ME 526 - Homework 1

Simulation study of the swirl flow around a rotating disk

Due Date:

9th February 2023

(Group № 2)

Member №1: Gabriel Goh

Member №2: Amirreza Rezaeepazhand

Member №3: Naiwen Hu

Member №4: Yury Luzhkov

Introduction

This homework report illustrates the COMSOL simulation results. We simulate a fluid behavior located in a cylindrical container with an inner rotating shaft (at the center of the cylinder). During this homework a plot of a 2-D velocity profile of the slice part of the cylinder was produced. Then streamlines were added to the plot, and the same plots were produced for the three different angular velocities. Next step was the plotting of isocontours for the azimuthal velocity component for angular velocities (left to right) for $\omega = 0.25\pi$, 0.5π, 2π, and 4π rad/s. The final plot of simulation was the turbulent viscosity and with the streamlines of the velocity field, with an angular velocity set to 500 π. Finally the report includes answers to some conceptual questions that could possibly arise from this simulation.

Section 1: Plots

This section illustrates the figures obtained while the simulation steps in HW 1 manual were executed.

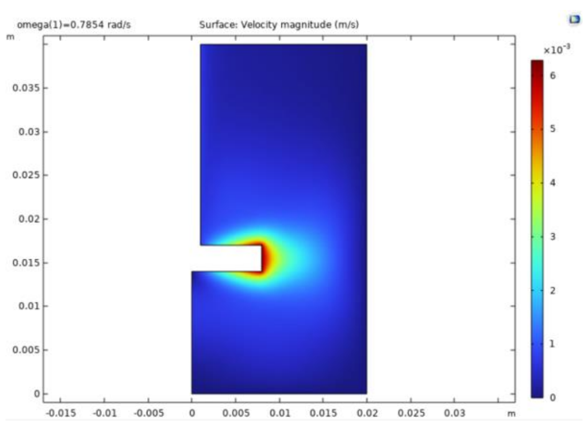

Figure 1: : Results for angular velocity ω= 0.25π rad/s. The surface plot shows the magnitude of the velocity field.

Figure 1 illustrates the 2-D velocity field inside a slice of cylinder. You could see how a circular platform at the end of the rotating shaft causes the fluid around it to rotate together with it. As you go closer to the center the fluid velocity decreases due to the velocity of surface becoming smaller.

