

Hands-on session 5 – Turbulent Fully Developed Flow in a Squared Duct

Abstract

This session will introduce you to the simulation of the **fully developed turbulent flow in a squared duct**. The student will start with the generation of the geometry (refer to Hands-on Session 3) with the most appropriate grid. For this exercise the Reynolds number is assumed to be equal to 40,000 with air which properties are the following: density $1.225 \text{ [kg/m}^3\text{]}$ and dynamic viscosity $1.789 \times 10^{-5} \text{ [kg/m-s]}$. These are the default values in Fluent. The exercise will be done with two different turbulence models, that is to say k- ϵ Standard and k- ω Standard.


Goal

The aim of this hands-on session is to strengthen the knowledge about geometry and mesh generation and confirm the inability of the Standard formulation of the most popular turbulence models to compute anisotropy of the turbulence and secondary flows.

Author	CFDLab@Energy		Page 1 of 13
CFD for Nuclear Engineering	Session 2		

Hands-on session 5 – Turbulent Fully Developed Flow in a Squared Duct.. 1


1	Introduction	3
2	Geometry Generation in ANSYS SpaceClaim.....	4
2.1	Generate the Geometry in SpaceClaim®	4
2.2	Define the Multizone Meshing	5
3	Define the Mass Flow Rate	7
3.1	Simulation with the k- ϵ Standard and Main Results	7
3.2	Simulation with the k- ω Standard	10
4	Conclusions	12

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

1 Introduction

After the great performance of Standard $k-\epsilon$ and Standard $k-\omega$ in the fully developed turbulent pipe flow we extend the study to the squared duct. The first challenge will be about grid generation. This step is guided. The rest of the steps, to judge convergence and plot the main quantities, is just hinted. The student should refer to past Hands-on Sessions.

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 13
CFD for Nuclear Engineering	Session 2		

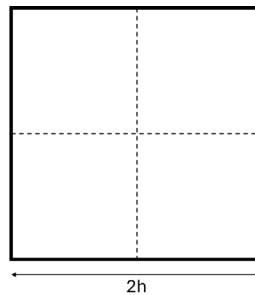
2 Geometry Generation in ANSYS SpaceClaim

2.1 Generate the Geometry in SpaceClaim®

We are referring to the paper by Pirozzoli et al. “Turbulence and secondary motions in square duct flow”, J. Fluid Mech. (2018), vol. 840, pp. 631–655. The results of the paper are given in a non-dimensional fashion so the duct can have any geometry. Let’s compute the case with:

$$Re_b = \frac{2h \cdot U_b}{\nu} = 40,000$$


Where $2h$ represents the duct side length as in the picture.

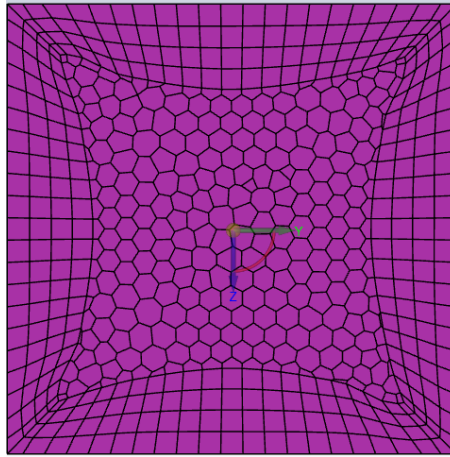


You will need to define the mass flow rate, later in the computation, based on the duct size length that you have defined.

To generate the geometry try to start following the steps provided in the Hands-on Session 3 choosing the parameters to obtain the appropriate mesh. Please consider, based on the previous discussions, whether it is suitable to have a Tet, Hexcore, Polyhexcore mesh.

If you do everything correctly you might reach the following result.


