

Hands-on session 6 – Low Reynolds k- ϵ Models Application to Fully Developed Turbulent Pipes and Fully Developed Square Duct

Abstract


In this session the student will become familiar with the low Reynolds k- ϵ models in particular with the modeling of Lam and Bremhorst (1981). It will be shown how the low Reynolds model can provide a good prediction of the turbulent quantities close to the wall in the fully developed turbulent pipe flows. The employment of the same model will be extended to the case of the squared duct to confirm whether a different wall treatment might be responsible for the prediction of the secondary flows.

Goal

The goal of this session is to consolidate the student's ability to generate a mesh with the appropriate y^+ depending on the wall treatment that they intend to employ and confirm the results against DNS data for the circular pipe and for the squared duct.

Author	CFDLab@Energy		Page 1 of 15
CFD for Nuclear Engineering	Session 2		


Hands-on session 6 – Low Reynolds k-ϵ Models Application to Fully Developed Turbulent Pipes and Fully Developed Square Duct		1
1	Introduction	3
2	Application to the Fully Developed Turbulent Pipe Case	4
2.1	Creation of the appropriate grid	4
2.2	Selection of the model in Fluent	5
2.3	Results	6
2.3.1	Achieve solution in case of poor convergence	6
2.3.2	Plot y^+ on the wall.....	8
2.3.3	Achieve solution in case of poor convergence	9
2.3.4	Compare the turbulence variance.....	11
3	Application to the Fully Developed Turbulent Square Duct Case	12
3.1	Creation of the appropriate grid	13
3.2	Selection of the model in Fluent	14
3.3	Results.....	14
4	Conclusions	15

Author	CFDLab@Energy		Page 2 of 15
CFD for Nuclear Engineering	Session 2		

1 Introduction

In the previous sessions we have seen how the Standard Wall Functions work well together with the Standard k- ϵ model and the Standard k- ω model to predict not only the velocity profile but also the stream-wise velocity variance. However, the prediction of the velocity and variance was limited to the log-law and the main turbulent region. We want to confirm whether the low Reynolds models are able to predict the quantities close to the walls. In addition we will see whether the employment of low Reynolds models brings any benefit in the prediction of the secondary flows in the square duct.

Note: It is advised to prepare an excel file to summarize the fluid properties and boundary conditions for the resolved variables (pressure, x- and y-velocity, temperature). Some of these data may require calculation depending on the information given in the case description.

Author	CFDLab@Energy		Page 3 of 15
CFD for Nuclear Engineering	Session 2		

2 Application to the Fully Developed Turbulent Pipe Case

As a first case let's apply the low Reynolds modeling to an existing working case such as the Fully Developed Turbulent Pipe case.

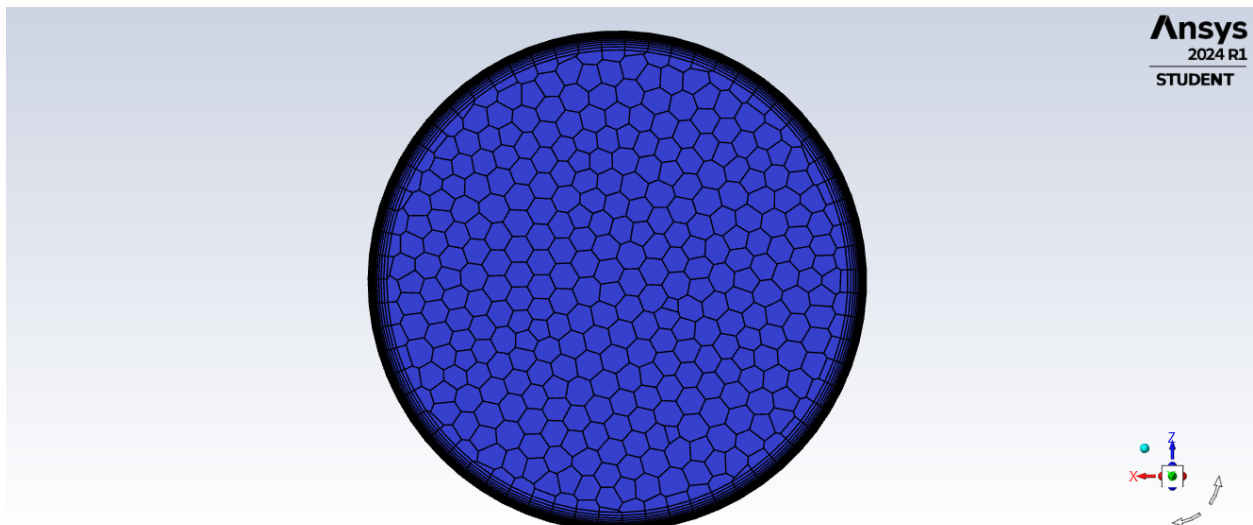
2.1 Creation of the appropriate grid


In order to employ the low Reynolds models, we need to resolve the boundary layer up to the viscous sub-layer. It is custom practice to generate the cells close to the wall so that:

1. The center cell of the first cell close to the wall results in a y^+ smaller than 1
2. The boundary layer is discretized with at least 20 layers

Reopen the file .msh in Fluent Meshing and modify the boundary layer according to the points above. The process is iterative so that once the mesh is created the condition of $y^+ < 1$ should be confirmed at the end of the solution, but you can have a first estimation of how large the first cell should be with the following approach:

- Evaluate the *wall shear stresses* $\tau_w = \frac{1}{2} C_f \rho U^2$ where the Fanning factor C_f is four times smaller than the Moody's friction factor
- Evaluate the *friction velocity* $u_\tau = \sqrt{\frac{\tau_w}{\rho}}$
- Evaluate the y from the definition of the y^+ $y^+ = \frac{\rho u_\tau y}{\mu}$



Author	CFDLab@Energy		Page 4 of 15
CFD for Nuclear Engineering	Session 2		

Save the mesh as “caseName_lowRe”.

2.2 Selection of the model in Fluent

Open the Standard k-ε case and Save as “caseName_lowRe_LamBremhorst” or any name of your choice and import the mesh only.

Despite the GUI in Fluent, the selection of the low Re k-ε has been hidden in the expert commands. To access it, it is necessary to set it manually on the Console. Click on the consol and press enter. You will see the following commands:

```
/define/models/viscous> q
/define/models> q
/define> q
>
adjoint/          parallel/          server/
define/           parametric-study/ solve/
display/          plot/           surface/
exit              preferences/    turbo-workflow/
file/             print-license-usage views/
mesh/             report/
> |
```

To define the low Re models type `define/models/viscous/turbulence-expert`


If you made a mistake and need to move back in the folder simply type “q”.

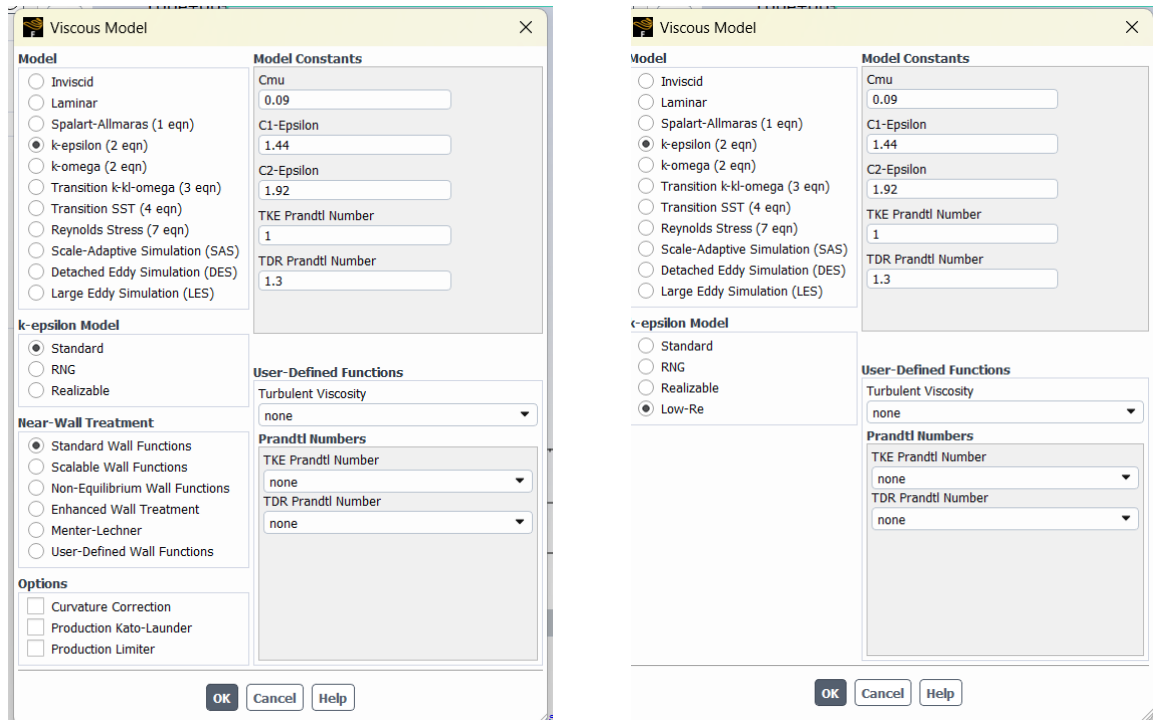
Once the turbulence-expert press enter, the following will appear

```
/define/models/viscous/turbulence-expert>
kato-launders-model?      production-limiter?
low-re-ke?                turb-non-newtonian?

/define/models/viscous/turbulence-expert>
```

Here we can define whether we want to set the low-Reynolds k-ε model. Type `low-re-ke` and press enter. Type Yes and press enter. Now the model is set. If you click on the Viscous model in the Case View on the left you will see that the k-epsilon model has now an additional slot named Low-Re. The figure below shows the difference between the Viscous model before and after introducing the low-Re models.