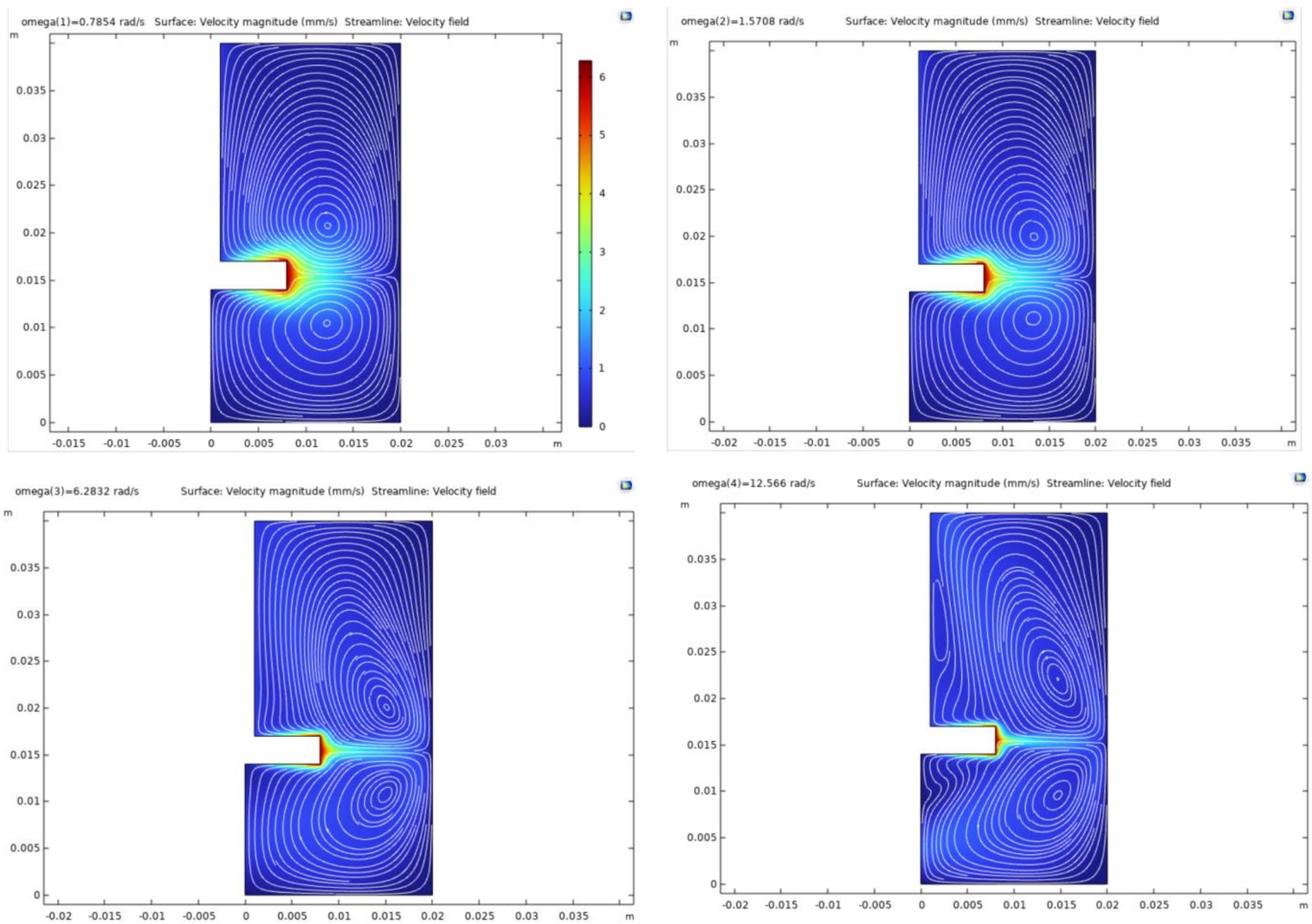

Figure 2-5: Results for angular velocities ω= 0.25π, 0.5π, 2π, and 4π rad/s. The surface plot shows the magnitude of the velocity and the white lines are streamlines of the velocity field.

Figures 2-5 illustrate the magnitude of velocity fields in the background. You could see how with increase in velocity of the shaft the fluid velocity layer around becomes relatively thinner, and the streamlines get skewed away from the shaft.

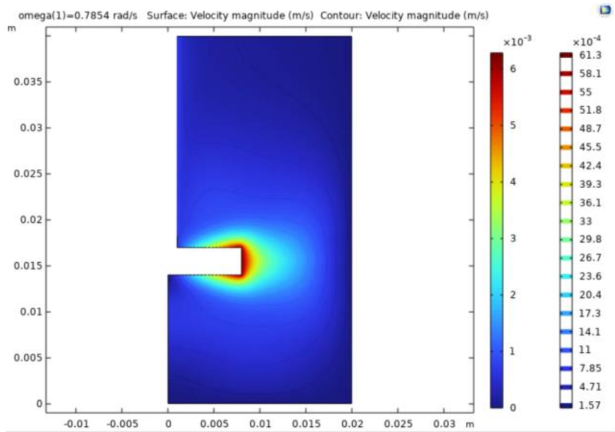

Figure 6: The surface plot shows the magnitude of the velocity.

