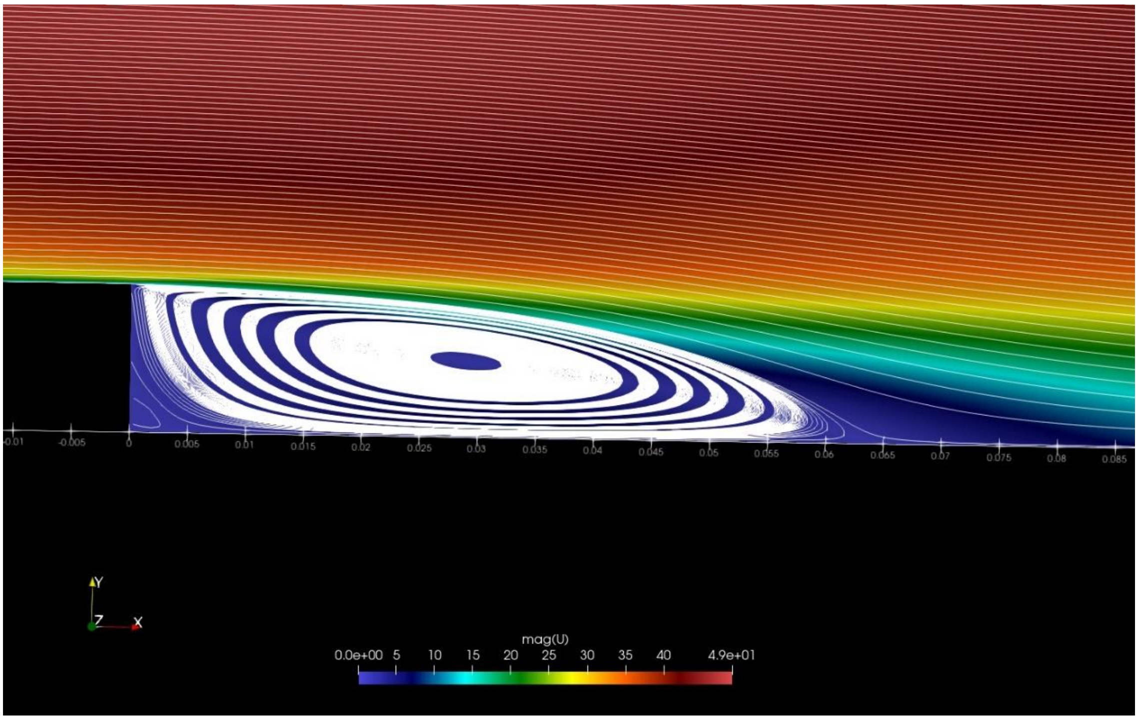

Figure 2: model $k-\epsilon$

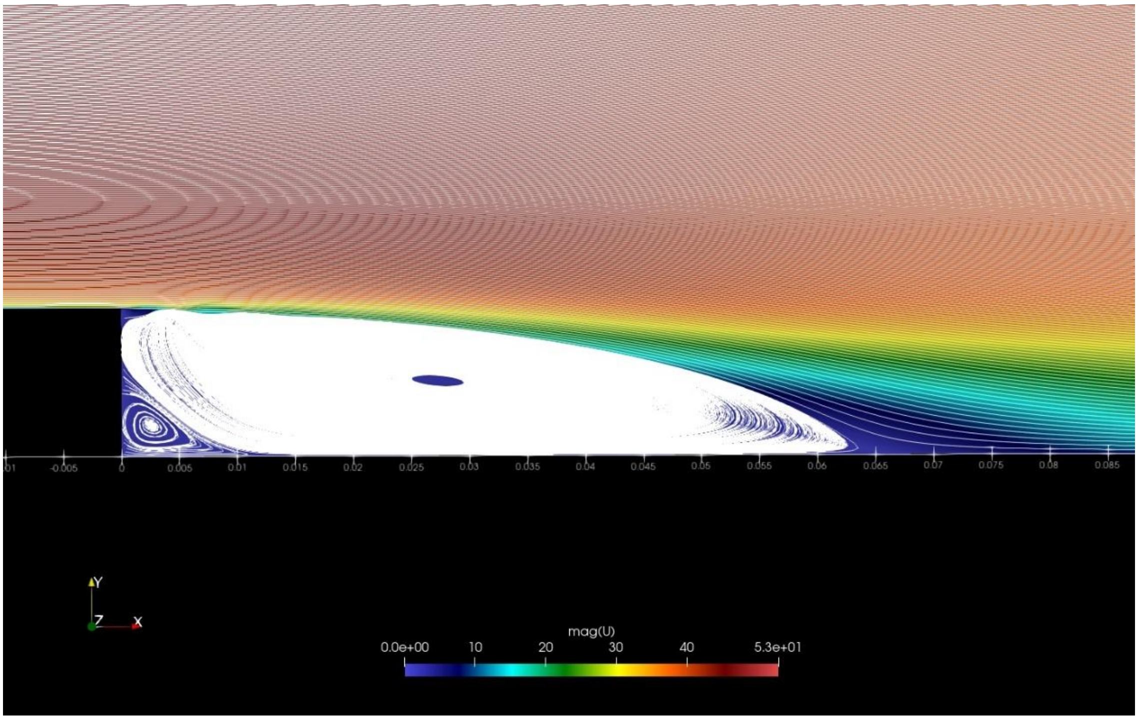

Figure 3: model $k - \omega$

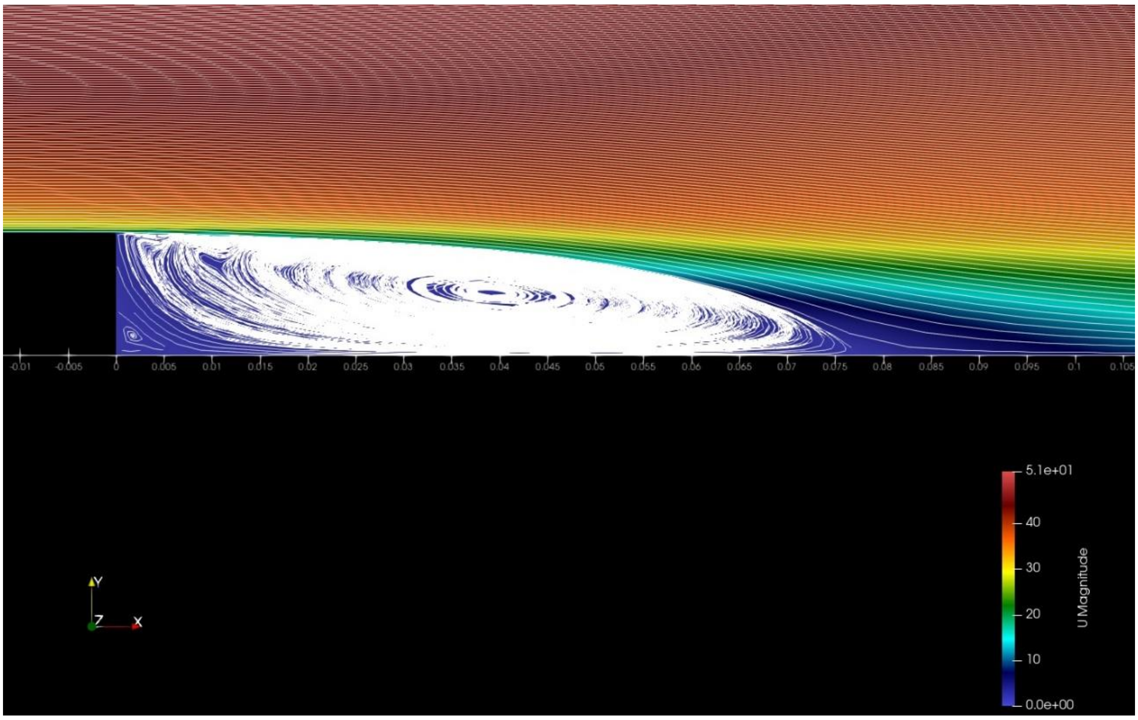

Figure 4: model realizableKE

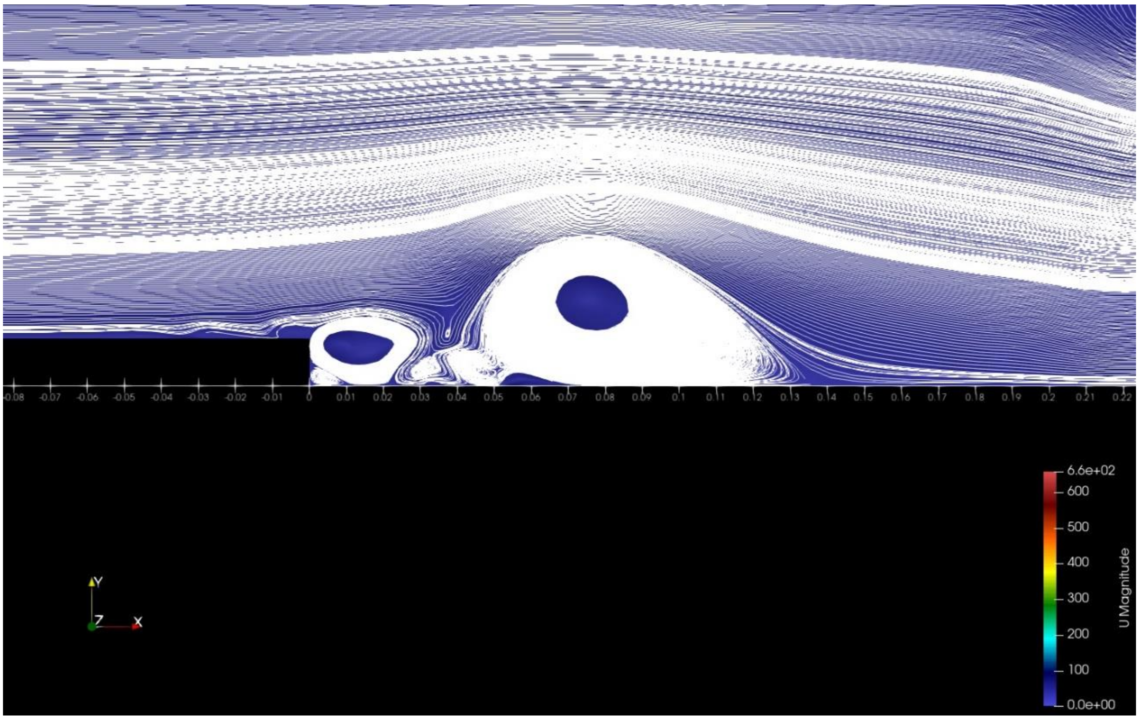

Figure 5: model spalartAllmaras

Comparing the results of the turbulence model spalartAllmaras (figure 5) with the rest of the turbulence models, it can be noticed that the spalartAllmaras model yields other results in the recirculation zone. The Spalart-Allmaras turbulence model does not directly solve for the turbulent kinetic energy (TKE) like the two-equation turbulence models such as the k-epsilon or k-omega models. Instead, it relies on solving a single transport equation for the eddy viscosity (or nuTilda) to capture the effects of turbulence.

In the Spalart-Allmaras model, the eddy viscosity is used to approximate the turbulent viscosity, which is then used to modify the molecular viscosity in the Reynolds-averaged Navier-Stokes (RANS) equations. This modification allows the model to simulate the turbulent flow field without explicitly solving for the turbulent kinetic energy. However, it's worth noting that the Spalart-Allmaras model does indirectly affect the turbulent kinetic energy through its influence on the turbulent viscosity and turbulent dissipation rate. The model aims to capture the essential features of turbulent flows, including turbulent mixing and transport, **without the need for explicitly resolving additional turbulence quantities such as TKE**.

So, while the Spalart-Allmaras model does not solve for TKE explicitly, it is still able to simulate a wide range of turbulent flows effectively, particularly in aerodynamic applications where it has been widely used. Hence, the spallart Allmaras model is not suited for internal flows, as shown in figure 5.