MARMARA
U N I V E R S I T Y

FACULTY OF ENGINEERING

MECHANICAL ENGINEERING

ME4095

SPECIAL TOPICS

LID DRIVEN CAVITY FLOW ANALYSIS

STUDENT NAME   Oğuzalp ÇETİN

STUDENT NUMBER   150418031

SUBMITTED TO   PROF. DR. Mehmet Zafer GÜL

Content

List of Figures

1. Introduction

The main aim of this project is to show the interaction between fluid viscosity and a cavity. In this project, ANSYS Fluent software is used to model the 2-D lid driven cavity flow.

For the 2D lid driven cavity flow problem, we are required to present the ANSYS Fluent CFD analysis results for flows at different Reynolds number and developed in a lid driven square cavity.

U=1m/s, V=0 (Lid) ──────────➤

U=0 , V=0 (Wall)   CAVITY   U=0 , V=0 (Wall)

y

U=0 , V=0 (Wall)

x
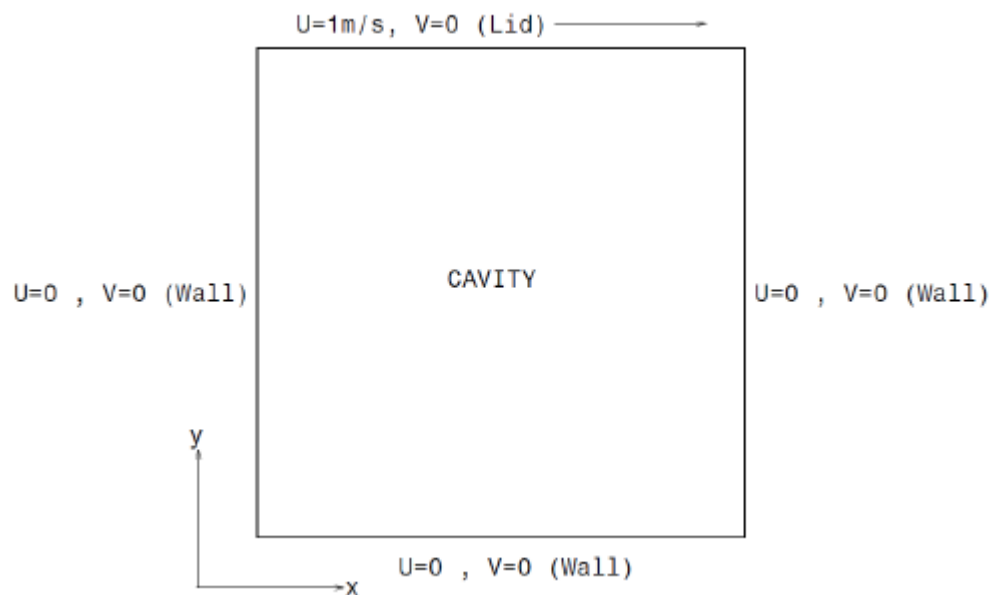
Figure 1: 2D Lid Driven Cavity Boundary Condition Diagram

2. Simulation Setup

a. MESH

In reference paper, mesh grids are concentrated at the boundaries. A Fine mesh of 120 x 100 shown in Figure-2 is used for both 3200 and 10000 Reynolds numbers.
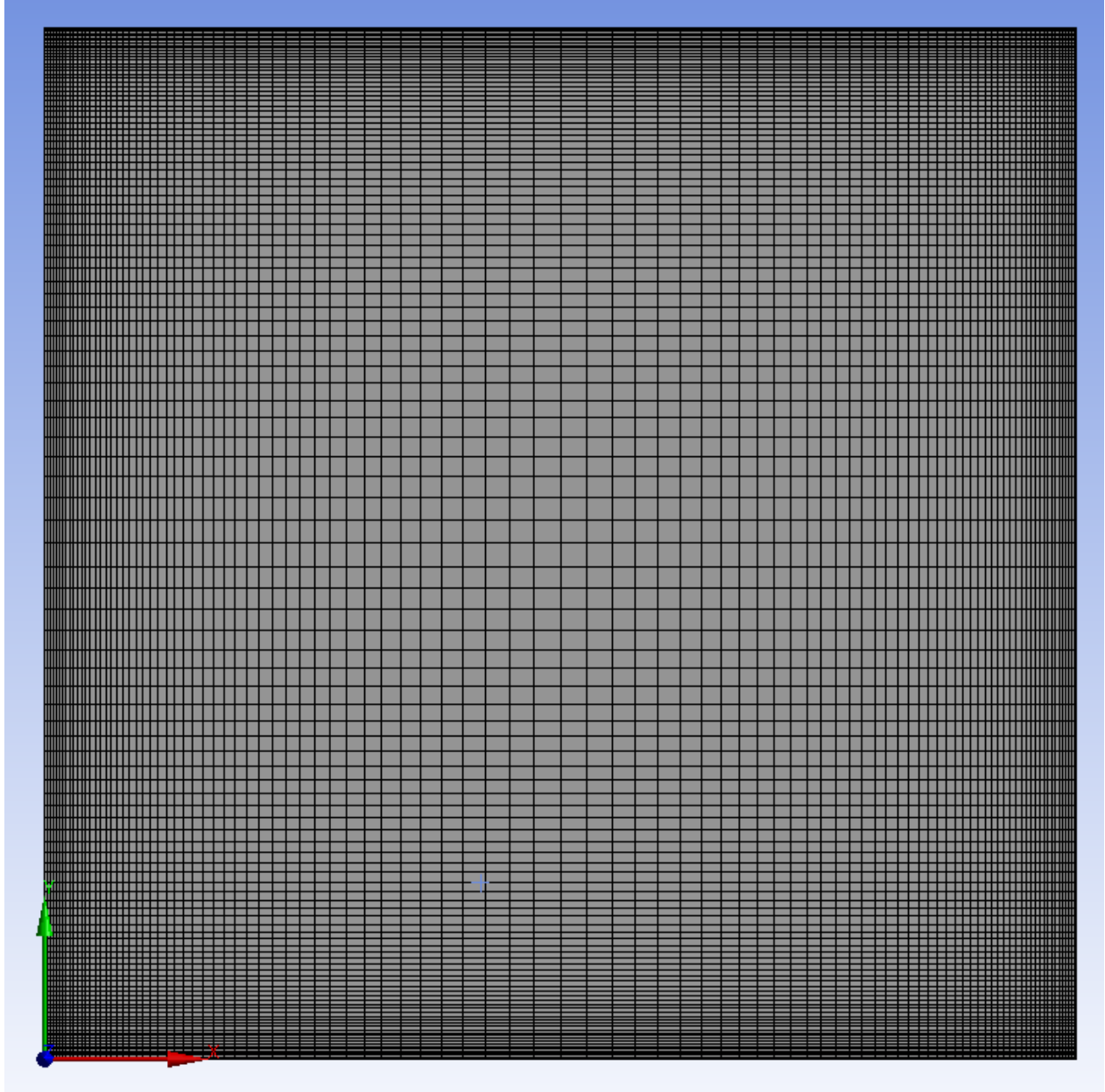

Figure 2: Mesh

To generate this mesh, first create a 1x1 m geometry. Then edge sizing is inserted for the lengthwise and breadthwise edges with edge definition changed from element size to number of divisions and number of divisions is given as 120 and 100 respectively. A double extreme end biased spacing is used with biased factor set to 12 and 10 respectively. (This concentrates the mesh at the edge boundaries). Finally, face meshing is used to keep the mesh lines straight.
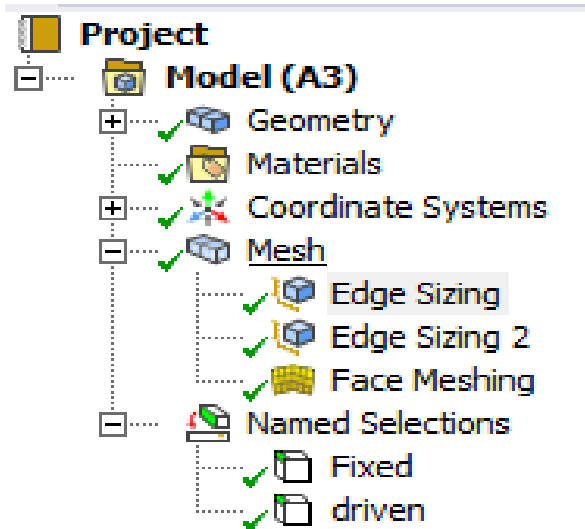
Figure 3: Mesh components

Details of "Edge Sizing 2" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	120
Advanced	
Behavior	Soft
Growth Rate	Default (1.2)
Capture Curvature	No
Capture Proximity	No
Bias Type	– --- ----- --- –
Bias Option	Bias Factor
Bias Factor	12.0

Figure 5: Edge sizing-2

Details of "Edge Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	100
Advanced	
Behavior	Soft
Growth Rate	Default (1.2)
Capture Curvature	No
Capture Proximity	No
Bias Type	– --- ----- --- –
Bias Option	Bias Factor
Bias Factor	10.0