Author	CFDLab@Energy		Page 5 of 15
CFD for Nuclear Engineering	Session 2		



Fluent has a number of models already implemented for the definition of the damping functions f_{μ} , $C_{\epsilon 1}$ and $C_{\epsilon 2}$, also in this case we need to use the console for setting them. Type in the Console “low-re-ke-index” and type the value of the model that you intend to use. For this case let’s type 1 and use the well-known Lam and Bremhorst model.


Low-Re models implemented in fluent and their index to be chosen in the console.

- [0] Abid
- [1] Lam-Bremhorst
- [2] Launder-Sharma
- [3] Yang-Shih
- [4] Abe-Kondoh-Nagano
- [5] Chang-Hsieh-Chen

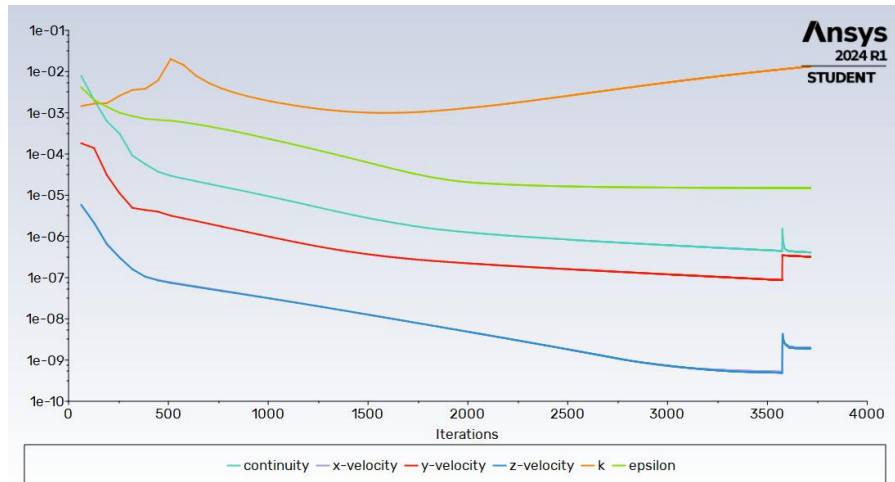
Let’s run the simulation!

2.3 Results

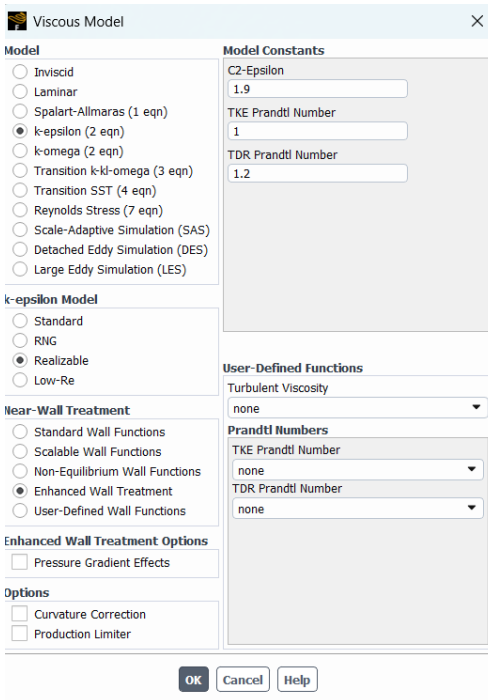
2.3.1 Achieve solution in case of poor convergence

Author	CFDLab@Energy		Page 6 of 15
CFD for Nuclear Engineering	Session 2		

The simulation with sophisticated models such as the low Re k- ϵ might not lead to immediate convergence despite the very simple geometry and even if the first cell was created properly to maintain a y^+ lower than 1. The residuals might look like the plot below where in particular the k -value keeps increasing.