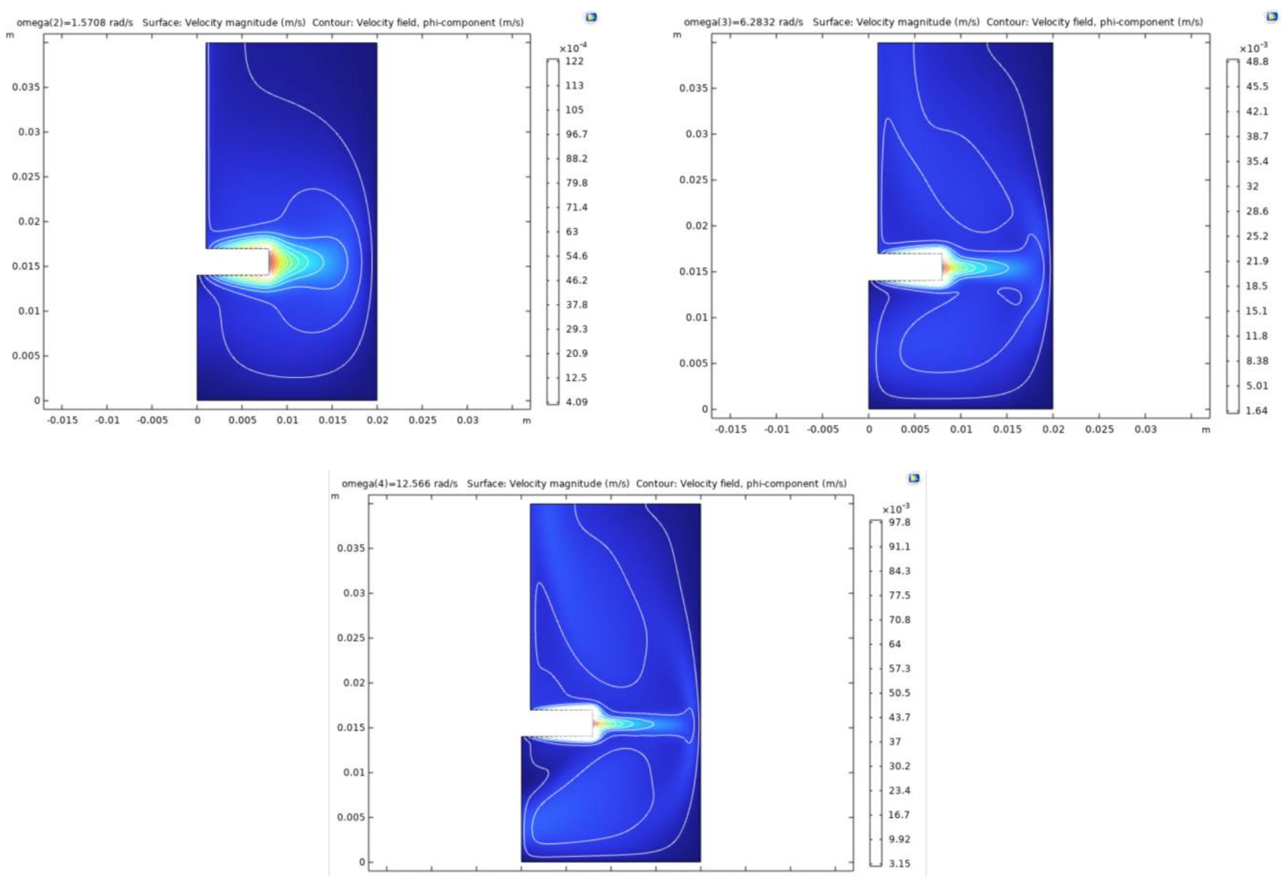

Figure 7-9: Magnitude of the velocity field (surface) and isocontours for the azimuthal velocity component for angular velocities (left to right) ω = 0.25π, 0.5π, 2π, and 4π rad/s.

Figures 7-9 illustrate the streamlines for the azimuthal velocity. You could see a radical change from *0.5π, 2π.* The steamline that was close to the axial shaft got blown away and formed an elliptic shape.

