

---


## Hands-on session 4 – Turbulent Pipe Flow

### Abstract

This session will introduce you to the simulation of the simplest turbulent flow, the **fully developed turbulent pipe flow**. The student will start with the geometry generated in Hands-on Session 3 with the most appropriate grid. For this exercise the Reynolds number is assumed to be equal to 133,000 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 diameter of the pipe is 50 mm. The exercise will be done with two different turbulence models, that is to say k- $\epsilon$  Standard and k- $\omega$  Standard.


### Goal

The aim of this hands-on session is to get familiar with the existing turbulence modeling and wall treatment. The details of the related theory will be given in the next lectures. The students will learn how to simulate a turbulent pipe flow in a periodic domain and compare the velocity profile with data existing in literature for Direct Numerical Simulation from “Pirozzoli et al. ‘One-point statistics for turbulent pipe flow up to  $Re_\tau \approx 6000$ ’, Journal of Fluid Mechanics”.

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

---

<b>Hands-on session 4 – Turbulent Pipe Flow .....</b>	<b>1</b>
<b>1 Introduction .....</b>	<b>3</b>
<b>2 Restart from the Fully Developed Laminar Flow Case .....</b>	<b>4</b>
2.1 Read the case .....	4
2.2 Define the Mass Flow Rate .....	4
2.3 Attempt the Simulation without Turbulence Model .....	4
2.3.1 Results of the simulation without turbulence model .....	4
2.4 Set-up k- $\epsilon$ Standard Turbulence Model.....	6
2.4.1 Save the Case.....	7
2.4.2 Results.....	7
2.5 Set-up k- $\omega$ Standard Turbulence Modeling.....	11
2.5.1 Write the Case .....	11
2.5.2 Results.....	11


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

---

# 1 Introduction

The fully developed turbulent pipe flow will be used to test the necessity of turbulent models to simulate high Reynolds number flows. The fully developed turbulent pipe is the fundamental and simplest case to start the discussion about turbulence and wall treatment. As always it is also a good way to strengthen the knowledge of the code itself: plotting contours, velocity profile and check the convergence.

**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 Restart from the Fully Developed Laminar Flow Case

### 2.1 Read the case

It is convenient to restart from the laminar case with the appropriate mesh selected, and restart the turbulent simulation from the flow field already established in the laminar case.

### 2.2 Define the Mass Flow Rate

In the periodic case we need to set the mass flow rate to guarantee the prescribed Reynolds number, 133,000. In this case, please input  $9.35 \cdot 10^{-2} \text{ kg/s}$  in the Periodic Conditions.

### 2.3 Attempt the Simulation without Turbulence Model

When we set in Fluent “Laminar” for the Viscous model we are simply solving the plain Navier-Stokes equations as below:

Continuity

$$\frac{\partial \rho}{\partial t} + \text{div}(\rho \mathbf{u}) = 0$$

Momentum


$$\frac{\partial \rho \mathbf{u}}{\partial t} + \text{div}(\rho \mathbf{u} \mathbf{u}) = -\text{div}(p) + \text{div}(\mu \text{grad } \mathbf{u})$$

The Navier-Stokes equations are derived from mass and momentum balance and the generalization of the Newton law of viscosity. Hence, for Newtonian fluids which are all the fluids where the viscous stresses are linearly correlated to the strain rate, the equations do not contain any approximation. So, it would be logical to try to simulate a turbulent flow with such an approach.