Author	CFDLab@Energy		Page 4 of 13
CFD for Nuclear Engineering	Session 2		

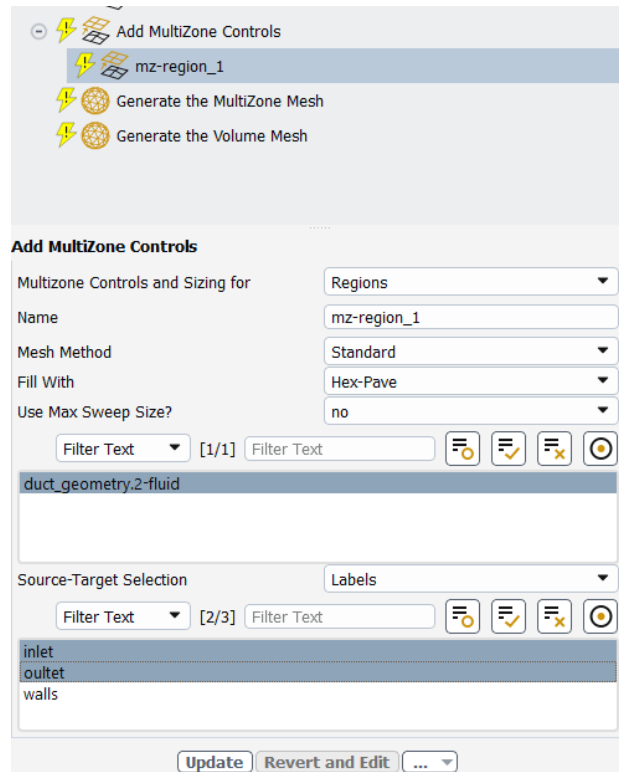


This figure shows the boundary layer on each side of the duct as we want. However, the size of the boundary layer is distorted close to the corner which is not ideal for the computation. To remedy to this issue Fluent provides the so called “Multizone Meshing” which allows to create a sweep-able mesh from the inlet to the outlet. To do so please follow the following steps.

2.2 Define the Multizone Meshing


If you reached the mesh above you can click on “Describe Geometry” and click “Revert and Edit” at the bottom. Once all the steps are reverted click Yes in “Enable Multizone Meshing”. In the tree select “Add MultiZone Controls”. Leave all the default options and select the region “duct_geometry.2-fluid” as in the picture below and select Inlet and Outlet:

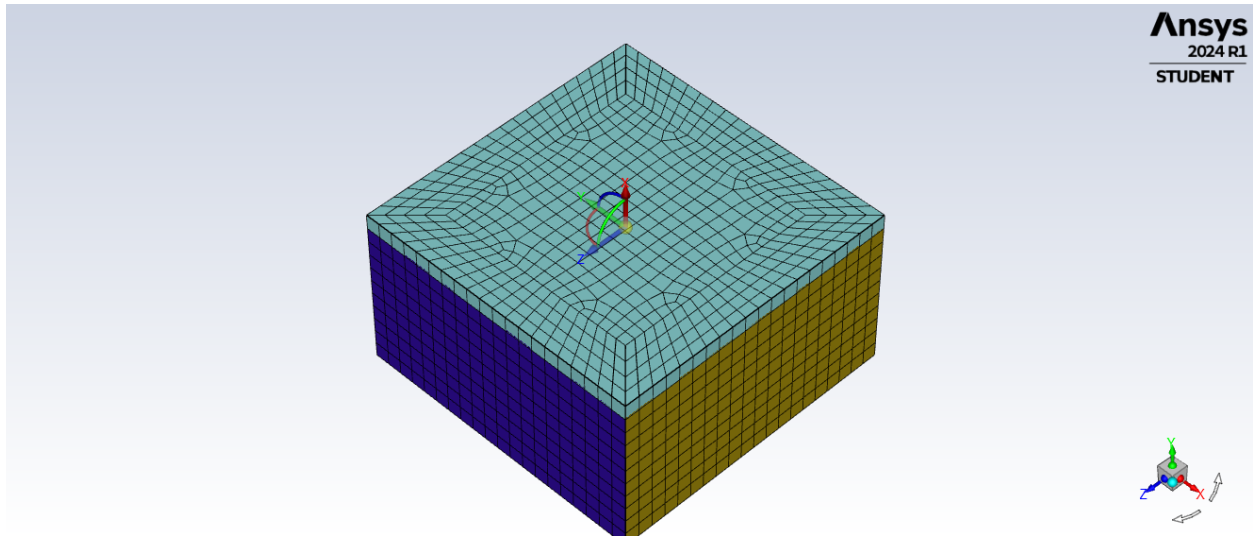
Author	CFDLab@Energy		Page 5 of 13
CFD for Nuclear Engineering	Session 2		



Click Generate the Multizone Mesh and select the name of the geometry duct_geometry.2-fluid. Click Update.

The grid should look like the one below where the boundary layer is uniform across all the edges.

Author	CFDLab@Energy		Page 6 of 13
CFD for Nuclear Engineering	Session 2		



Note: the Periodic faces, as in the picture below, have quadrilateral faces, while with the Volume mesh explored in Hands-on session 3, we were not able to obtain Quads on the faces. The student interested might want to test the MultiZone meshing for the turbulent pipe flow to consider whether it provides a better convergence in the fully developed turbulent pipe flow case.