Figure 4: Edge sizing-1

Details of "Face Meshing" - Mapped Face Meshin

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Suppressed	No
Mapped Mesh	Yes
Method	Quadrilaterals
Constrain Boundary	No
Advanced	
Specified Sides	No Selection
Specified Corners	No Selection
Specified Ends	No Selection

Figure 6: Face meshing

Details of "Mesh"

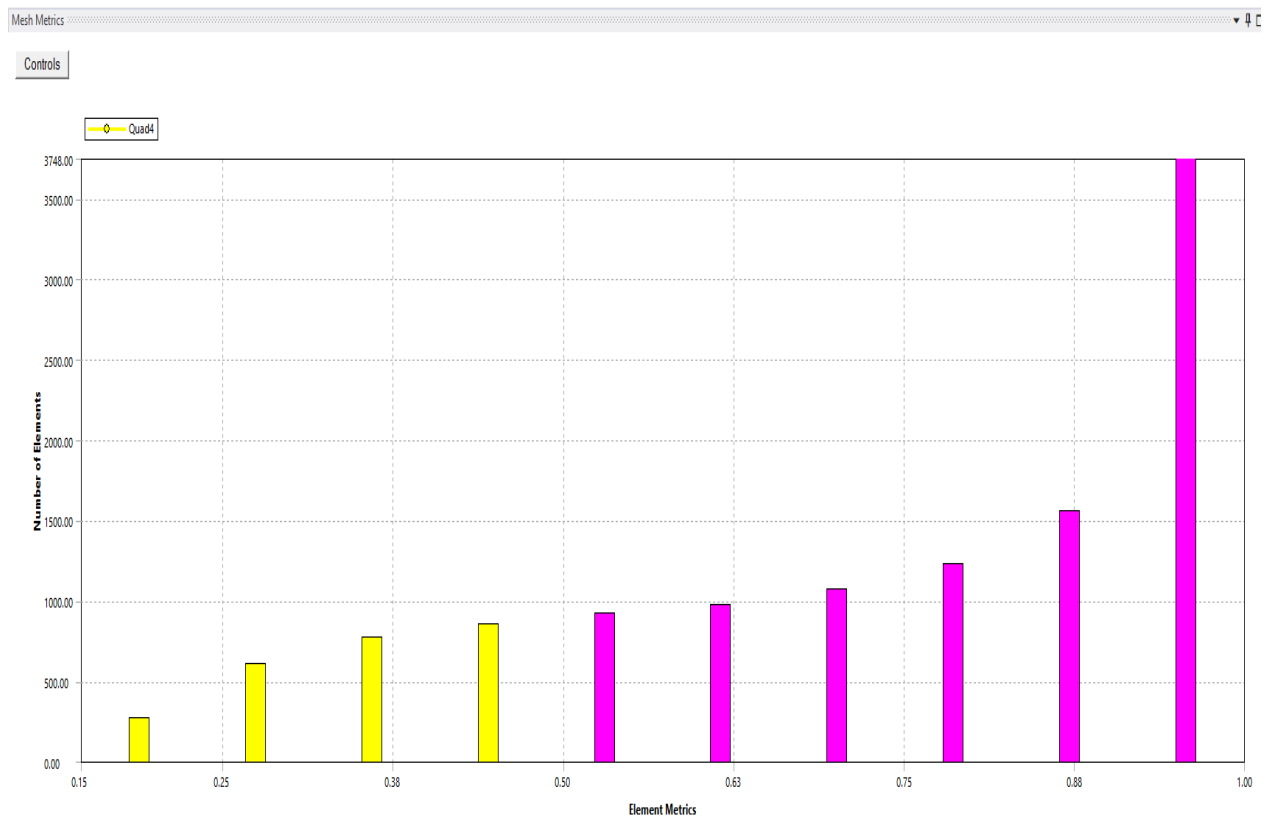|		
⊟	**Display**	
	Display Style	Use Geometry Setting
⊟	**Defaults**	
	Physics Preference	CFD
	Solver Preference	Fluent
	Element Order	Linear
	☐ Element Size	Default (70.711 mm)
	Export Format	Standard
	Export Preview Surface Mesh	No
⊞	**Sizing**	
⊞	**Quality**	
⊞	**Inflation**	
⊞	**Batch Connections**	
⊞	**Advanced**	
⊞	**Statistics**	

Figure 7: Mesh

Figure 8: Element quality

As seen in Figure-8, quality of many of the elements are near one which is intended. In Figure-9, we can see which elements are with quality over 0.5.
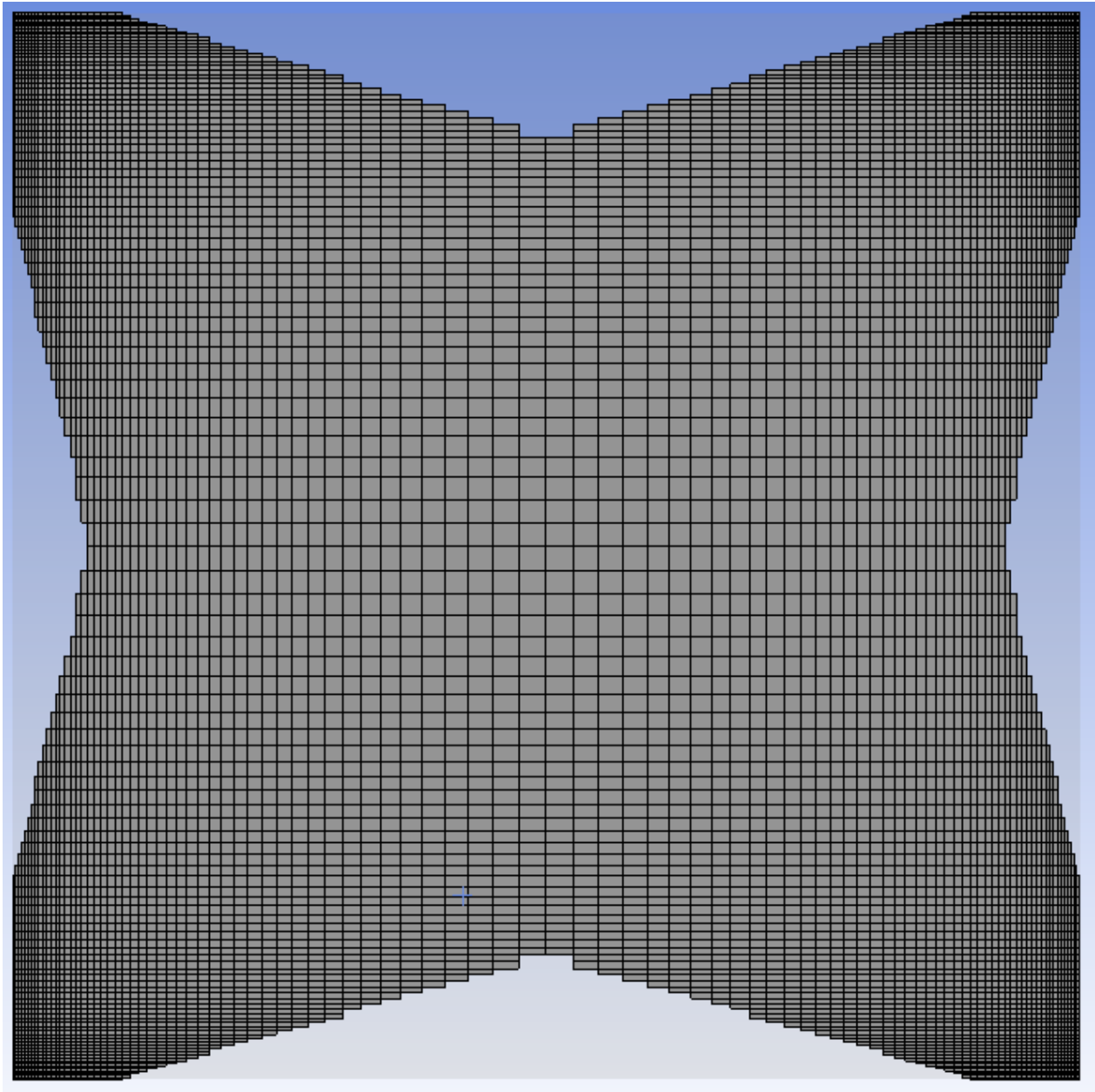
Figure 9: Quality > 0.5 Mesh elements

This is actually what we wanted because we want to focus on corners. So far, mesh is ready to work with.

b. SOLVER

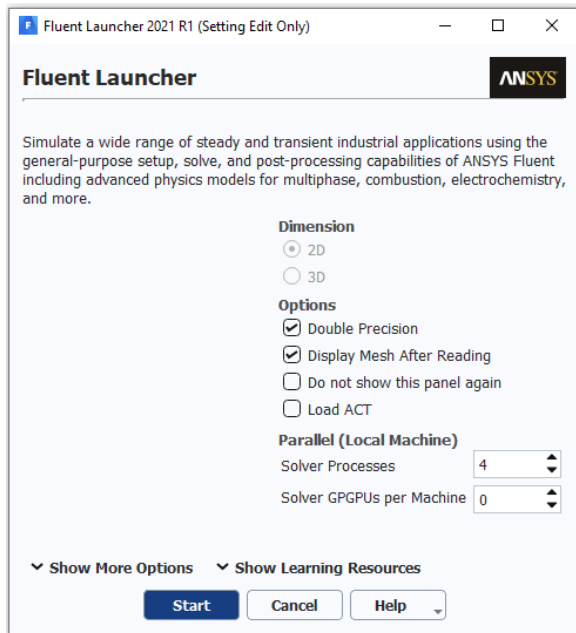Use double precision and set solver processes to 4.