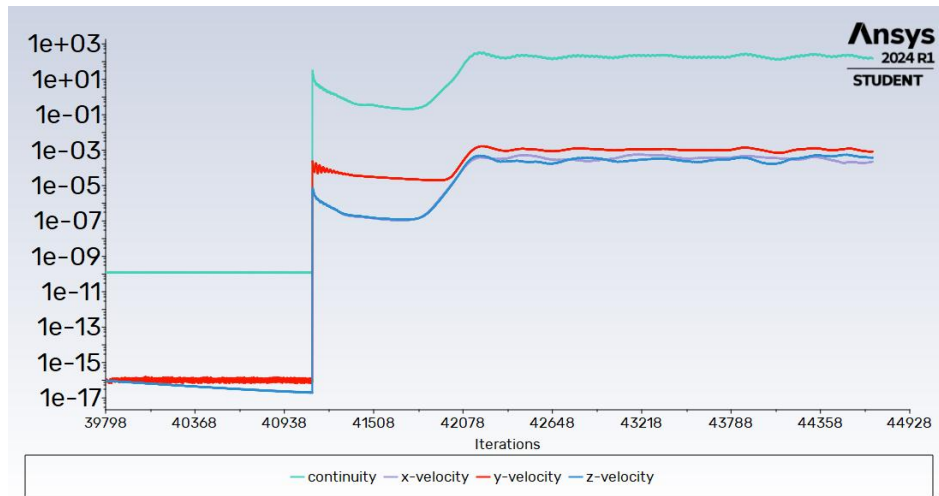
#### 2.3.1 Results of the simulation without turbulence model

For this simulation simply write a new case which you could be named for example: “FullyDevelopedTurbulentPipeFlow\_noTurbulenceModel.cas.h5” and run the simulation.

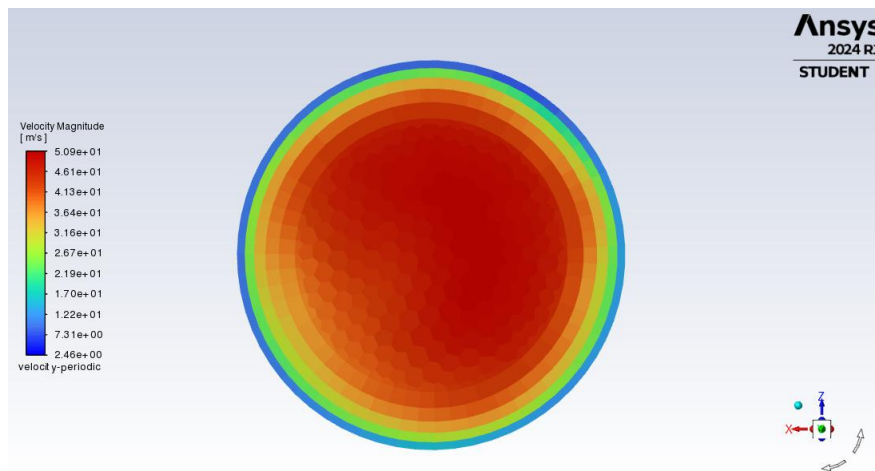
How do the residuals behave?


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

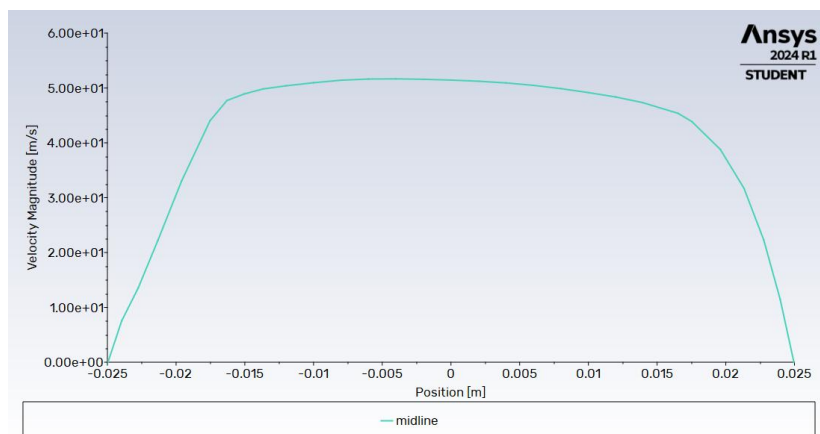
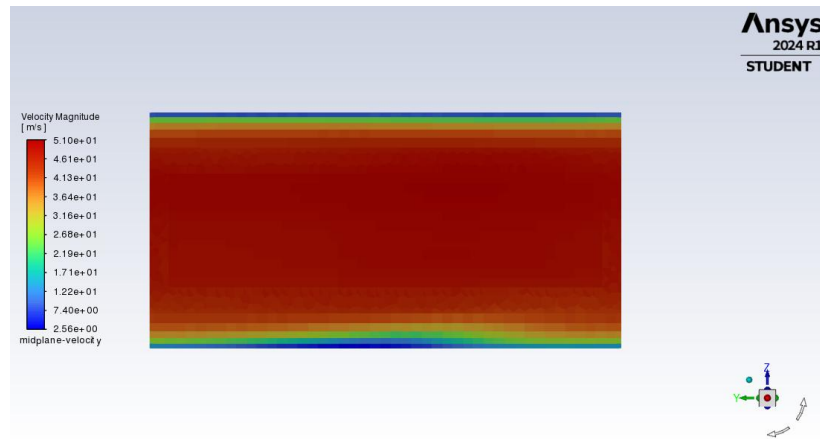
How about the velocity contours and the velocity profile?