Save the grid and switch to Fluent Solution.


3 Define the Mass Flow Rate

As stated above we have chosen to simulate the case with $Re = 40,000$. The value of the mass flow rate to be input in your simulation will depend on the size of your domain that you have chosen. Calculate the appropriate mass flow rate according to this formula:

$$Re_b = \frac{2h \cdot U_b}{\nu} = 40,000$$

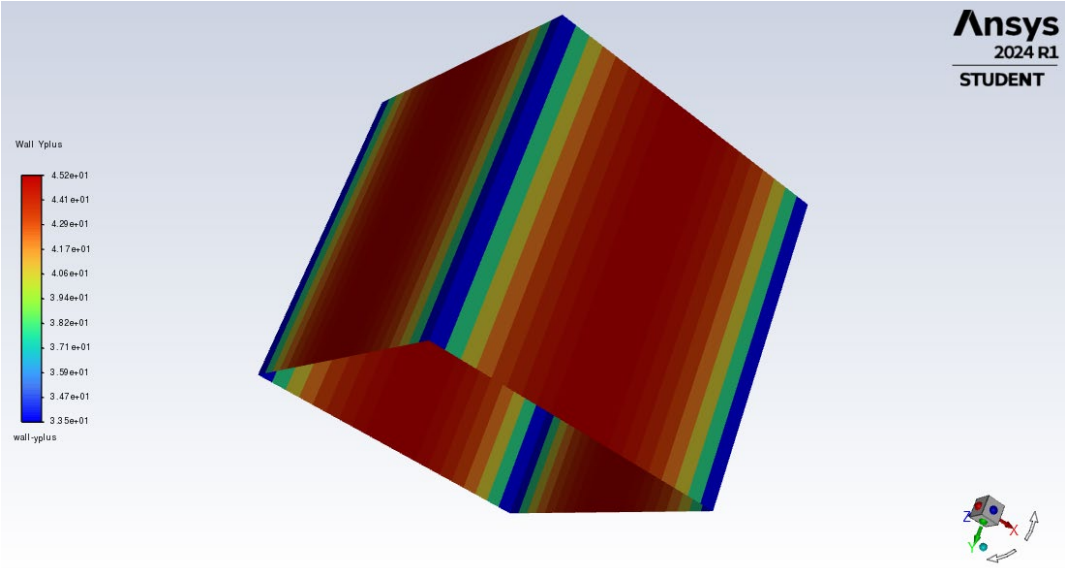
3.1 Simulation with the k-ε Standard and Main Results


- Define the viscous term as Standard k-ε.
- Judge the convergence based on: residuals, reports of meaningful physical quantities.

Author	CFDLab@Energy		Page 7 of 13
CFD for Nuclear Engineering	Session 2		

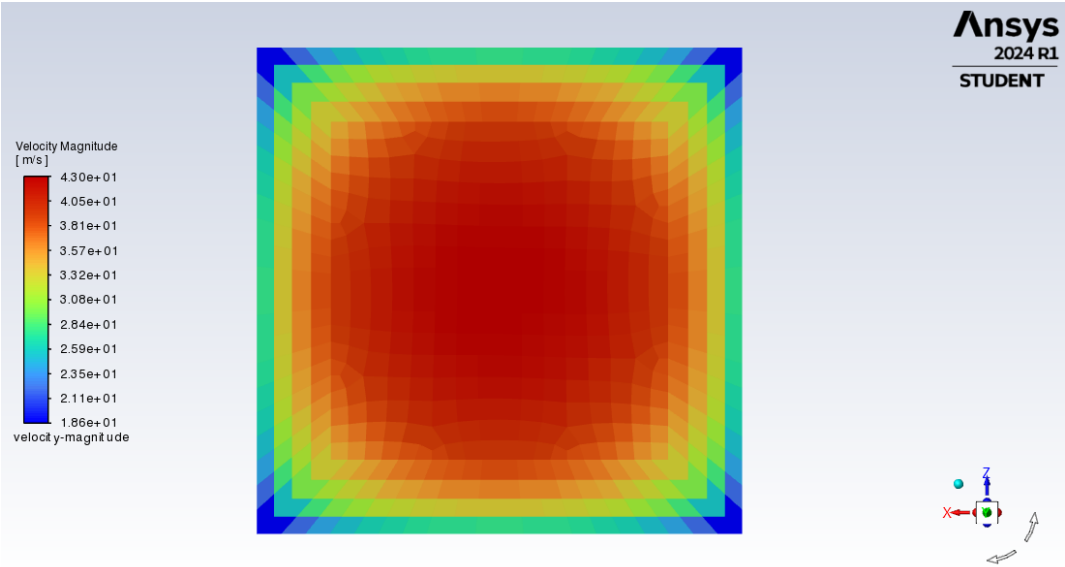
- Define proper contours and plots to visualize the data. Confirm that the y^+ is greater than 30.