Figure 10: Fluent Launcher

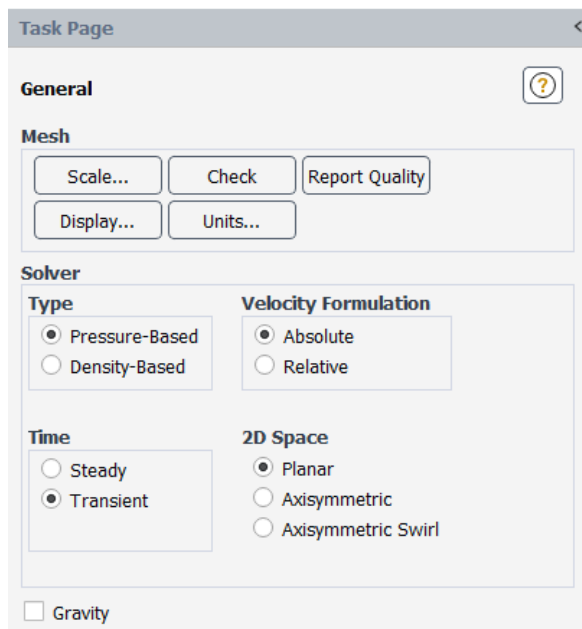From Setup->General select pressure based and transient time.

From Setup->Models->Viscous select k-epsilon (2 eqn).

Figure 12: Viscous model settings

Figure 11: General settings at Setup part
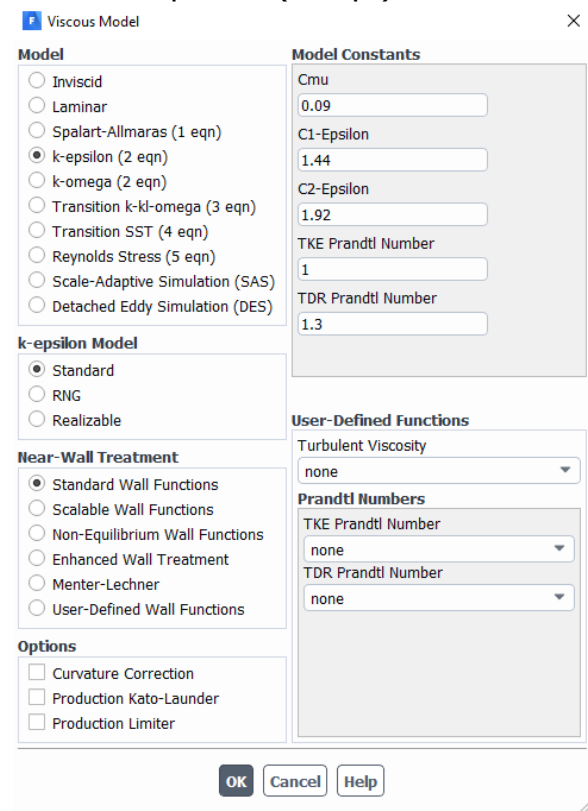
Change properties of the working fluid from Setup->Materials->Fluid. I used named expressions (parameters) to change them easily from main menu.
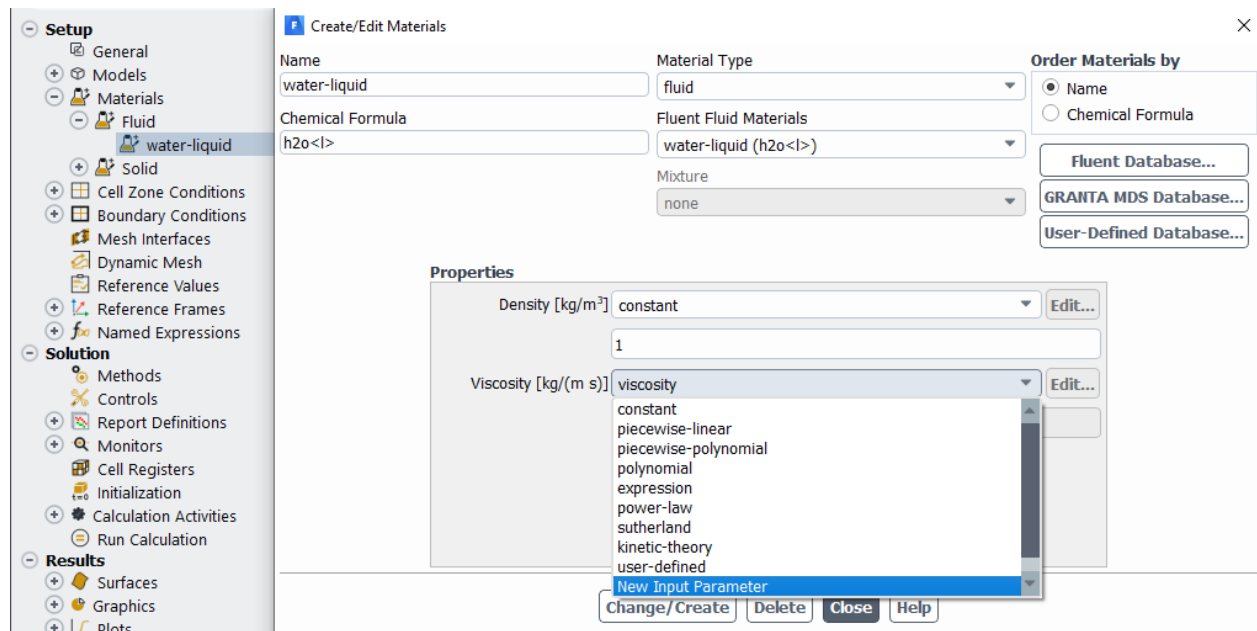

Figure 13: Density and viscosity settings

As we know Reynolds Number is about the flow, not the fluid itself, we can set it to a desired value by changing variables. For our case L=1m, V=1m/s. Also we can set ρ to 1 kg/m^3 to perform the calculations easily.

$$Re = \frac{\rho\, V\, L}{\mu} = \frac{(1\, kg/m^3)\, (1\, m/s)\, (1\, m)}{\mu} = \frac{1}{\mu}$$

For Re = 3200, set µ to 1/3200 which is 0.0003125.

For Re = 10000, set µ to 1/10000 which is 0.0001.

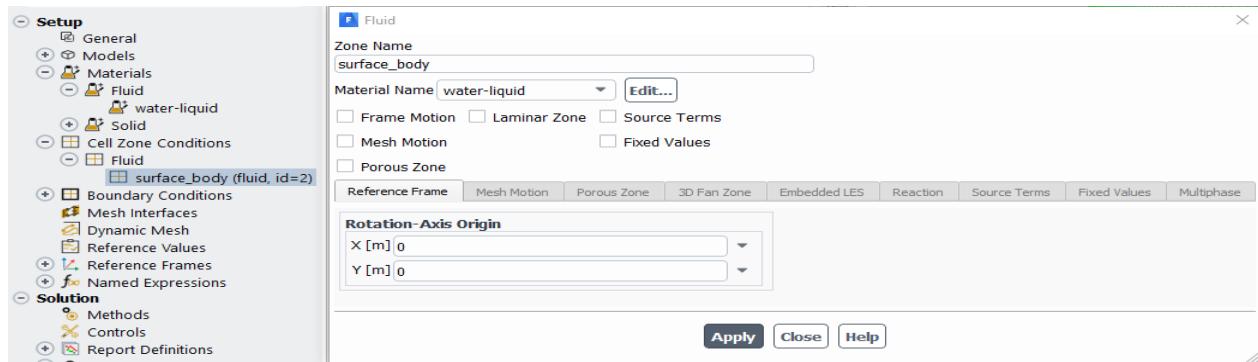Also, we should select the fluid from Setup->"Cell Zone Conditions"->Fluid.