Despite starting from a fully converged solution the code clearly does not converge. The velocity contours on the midplane and on the periodic boundary, and the velocity distribution also show clearly the inability of the code to converge as the flow is not simetrical and the velocity fields keep changing at each iteration.




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



Why do you think the calculation behaves in such a way?

## 2.4 Set-up k- $\epsilon$ Standard Turbulence Model

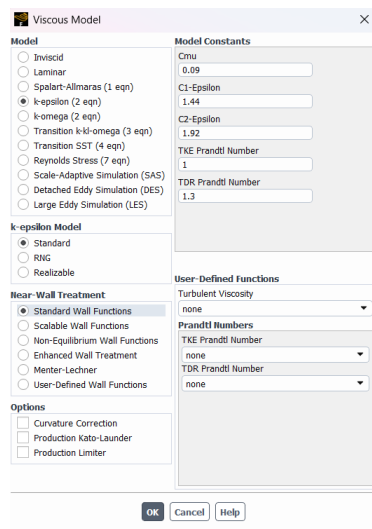
Despite the simplicity of the flow, even for a straight pipe the turbulent structures are very complex and very small. To simulate such flow, it would be necessary to have an extremely fine mesh, smaller than the turbulent structures, and solve an unsteady flow. Fortunately, in many engineering problems we do not need to know such details but the knowledge of the **average** values is enough. The averaged equations take the name of Reynolds Average Navier-Stokes equations, or RANS. The details will be given in future classes.

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

We are asked to compare two RANS turbulence models. There are a very large number of models but for the time being we will compare the 2 equations models, that is to say: k- $\epsilon$  and k- $\omega$  in their standard definition. For the definition of the turbulence modeling double click on Models -> Viscous and select:

- Model: k-epsilon (2 eqn)
- k-epsilon Model: Standard
- Near-Wall Treatment: Standard Wall Functions

Leave other options unchecked and do not modify the Model Constants and the Prandtl numbers. The input should look like as below.



## 2.4.1 Save the Case


Write the Case with a meaningful name.