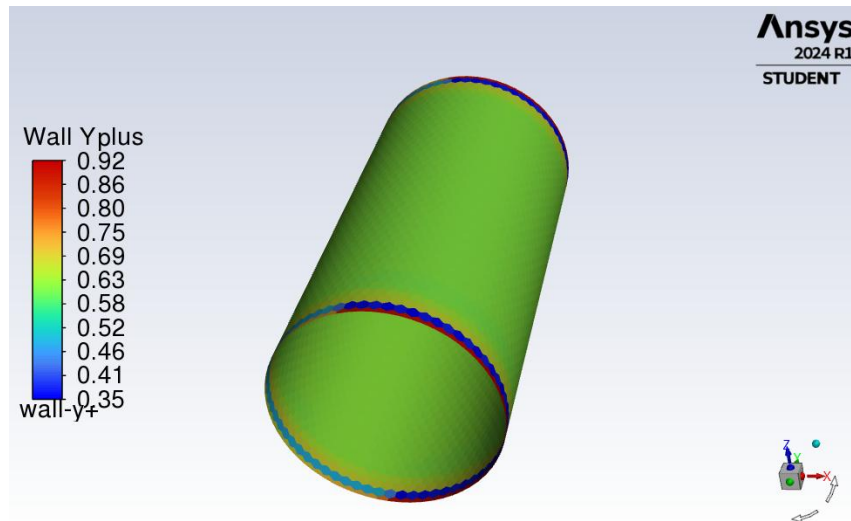
One solution is to run first the simulation with a “more stable” model such as the k- ϵ Realizable or even the k- ϵ Standard first. As the mesh is very fine at the wall it would not be suitable to select a Standard Wall function model at the wall, hence select “Enhanced Wall Treatment”. The selection of the viscous model should look like the picture below.



Once the solution has reached a good level of convergence, switch (without initializing!) to the low Re k-ε.

2.3.2 Plot y^+ on the wall

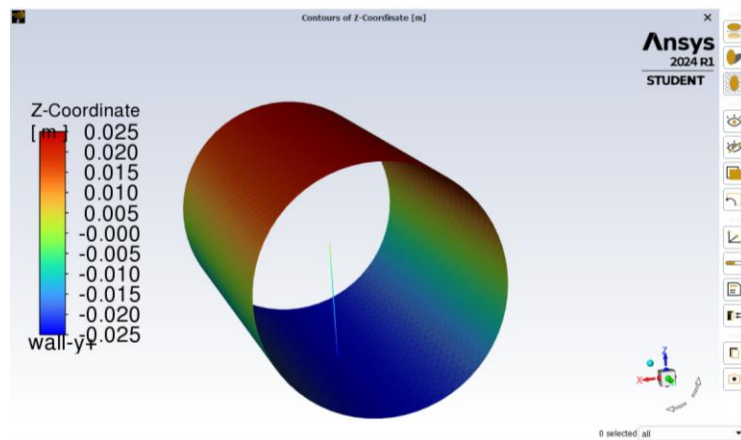
To consider the results first let's confirm that the y^+ on the wall is lower than 1 as expected.




2.3.3 Achieve solution in case of poor convergence

If we want to export the data already for direct comparison with the experimental one we need to export the normalized variance on y^+ . This is possible in Fluent by creating another Custom Function for y^+ and plotting over this function. Please note that using the evaluated y^+ in Fluent will not work because the y^+ value is evaluated only at the wall, while it is zero everywhere else in the domain.

First let's create a line which extends from one wall to the center of the domain. In our case the line is aligned with the z axis. As shown below.



We need to notice that the center of the domain is 0, while the location of the wall along this line is, in our case, -0.025 [m]. However, the definition of y^+ is the following:

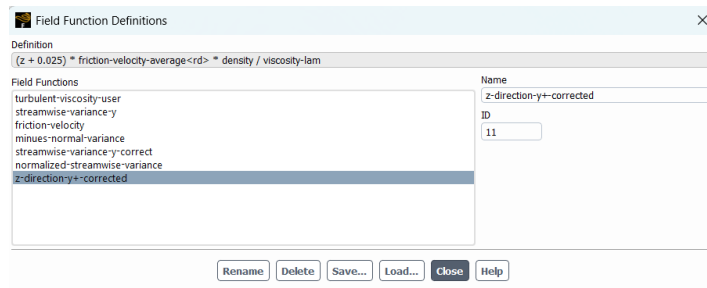
Author	CFDLab@Energy		Page 9 of 15
CFD for Nuclear Engineering	Session 2		