Figure 14: Fluid selection(Set it from material name)

As shown in Figure-15, set the speed to 1 m/s from Setup->"Boundary Conditions"->Wall->Driven. Make sure fixed parts are correctly selected "Stationary Wall" from Setup->"Boundary Conditions"->Wall->fixed.
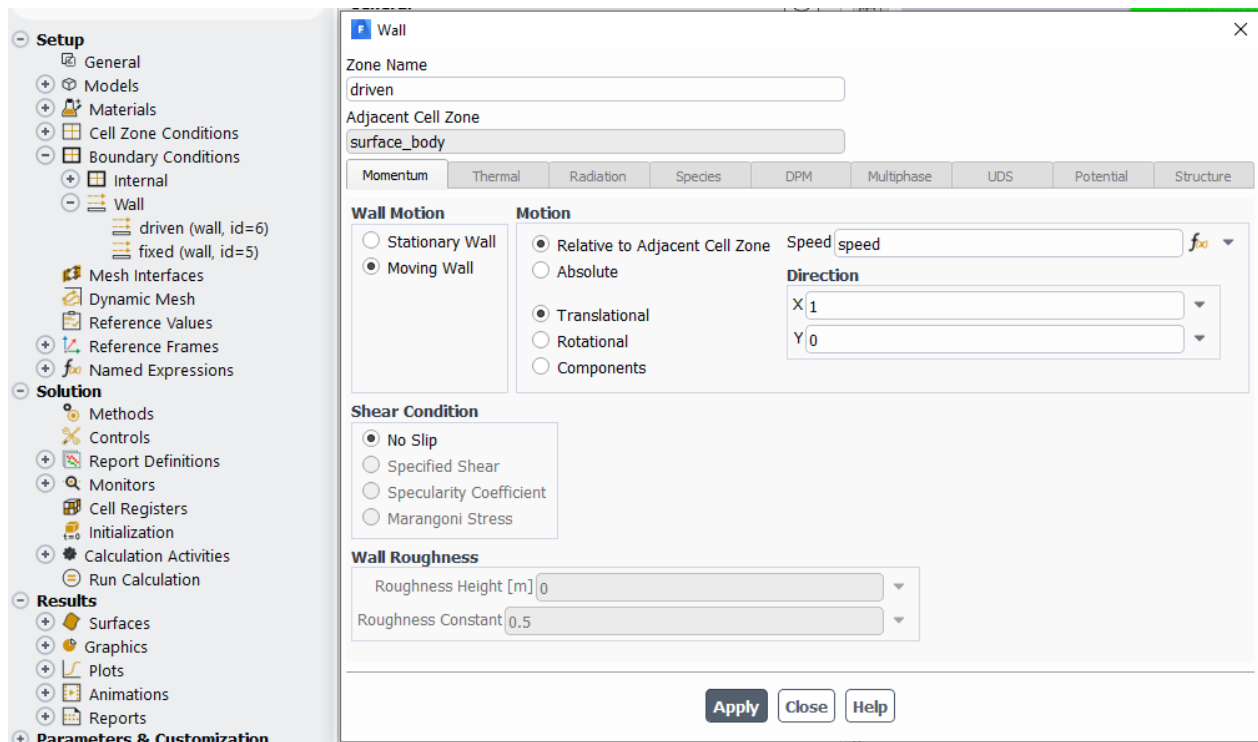

Figure 15: Lid speed (Set it from speed)

From Solution->Monitors->Residuals, set the residuals to 1e-6.

From Solution->"Run Calculation"; set the number of time steps to 150, time step size to 1, max iterations per step to 1000. Finally start iterations by pressing "Calculate" button. Configurations shown in Figure-16.
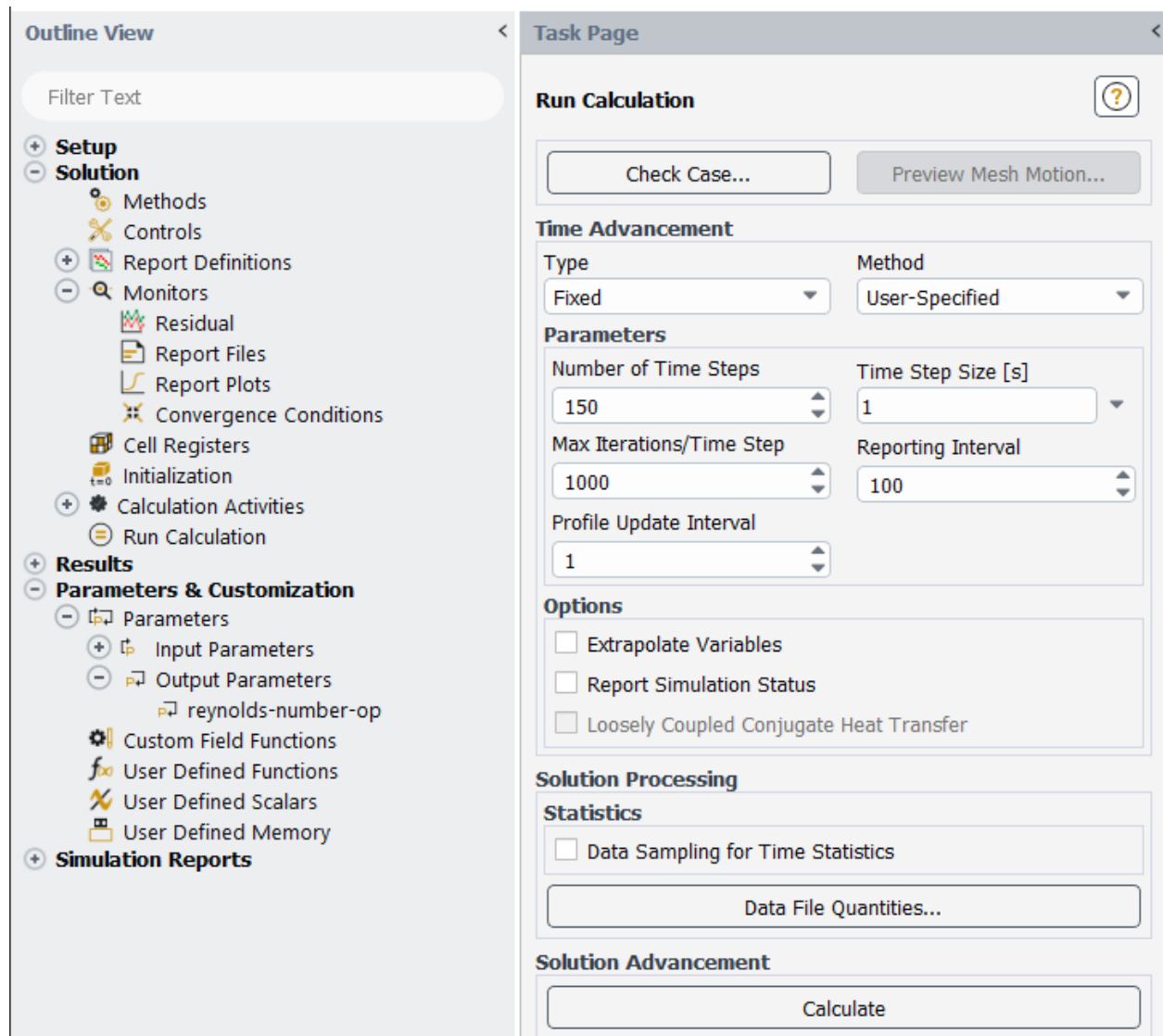

Figure 16: Run Calculation

3. Results

a. MY RESULTS

After calulations, we can see results from ansys post-process model of "6-Results" shown in Figure-17. To change the fluid properties, I entered them as parameters. I changed viscosity to get desired value. I didn't change the density, it is constant and equal to 1. Also, to validate Reynolds Number, I added an output parameter which is basically ρ/μ. See Figure-17 and Figure-18.
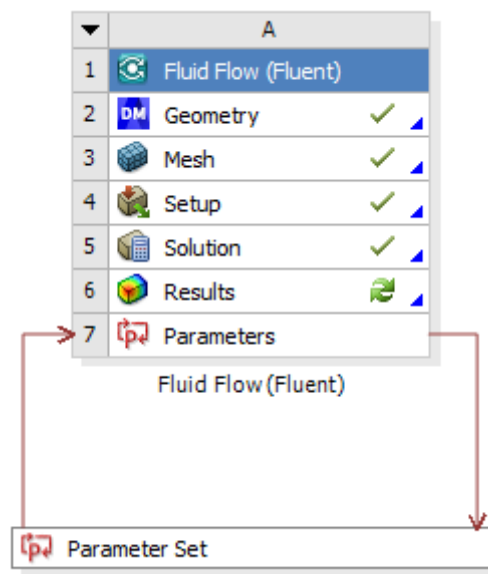

	A
1	Fluid Flow (Fluent)
2	DM Geometry ✓
3	Mesh ✓
4	Setup ✓
5	Solution ✓
6	Results ♻
7	Parameters

Fluid Flow (Fluent)

Parameter Set

Figure 17: Fuent Block

Outline of All Parameters

		A	B	C
1		ID	Parameter Name	Value
2		Input Parameters		
3		Fluid Flow (Fluent) (A1)		
4		P1	density	1
5		P2	viscosity	0.0003125
*		New input parameter	New name	New expression
7		Output Parameters		
8		Fluid Flow (Fluent) (A1)		
9		P3	reynolds-number-op	3200
*		New output parameter		New expression