## 2.4.2 Results

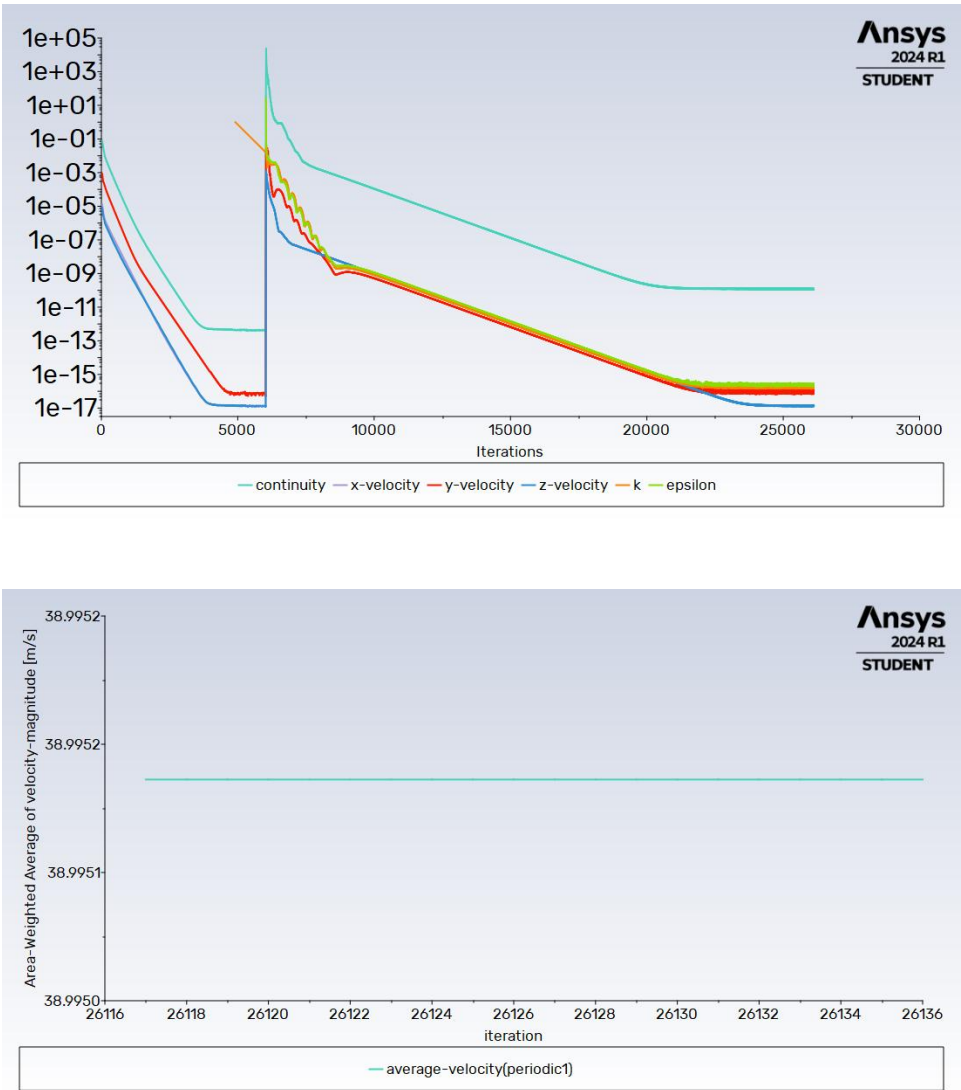
The organization of the results generally consists of the following steps:

1. Confirm convergence
2. Plot qualitative results, generally contours
3. Plot quantitative results and compare them with existing analytical, experimental or numerical data