y^+ -plus

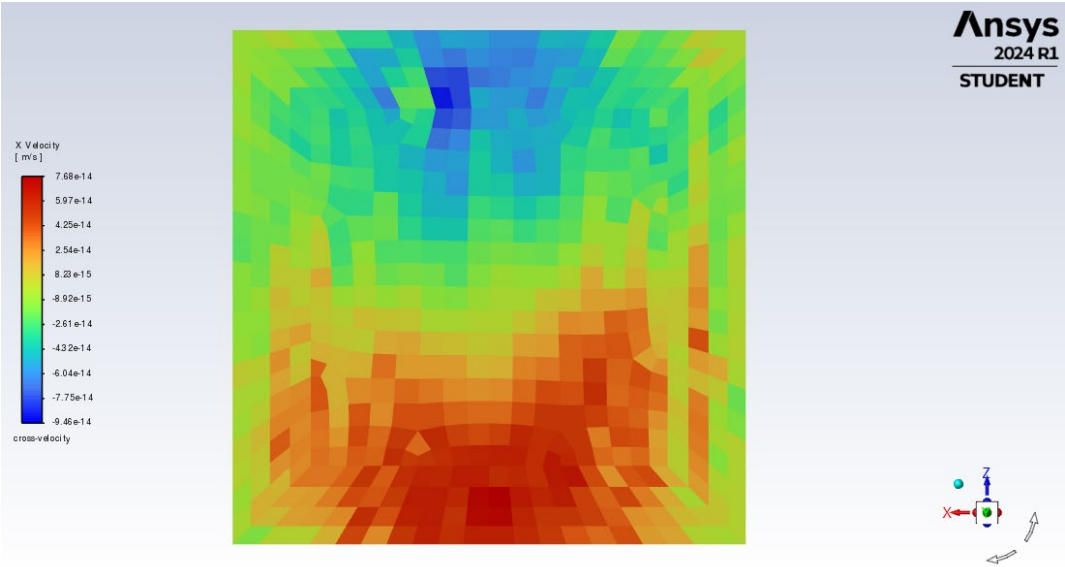


Author	CFDLab@Energy		Page 8 of 13
CFD for Nuclear Engineering	Session 2		

Streamwise velocity

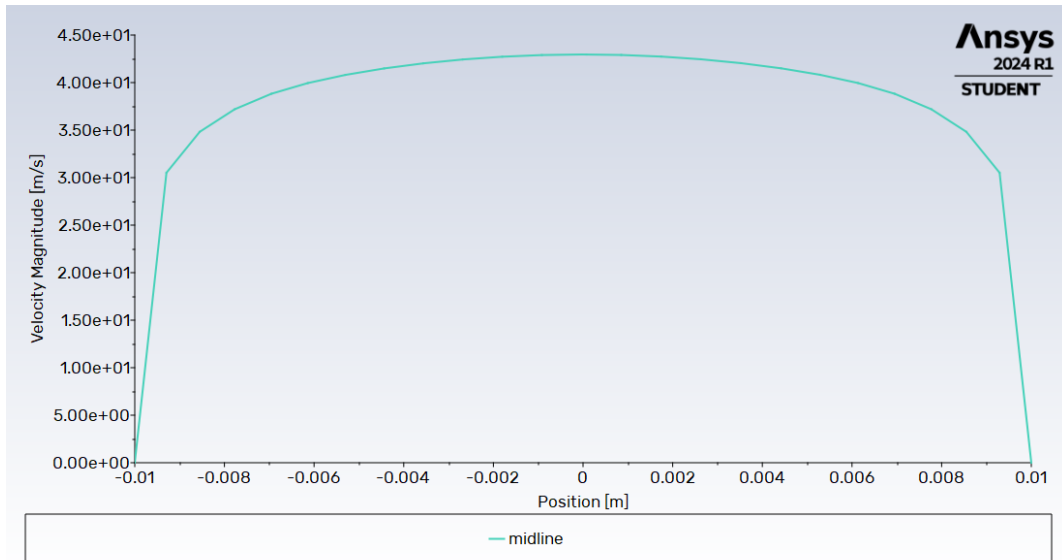


Cross-stream velocity




The cross-stream velocity is **zero** despite clear secondary flows exist in reality.