$$y^+ = \frac{\rho u_\tau y}{\mu}$$

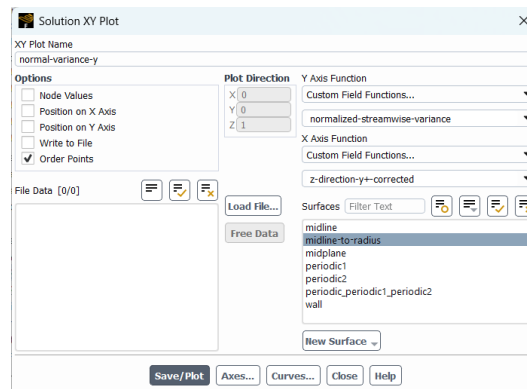
where u_τ is the friction velocity and y is the distance from the wall. Hence, either we translate our domain so that the wall is located in 0 or the definition of y^+ should be written in the custom functions as:

$$y^+ = \frac{\rho u_\tau (y - R)}{\mu}$$

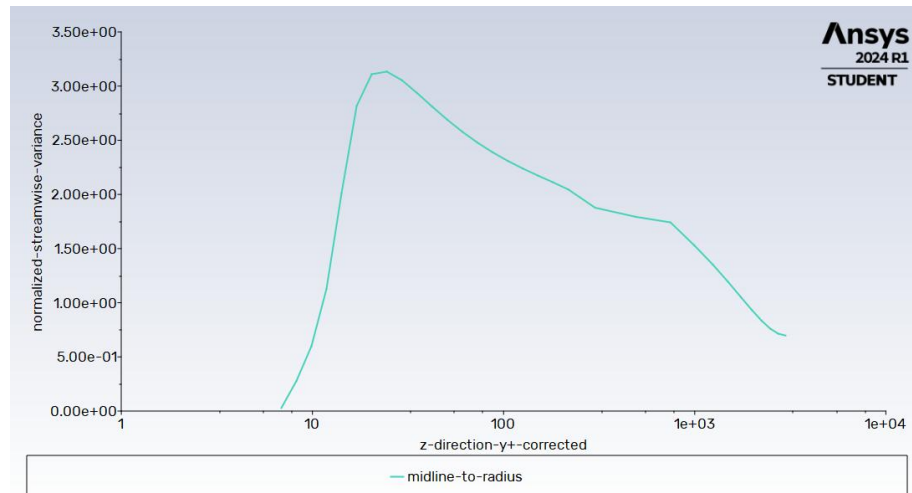
Where R is the radius of the pipe. The definition of the custom function for y^+ would look like below:



For plotting a variable against another variable (not a location either in the x, y, or z direction) open a XY-plot and deslect “ Position on X Axis” on the left side. On the right side you will be able to select now a Function for the X-axis. Select Custom Functions and select the function you have created for the y^+ as below:

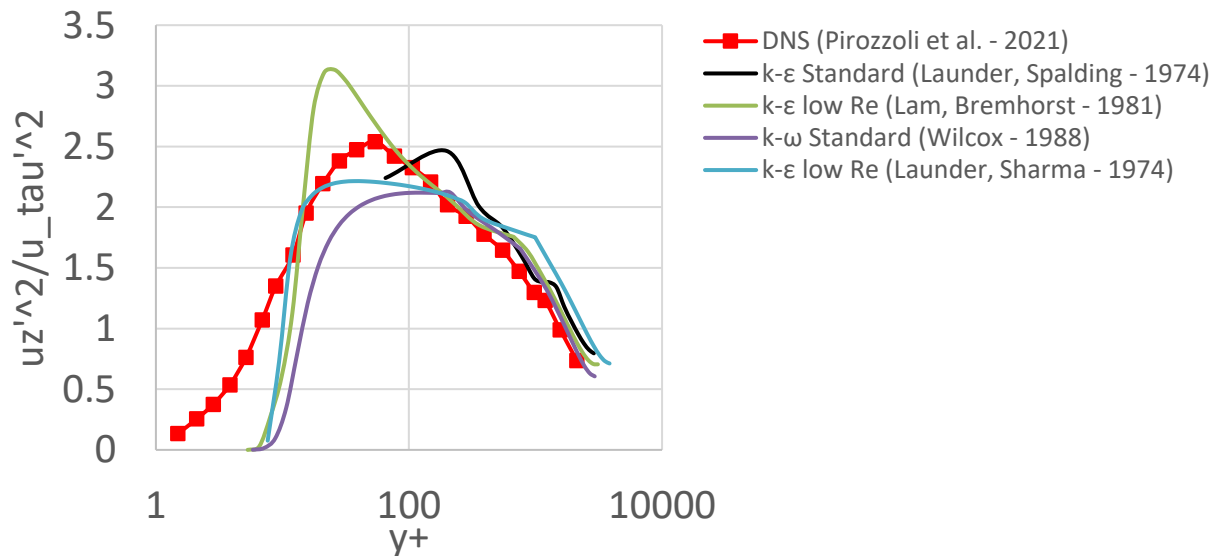



The plot should look like the picture below:



2.3.4 Compare the turbulence variance

Let's compare again the streamwise variance as done previously and compare with the DNS data of Pirozzoli et al. 2021 as done in the picture below. The picture below reports more than one model for comparison.