### 2.4.2.1 Confirm the Convergence

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

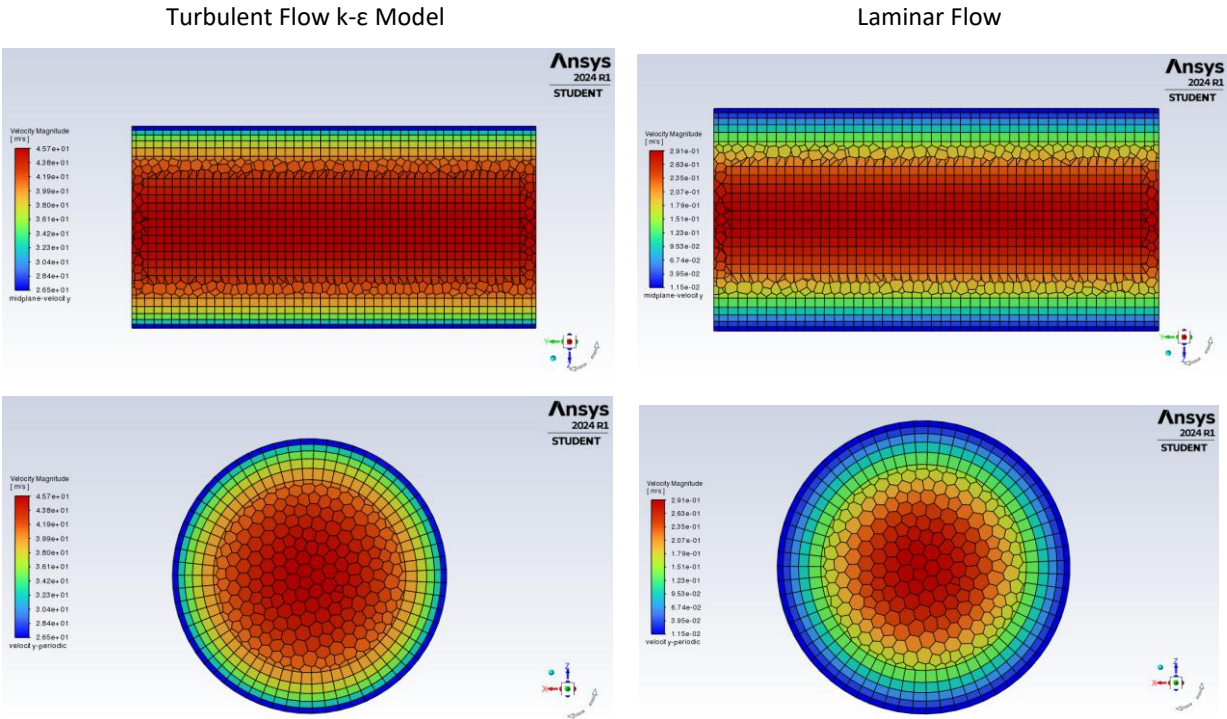
In the turbulent case the level of convergence is completely satisfactory. The plot of the quantities confirms this aspect.





2.4.2.2 Plot the Contours of Velocity

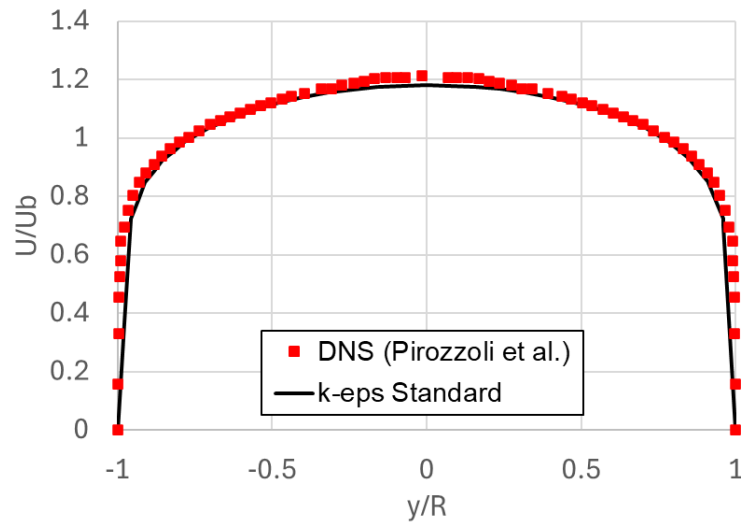
It is interesting to compare the contour plot between the turbulent and the laminar case.



What do you notice?

2.4.2.3 Plot the velocity profile

The results of the k-ε Standard agrees very well with the DNS results.