Figure 18: Re3200, outline of parameters

Set "Start From" part to Vertices, because our meshing idea was to focus on corners, so we should see our results considering it. Also, change the "Max Points" area by trial and error method to get better results. After all, click to "Apply" button.
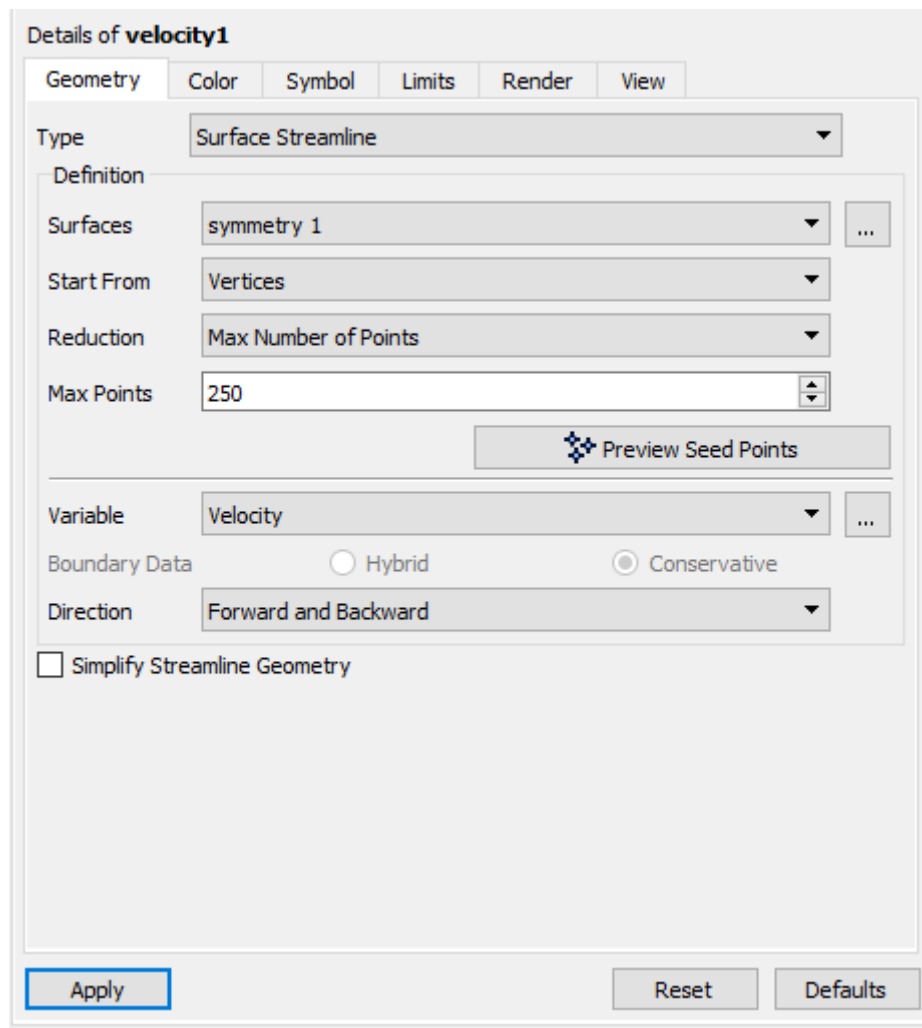

Figure 19: Velocity streamline

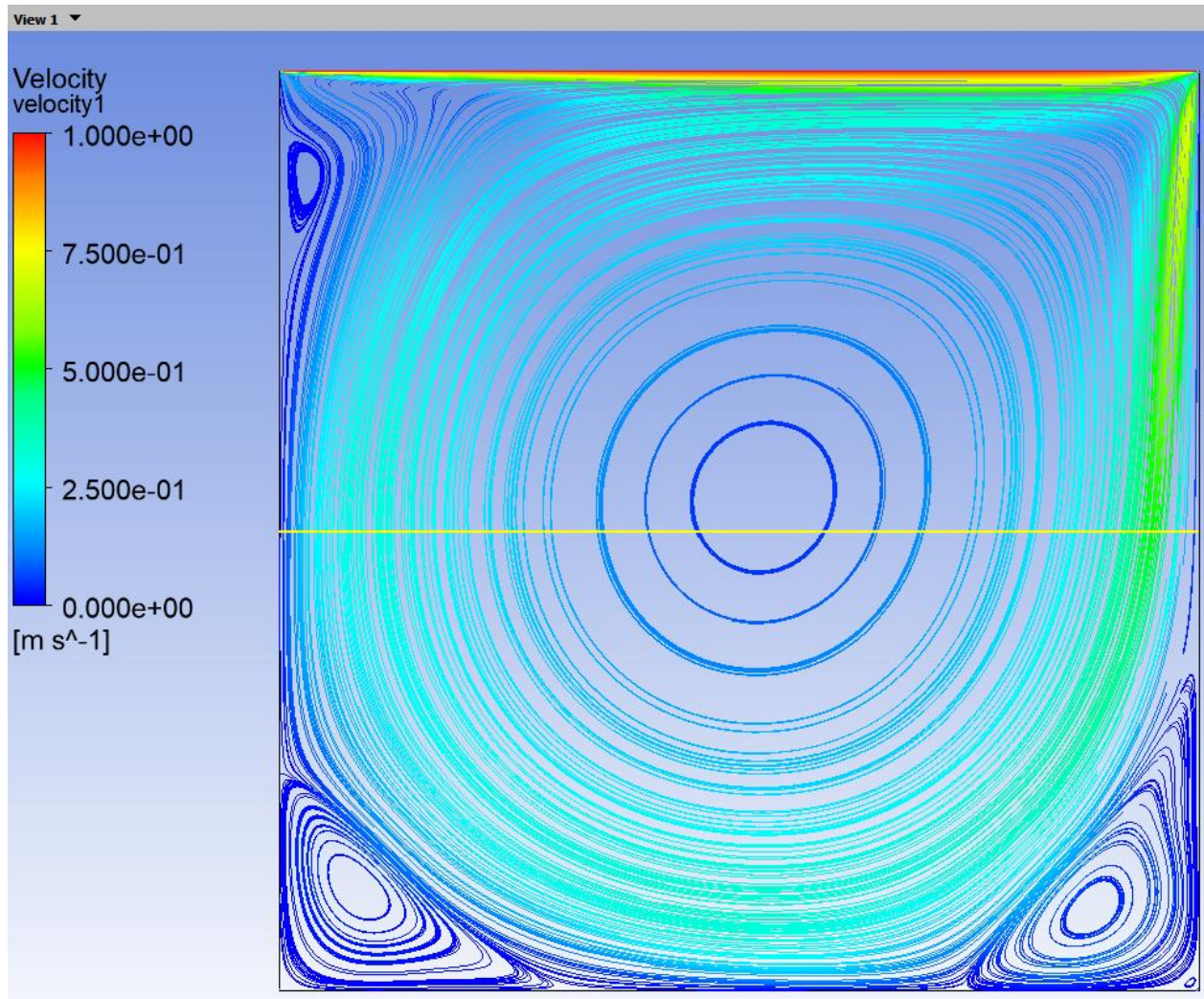Here is the results I get by SIMPLE algorithm, k-epsilon model.
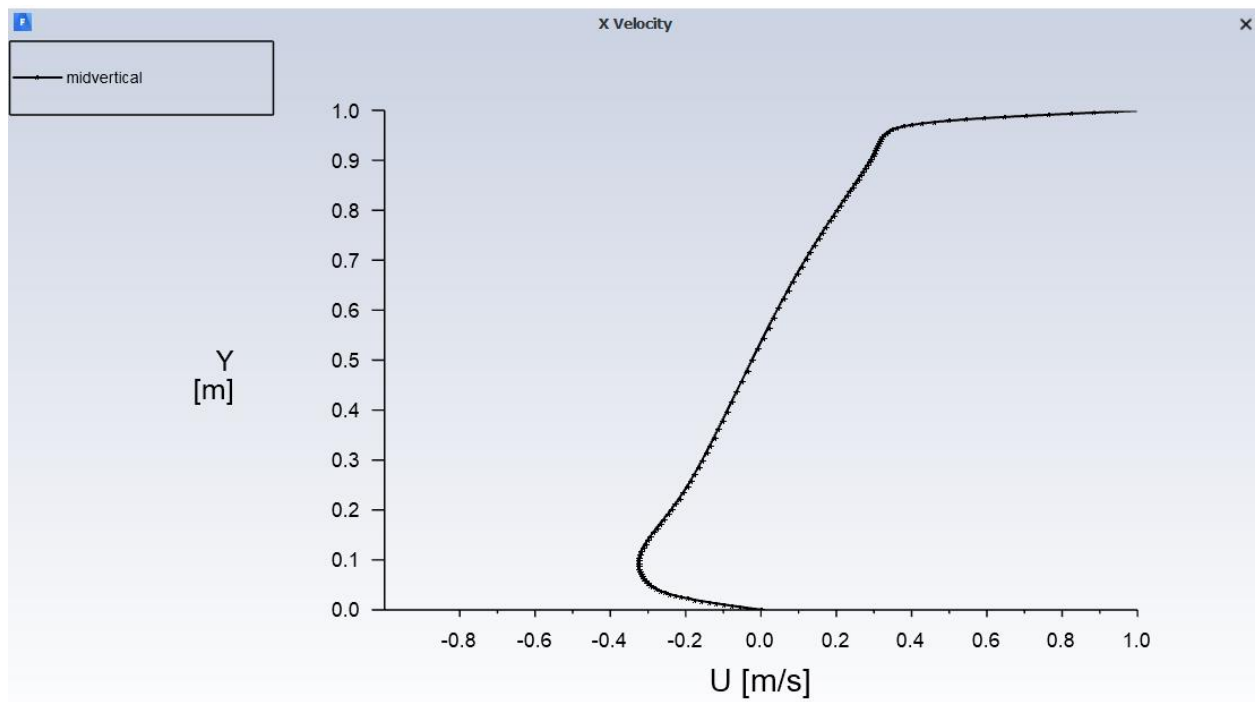