Velocity Profile

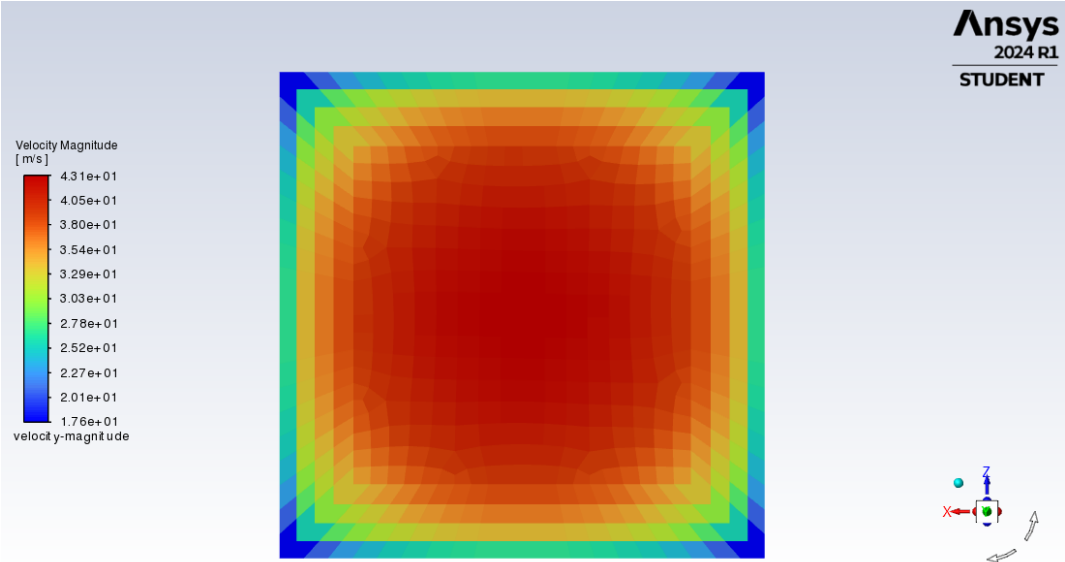


3.2 Simulation with the k- ω Standard

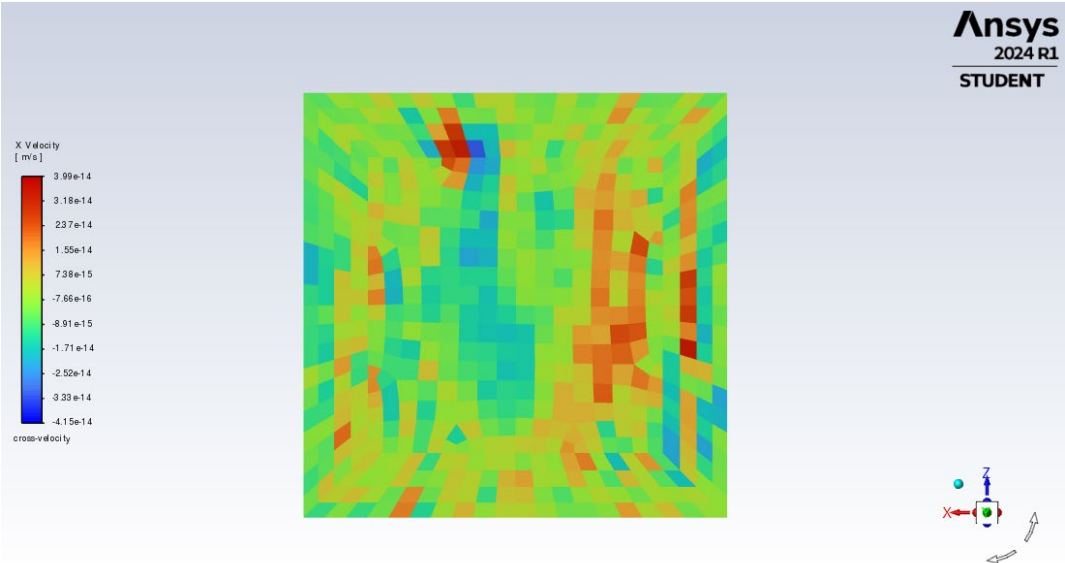
- Define the viscous term as Standard k- ω without any correction or options
- Judge the convergence based on: residuals, reports of meaningful physical quantities.
- Define proper contours and plots to visualize the data.