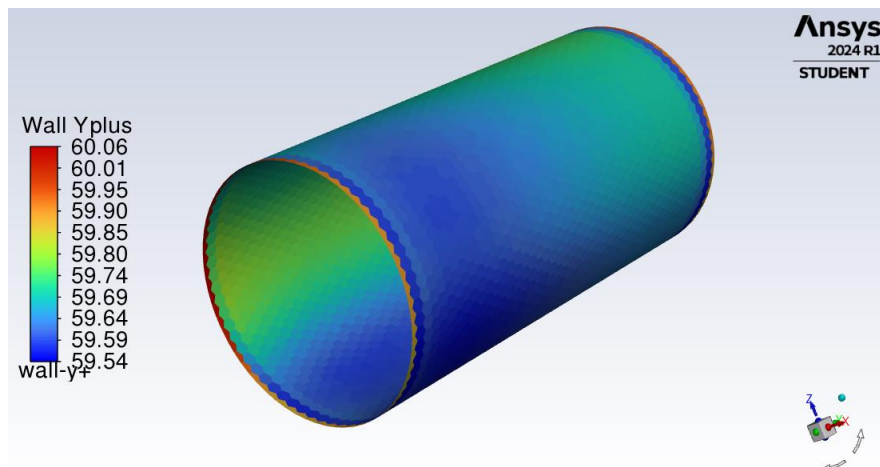
#### 2.4.2.4 Plot the Wall $y^+$

As it will be presented in the future classes the adoption of the “Standard Wall Functions” requires the Wall  $y^+$  to be larger than 30.


To show the contours of the Wall  $y^+$  create a new contour and choose:

- Contours of: Turbulence -> Wall Yplus

The result should appear similar to the picture below:



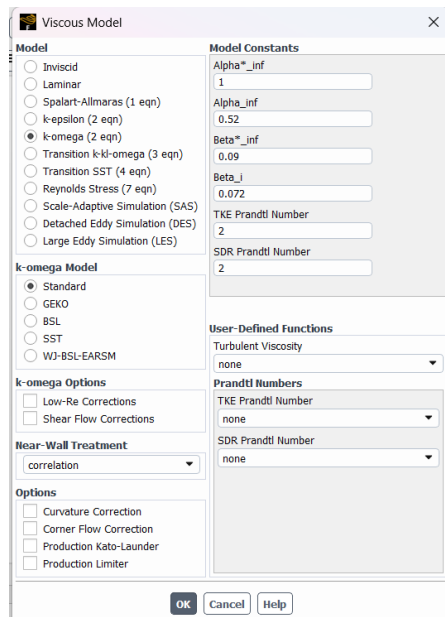
We can confirm that the requirement of  $y^+ > 30$  is satisfied in all the wall cells in the present simulation.

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

## 2.5 Set-up k- $\omega$ Standard Turbulence Modeling

For the definition of the turbulence modeling double click on Models -> Viscous and select:

- Model: k- $\omega$  (2 eqn)
- k- $\omega$  Model: Standard
- k- $\omega$  Options: deselect Shear Flow Corrections
- Near-Wall Treatment: correlation
- Options: deselect Production Limiter




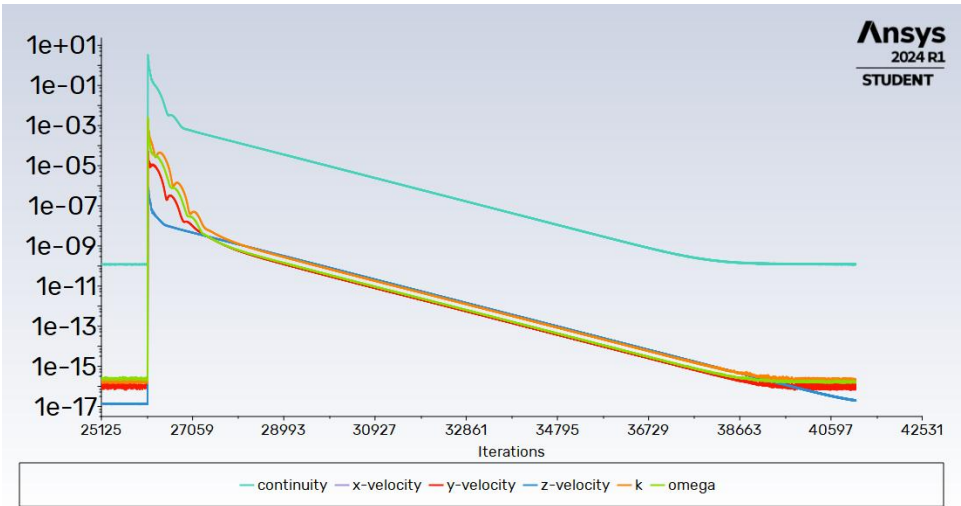
### 2.5.1 Write the Case

Write the Case with a meaningful name.