Figure 20: Re3200, Velocity Streamlines

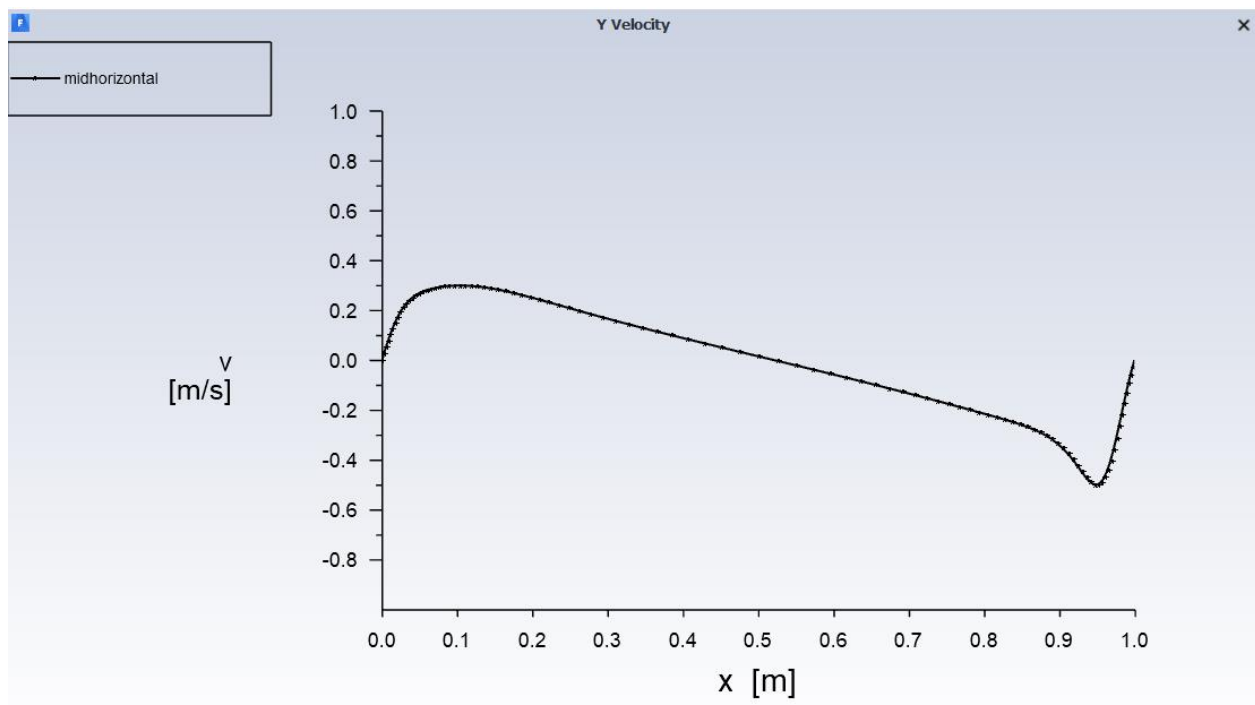*Figure 21: Re3200, U-Component of velocity along mid vertical line*

Figure 22: Re3200, V-Component of velocity along mid horizontal line

Outline of All Parameters			
	A	B	C
1	ID	Parameter Name	Value
2	⊟ Input Parameters		
3	⊟ 🅲 Fluid Flow (Fluent) (A1)		
4	P1	density	1
5	P2	viscosity	0.0001
*	New input parameter	New name	New expression
7	⊟ Output Parameters		
8	⊟ 🅲 Fluid Flow (Fluent) (A1)		
9	P3	reynolds-number-op	10000
*	New output parameter		New expression

Figure 23: Re10000, outline of parameters

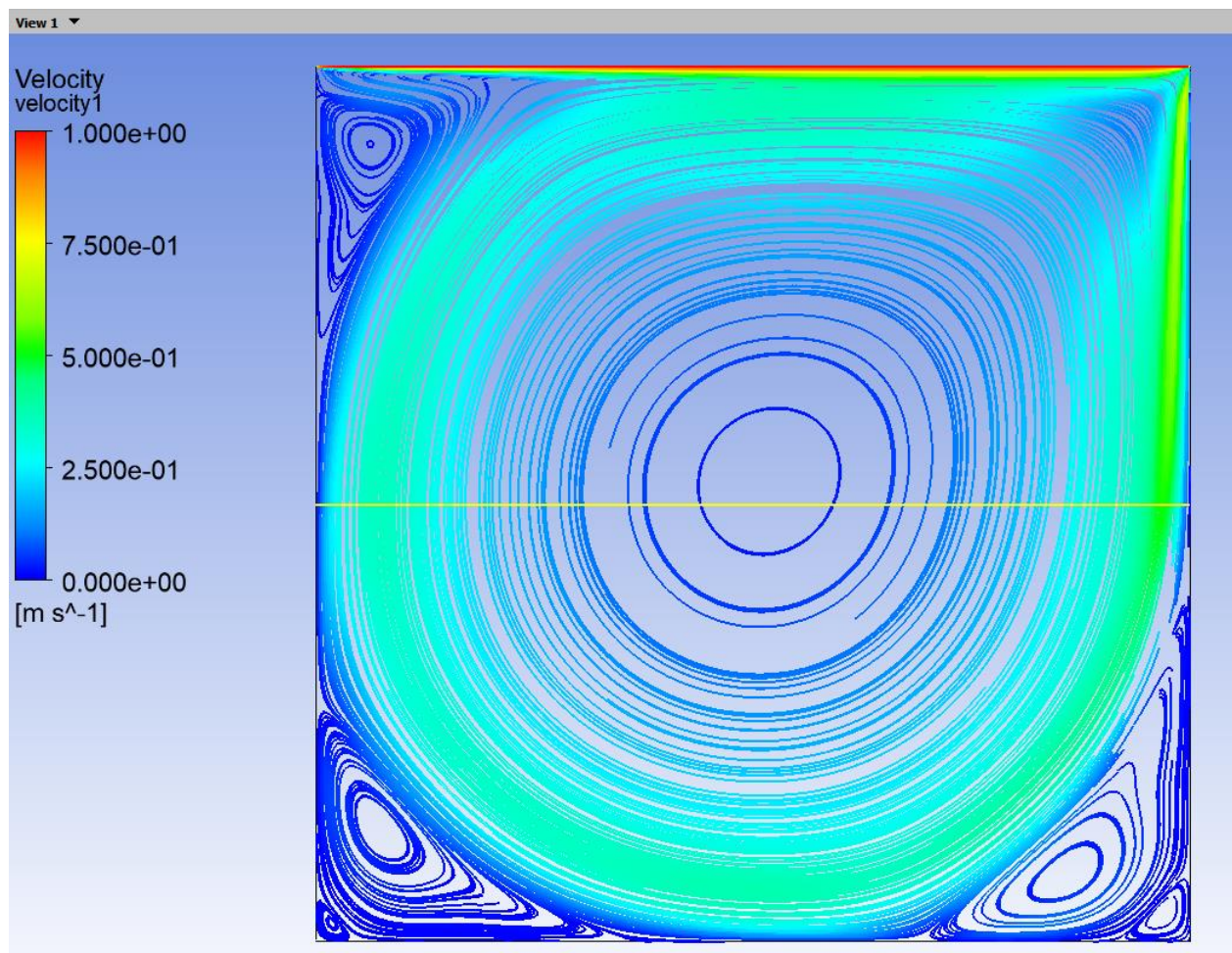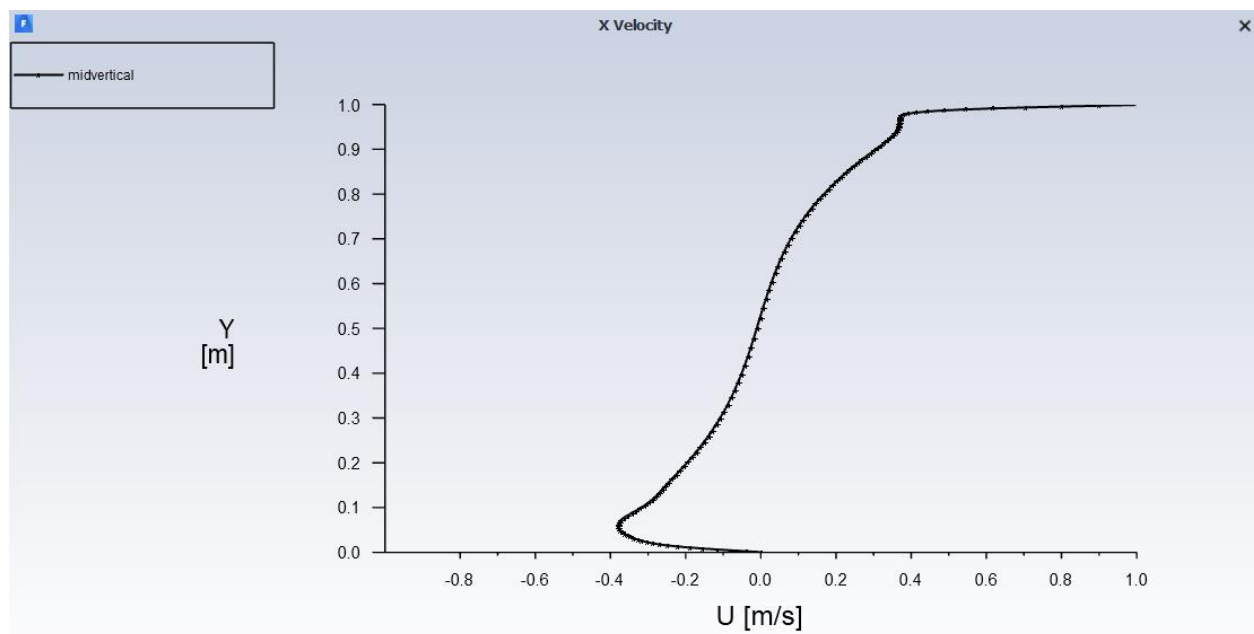Figure 24: Re10000, Velocity Streamlines