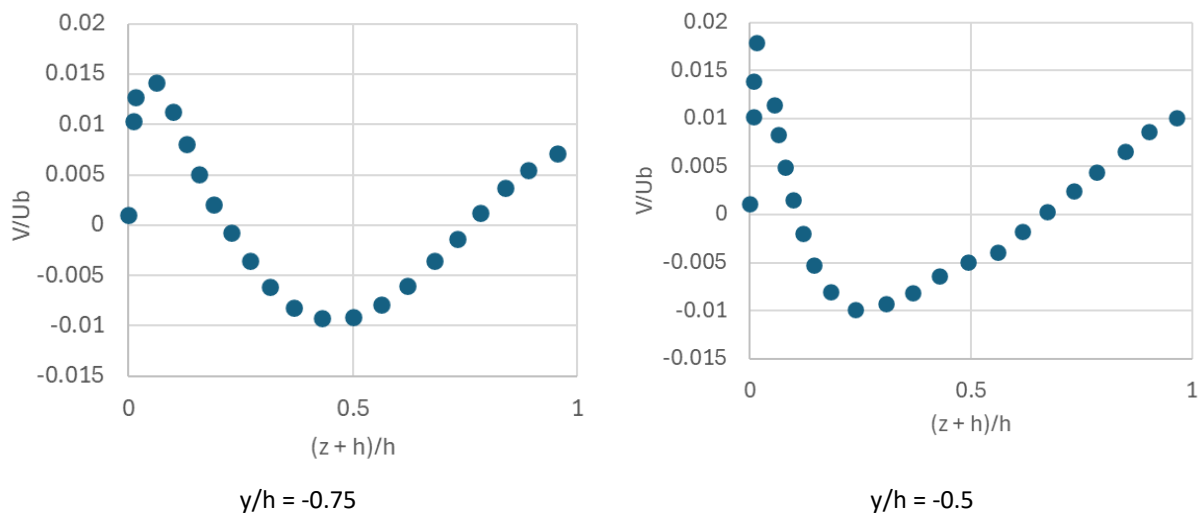


Author	CFDLab@Energy		Page 11 of 15
CFD for Nuclear Engineering	Session 2		


We can notice that the low Reynolds models resolve the boundary layers in comparison with the k- ϵ Standard which neglects the region below $y^+ < 30$. The results show a good agreement in the main turbulent region $y^+ > 500$ and a similar trend in the log law layer and in the viscous sublayer. There isn't a model which appears completely superior than others.

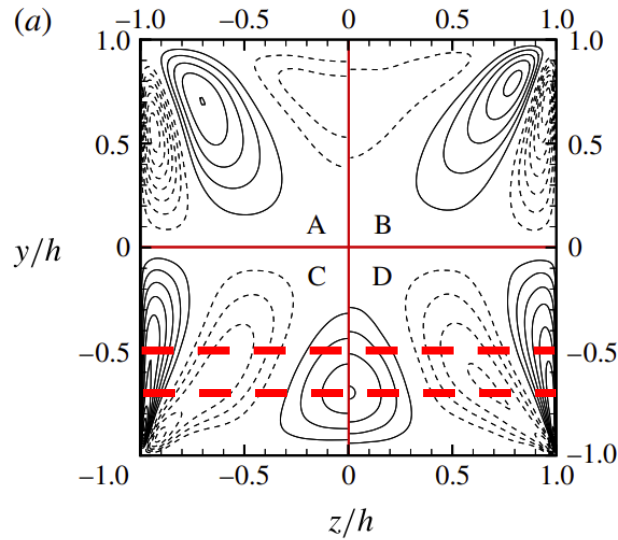
3 Application to the Fully Developed Turbulent Square Duct Case

Before we start the simulation let's remember that the velocity profile in the Hands-on session 5 for the squared duct did not resemble qualitatively the DNS data of Pirozzoli et al. 2018. In particular secondary flows were not visible. Secondary flows create cross-stream velocities which is shown in the plots below. The comparison with the Standard k- ϵ and Standard k- ω is not reported as the value is zero.



For understanding the location where the data are provided please refer to the picture below.