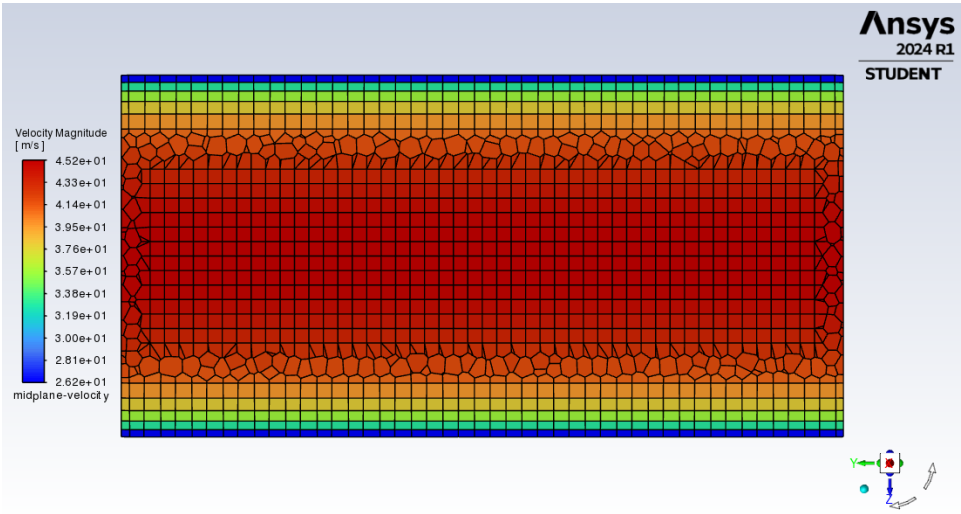
### 2.5.2 Results

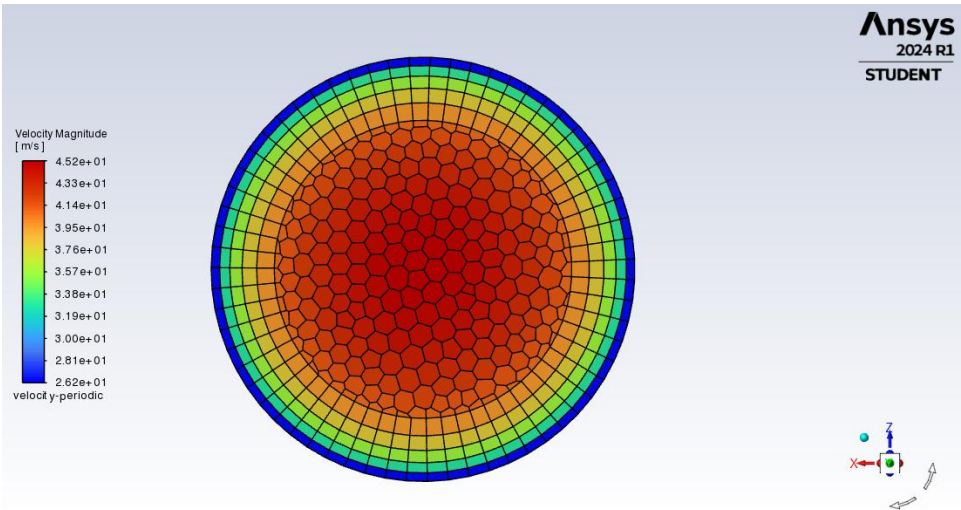
#### 2.5.2.1 Confirm the Convergence

Author	CFDLab@Energy	 Page 11 of 13
CFD for Nuclear Engineering	Session 2	



2.5.2.2 Plot the Contours of Velocity





2.5.2.3 Plot the velocity profile

The k- $\omega$  Standard results also show very good agreement with the DNS data. In this example there is basically no difference between employing k- $\epsilon$  Standard and k- $\omega$  Standard model.