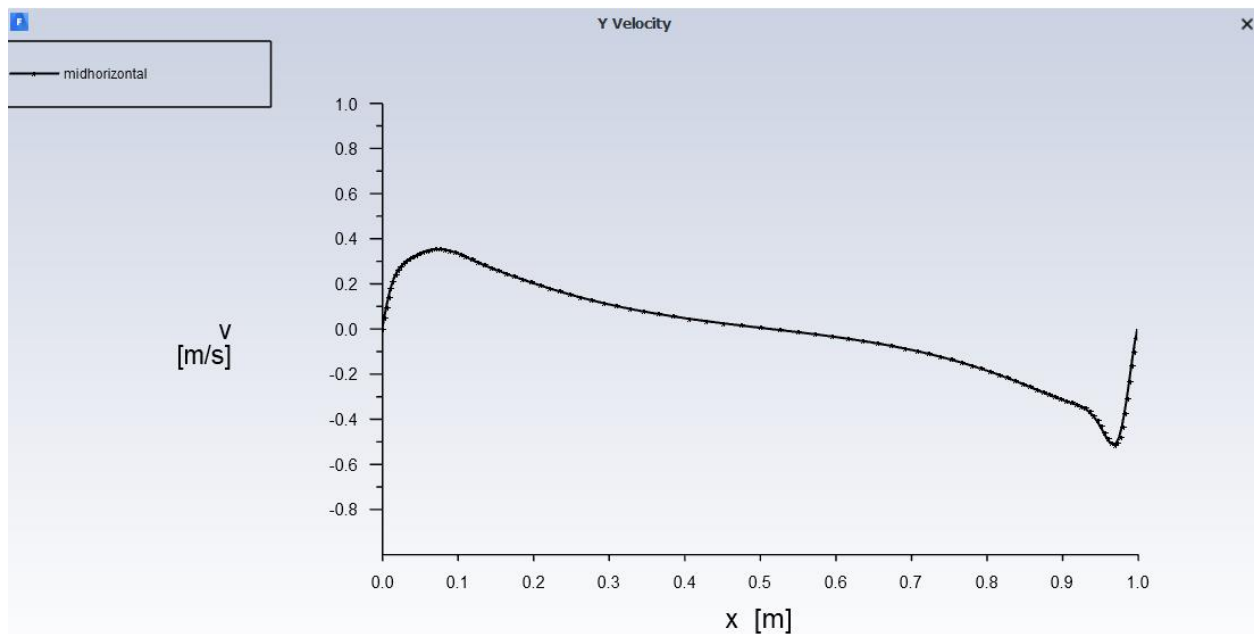*Figure 25: Re10000, U-Component of velocity along mid vertical line*

Figure 26: Re10000, V-Component of velocity along mid horizontal line
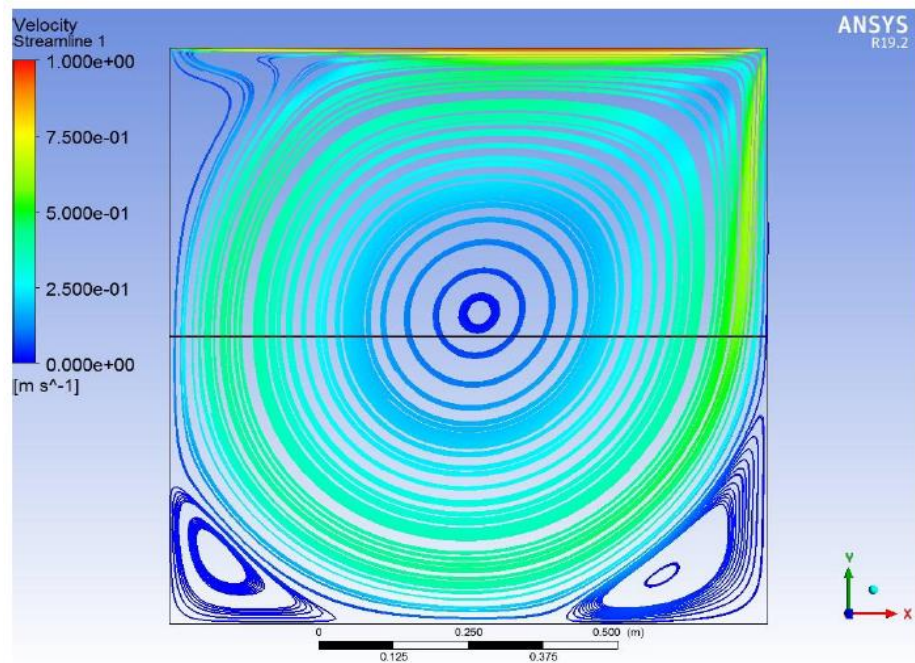
b. RESULTS OF REFERENCE PAPER

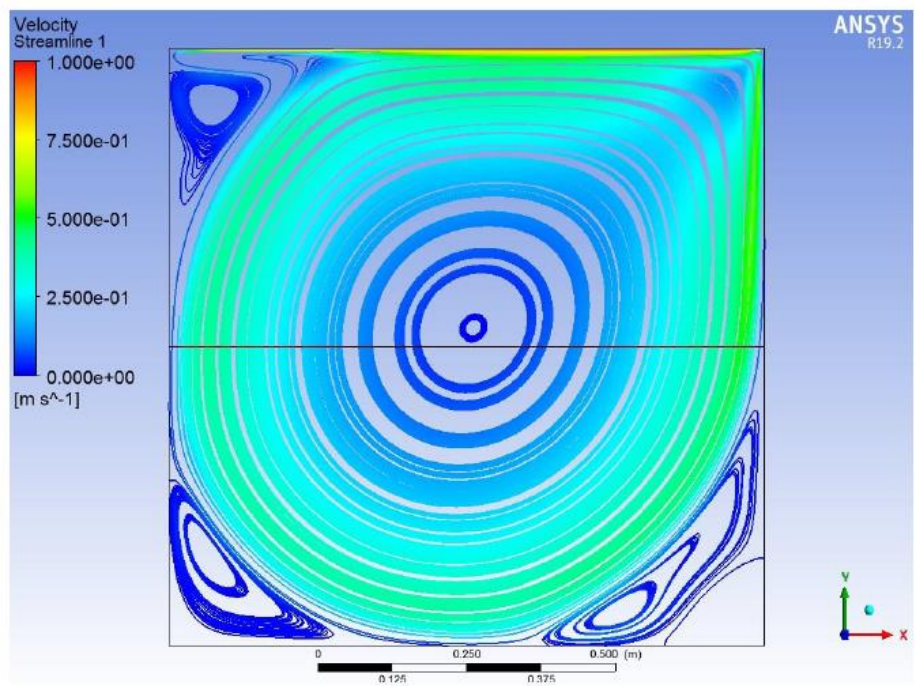Figure 27: Cavity Flow at Re 3200

Figure 28: Cavity Flow at Re 10000

For the velocity components, **blue lines** are the results of my reference paper.