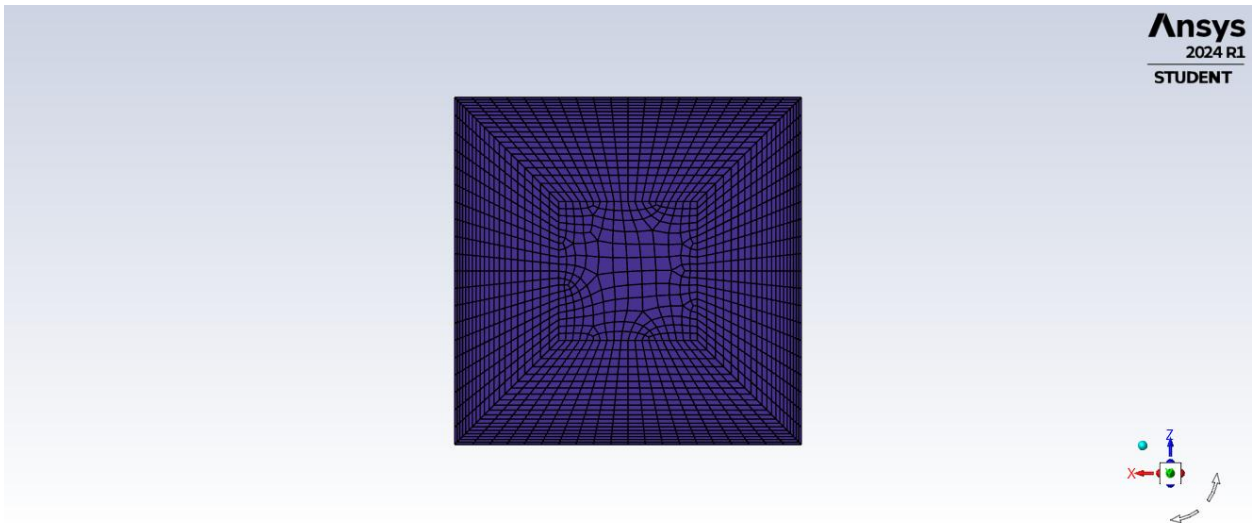
Author	CFDLab@Energy		Page 12 of 15
CFD for Nuclear Engineering	Session 2		




The result was that both Standard k-ε and Standard k-ω fail to predict the secondary flows. We want to confirm whether the employment of Standard Wall Functions was responsible for the misprediction of the secondary flows.

3.1 Creation of the appropriate grid

As in chapter 2.1 above, generate the appropriate grid in order to have $y^+ < 1$ on the cells on the walls.



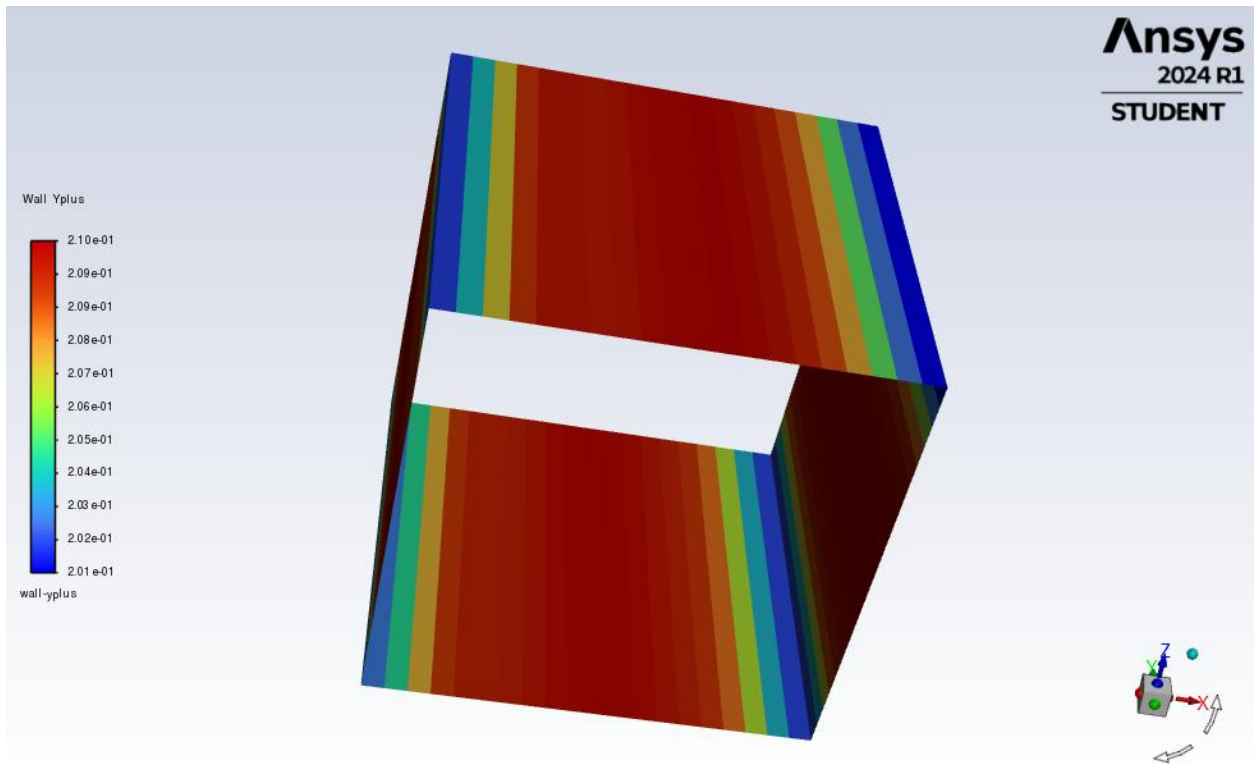
Author	CFDLab@Energy		Page 13 of 15
CFD for Nuclear Engineering	Session 2		