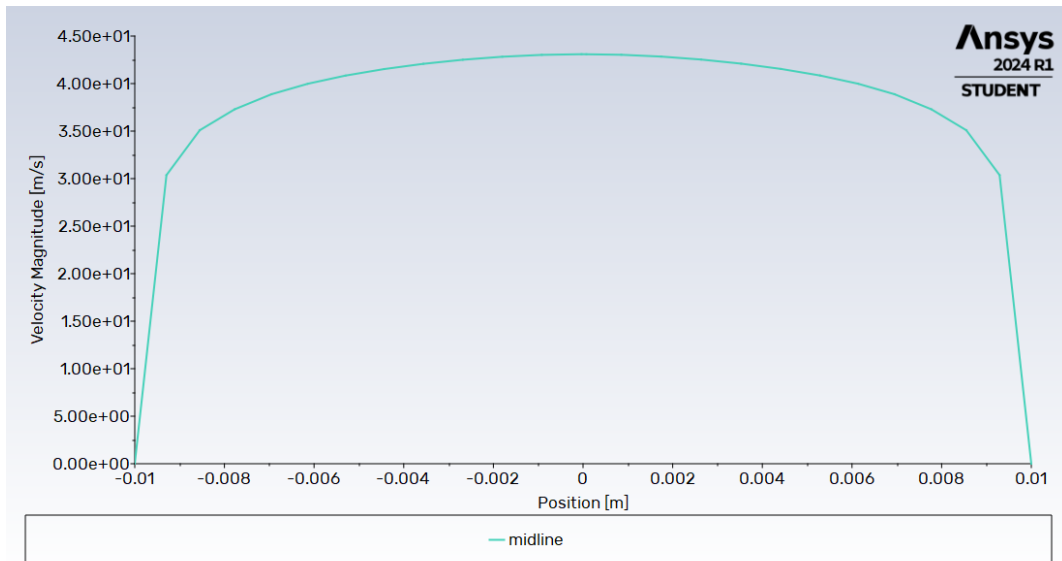
Author	CFDLab@Energy		Page 10 of 13
CFD for Nuclear Engineering	Session 2		

Streamwise velocity




Cross-stream velocity

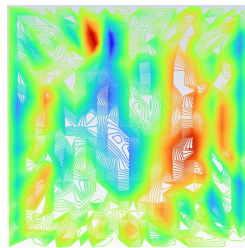
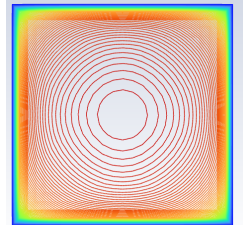




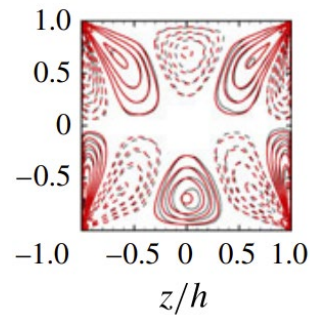
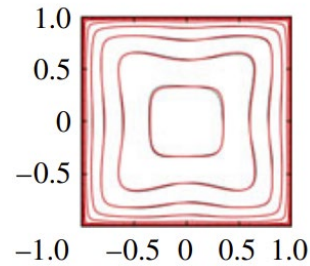
4 Conclusions

The formulation of Standard $k-\epsilon$ and Standard $k-\omega$ cannot predict the cross-stream velocity because of the assumption of isotropic turbulence. However, the corners in the squared channels create anisotropy of turbulence which results in cross-stream velocity and secondary flows. We will see in the next section how to take into account the anisotropy of turbulence.


Author	CFDLab@Energy	 Page 12 of 13
CFD for Nuclear Engineering	Session 2	

k- ϵ Standard

Pirozzoli et al, 2018, DNS

Contours of
streamwise
velocityContours of
cross stream
velocity

The picture above shows the k- ϵ Standard results, but they would be exactly the same for k- ω Standard. We can qualitatively see that secondary flows do not exist in the CFD case and what we see is just a numerical drawing without any meaning (the scale is 10^{-14} m/s). The streamwise flow also appears as the one of a pipe which is not affected by the secondary motion. The effect of the secondary motion is clear in the contours of the streamwise velocity in the DNS results.

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