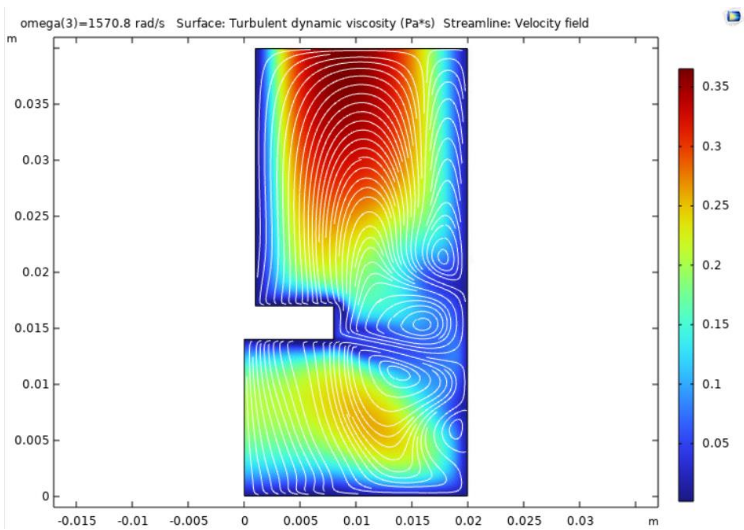

Figure 10: Results for angular velocity ω=500π rad/s. The surface plot shows the turbulent viscosity and the white lines are streamlines of the velocity field.

The colour background in figure 10 indicates that the viscosity is the lowest close to the shaft and close to the side of the container, and that the viscosity is the greatest aways and to the top of the shaft. The velocity field of fluid has contacted the no slip condition wall at this speed and has resulted in 2 new velocity streamlines circles.

Section 2: Answer questions

(a) Comment on the axial symmetry used and why this works, from a physics and mathematical point of view. Suppose this implementation did not have the “2D axisymmetric option” that COMSOL provides, but was simply a “2D simulation”, which is offered as an option in COMSOL. Thus, I want you to consider two cases, namely: (1)

your present case of simulating of a slice through the axis of the rotor, where the fluid is flowing in a cylindrical manner, using the “2D axisymmetric option” in COMSOL, versus (2) you use the 2D planar-like option in COMSOL, but where the height of the fluid in this 2D “slice” in COMSOL is the same as the height of the fluid in what you are presently running, and where the width of the 2D simulation of fluid is the same as the diameter of the cylindrical vessel you are treating. Would you get the same answers? (You do not need to rerun - just think about it and answer.) What is different? In the new 2D slice I am proposing, what would this simulation be physically treating? The same thing?

The axial symmetry is used because it reduces the complexity of the problem. This is because it assumes the flow field is symmetrical around the axial direction so from the physics point axial symmetry will work because of the assumption of cylindrical symmetry. Since the flow field is normally symmetrical around the axial direction in situations requiring cylindrical geometries - which means that the flow characteristics and behavior are the same at any given radial distance from the axis - the flow field may be represented in a single cross-sectional plane rather than a whole three-dimensional structure. This reduces the number of dimensions that must be solved. From a mathematical perspective, assuming the symmetry reduces the amount of formulas and calculations that need to be done, which works well for COMSOL.

If the axisymmetric option was not used and 2D planar simulation was run the results would be different because in the 2D planar simulation the height of the fluid would be the similar, but the width of the fluid will be smaller. This would result in a different flow field, as the flow dynamics would not be symmetrical around the axial direction, and the effects of the sidewalls would not be very accurate.

In the 2D planar simulation, the simulation would be physically treating a slice of the fluid instead of a whole cylindrical flow field which will cause more inaccurate results because the full flow field will not be captured. The axisymmetric simulation will provide more accurate representation of the flow field in a cylindrical vessel, because it measures the effects of the sidewalls and the axial symmetry of the flow.