3.2 Selection of the model in Fluent

As in chapter 2.2 above select the low Re k-ε with the model of Lam and Bremhorst and run the simulation.


3.3 Results

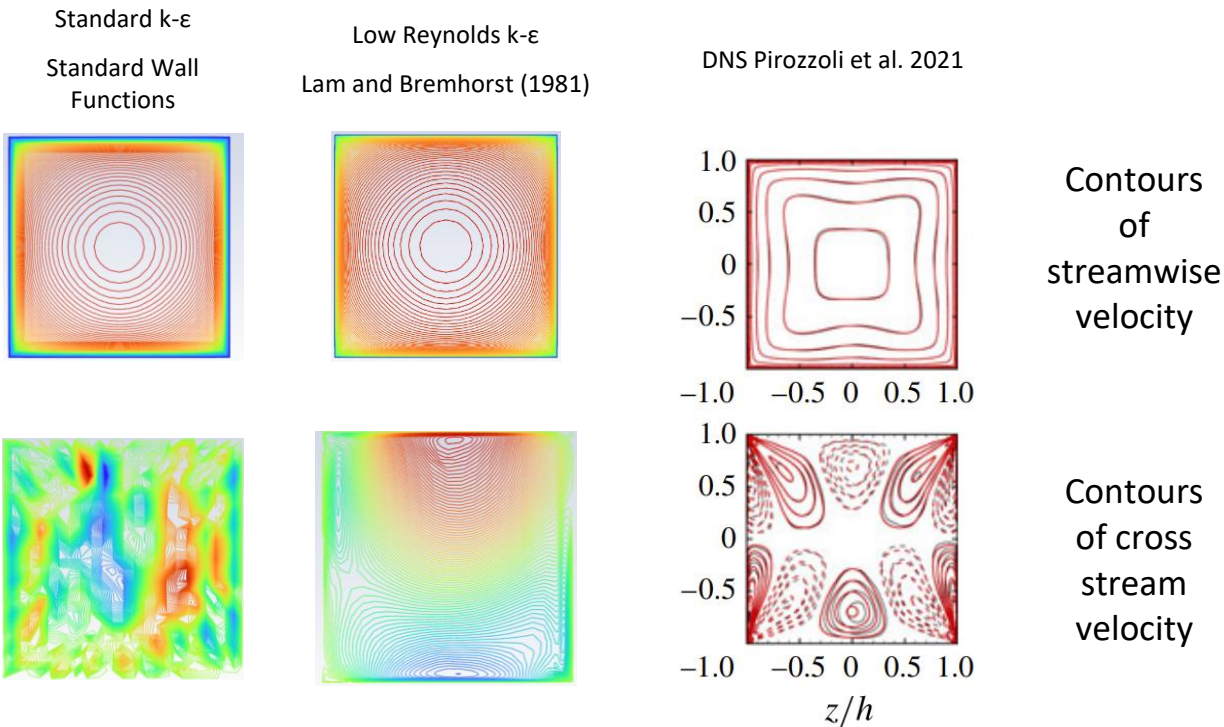
Lets's confirm first the y^+ values on the walls for the generated mesh



In this case the value is too conservative which is obviously acceptable but a larger cell on the wall would result in a faster computation.

The qualitative analysis of the flow in the picture below shows that the resolution of the subviscous layer creates a more realistic velocity field which is affected by the presence of the corners. Secondary flows are still not visible as their magnitude is very small and their position is not regular in the domain. For the prediction of secondary flows it will be necessary to add another element to the computation.

Author	CFDLab@Energy		Page 14 of 15
CFD for Nuclear Engineering	Session 2		



4 Conclusions

The formulation of low Reynolds k-ε modeling is capable to predict correctly the sub-viscous layer region this can be used in complex geometries where the standard wall functions fail. Even in the squared duct, resolving the flow down to the sub-viscous layer introduces the effect of the corners on the flow. However, for the correct computation of the secondary flows anisotropic effects need to be added to the model.