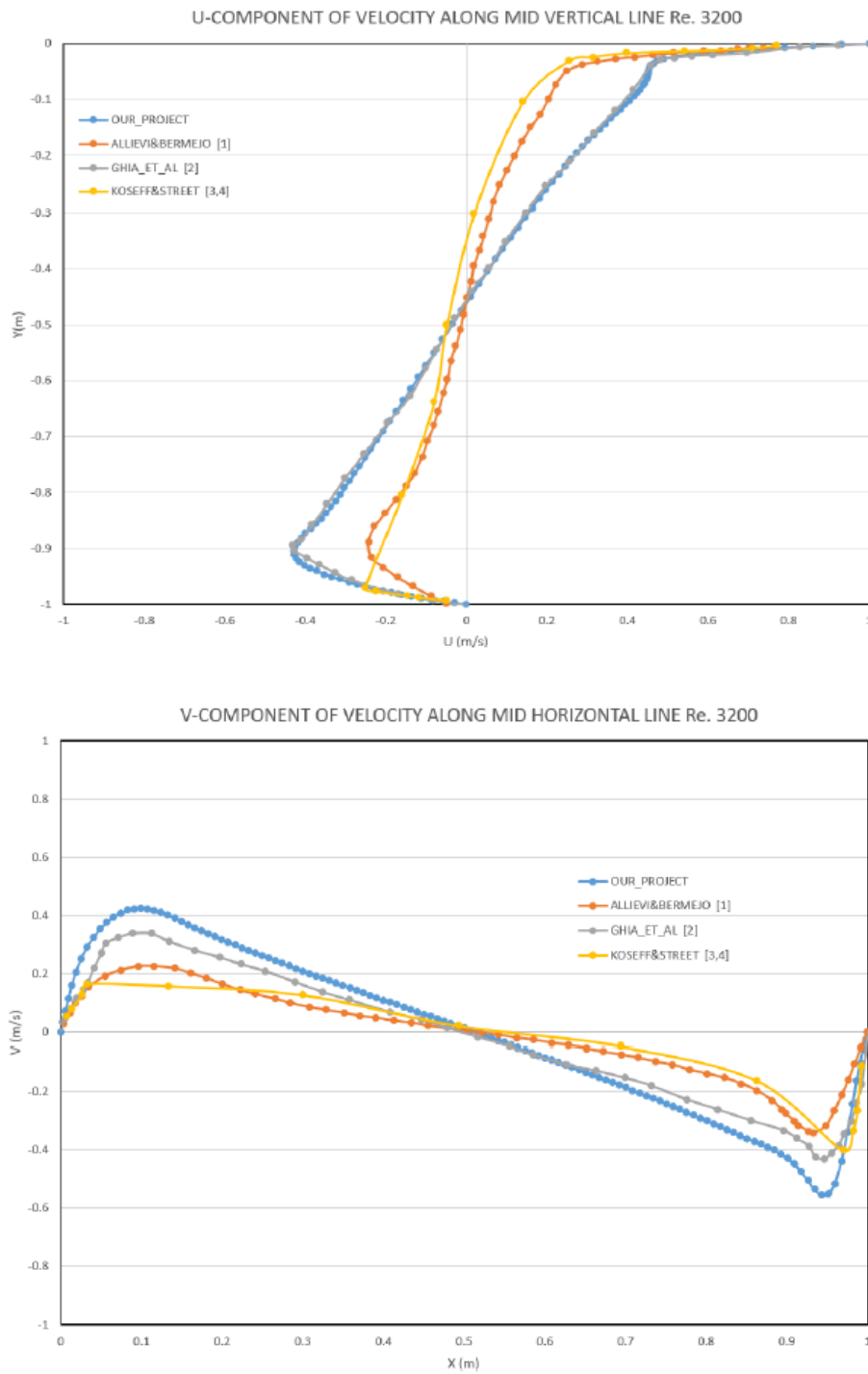➢ U and V velocity Components Comparison Plots for Re= 3200 and 10000

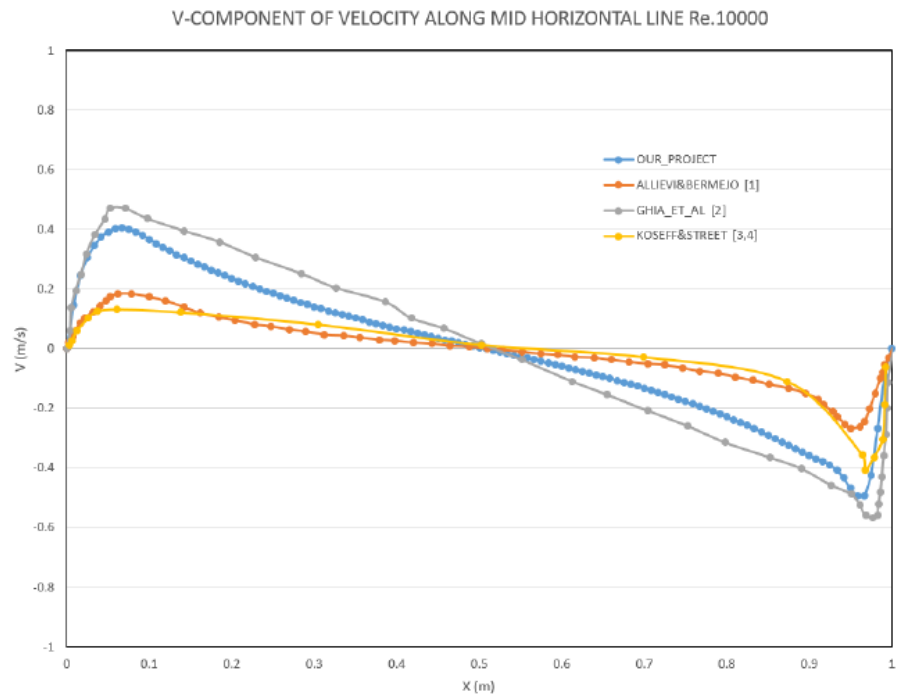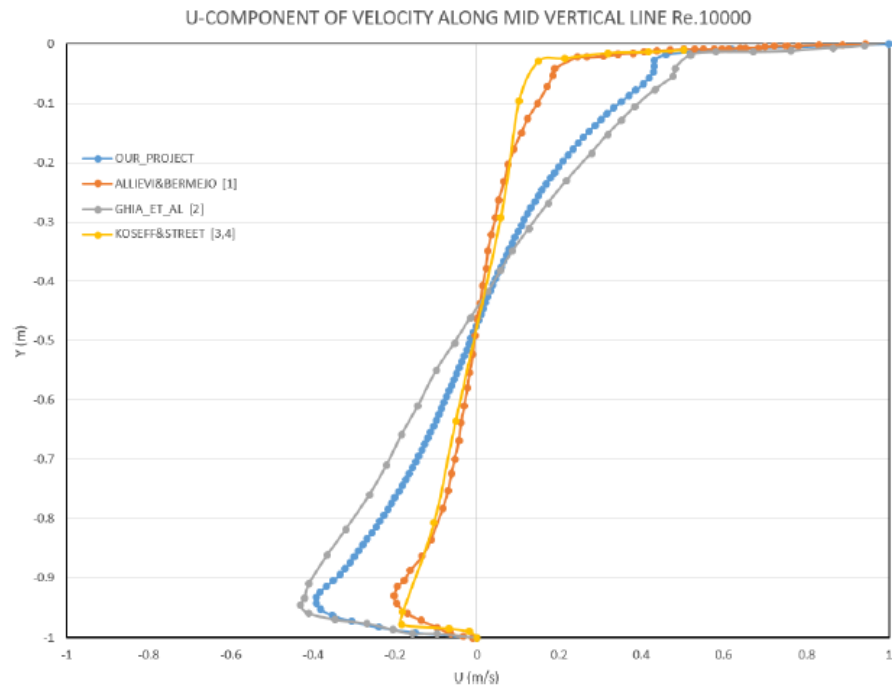*Figure 29: U and V velocity components for Re3200*

Figure 30: U and V velocity components for Re10000

c. COMPARISION OF RESULTS

My results are highly consistent when compared to the reference paper and Ghia et Al. Additionally, I think my velocity streamlines are a bit better. U and V velocities are similarly consistent. Only difference of the velocity plots is the locations of plots due to geometric setup reasons. In the reference study, I think they draw the geometry from (0, 0) to (1, -1).

4. Conclusion

In this project, I performed a lid driven cavity flow analysis in 2D using ANSYS Fluent. I have experimented the turbulent flow characteristics and also now I have a much better understanding in Reynolds Number and importance of mesh orientations.

5. References

[1] S. Ziaeiasl & C.E. Umeaka, Ing. (2022, March 24). Ansys fluent aided viscous flow modelling of lid driven cavity flow, 2D and 3D flow over a cylinder.
https://www.academia.edu/73697948/ANSYS_FLUENT_AIDED_VISCOUS_FLOW_MODELLING_OF_LID_DRIVEN_CAVITY_FLOW_2D_AND_3D_FLOW_OVER_A_CYLINDER

[2] U. Ghia, K. Ghia and C. Shin, "High-Re solutions for the incompressible flow using the Navier–Stokes equations and a multigrid method," vol. 48, p.387, 1982.

[3] https://www.linkedin.com/pulse/lid-driven-cavity-saba-golshaahi-sumesaraayi/