(b) This is sort of obvious, but please comment on it anyway. Why didn't the COMSOL manual simply have you make a full 2D cross section, or, why not just have you do the full 3D structure? Suppose you did create the full 3D structure ... How would your numerical accuracy compare with the structure they had you make, for similar number finite elements in the structure?

Axisymmetric simulations are more effective because they simplify the simulation and need less processing power. It might not be essential to fully solve a full 2D cross-section or 3D structure in order accurately capture the key features of the flow field in a cylindrical vessel.

As long as the axial symmetry of the flow is preserved, the axisymmetric simulation would probably have a similar degree of accuracy to the 3D version as long as there is axial symmetry of flow. The axisymmetric simulation would not be as accurate as 3D construction if fewer finite elements were used because the 3D structure would be a more accurate description of the flow field.

(c) In the beginning description of this example, the statement is made: "However, the velocities in the angular direction differ from zero, so the model must include all three velocity components, even though the geometry is in 2D." What do they mean? Is the statement fully correct? Explain.

Yes, it is correct. The angular direction indicates the rotational motion of the system. Even though the geometry is in 2D, the rotational motion still needs to be considered as the velocity in that direction is non-zero. If we only consider two dimensions of this model (x and y direction), the rotational motion would be ignored and the model could not be properly described. We used the 2-D cross section approximation as the structure is symmetric for each value of the angle, which means that the angular velocity would be the same (non-zero) throughout the structure. Therefore, by indicating it once using the color,we can display them for every value of data.

(d) In this problem, the “no-slip” wall condition was imposed in two places. Comment on how realistic this is. Good approximation? When would it not be? At room temperature, does the “no-slip” condition ever hold? If not, why not? How would it affect your results if you did not impose this condition?

In fluid dynamics, the ‘no-slip’ condition refers to the assumption that the fluid will have zero velocity relative to the boundary. This means that the fluid is in perfect contact with the wall without any relative motion. The ‘non-slip’ condition is a good approximation at certain conditions, like high density and viscosity fluid. However, this may not be a good approximation at all times even when room temperature is applied. For example, in the cases of low viscosity fluids and gas, molecules are in motion relative to the wall due to thermal fluctuations, and the resultant velocity is non-zero.

Besides, it is also not a good approximation at very low pressure (e.g. high altitude), where there are still a few molecules near the surface bouncing down the surface. It is also not a good approximation at conditions such as very low temperature, high fluid velocity, or existence of a smooth surface.

If no-slip condition was not imposed, the fluid velocity would be non-zero at the interface, which would result in a false forecast of fluid behavior and pressure distribution.

(e) In this problem, the “sliding wall” condition is imposed where the rotor is in contact with the liquid. You can read a little bit about the sliding wall condition in the manual, CFDModuleUsersGuide.pdf (see brief discussions on pp. 106, 163, 193, 416, 456, 457), at our Google drive folder. What condition is assumed between the rotor and the fluid where they are in contact? Is this realistic? How about when the rotor first starts turning?

The “sliding wall” is a condition that the wall actually has not any movement, but it is sliding in a tangential direction. The "sliding wall" condition is assumed for the contact between the rotor and the fluid, which means it is assumed that the contact between fluid and rotor is a perfect sliding contact with no slip. Well, although it would not occur completely in real life, it is a realistic assumption for steady state conditions. However, in a transient condition, for example, the time that the rotor starts turning, the error of this assumption would be high.

Conclusion:

During this exercise we learned how to use the COMSOL interface. We got familiar with the general layout and the operation of the program. We are looking forward to learning more functions of this physics simulation tool. In terms of this simulation we are able to visualize the fluid behavior inside the 3-D system which could be hard to be accurately measured in experiments. It is surprising to learn that with the increasing shaft velocity, the relative velocity spread around the shaft drops more radically within the same distance.