Study of Taylor Couette flow

Jose Paul B

College Graduate

Department of Mechanical Engineering

Government Engineering College, Thrissur

Abstract

The objective of the present project is to study the Taylor Couette flow between two cylinders in open source CFD package openFOAM. The velocity profile between the cylinders and wall shear stress are obtained and compared with the analytical results. Separate case studies with laminar and with turbulence properties are studied and graphs are plotted.

1. Introduction

Taylor-Couette flow is a classical problem in fluid mechanics. The stability of couette flow was investigated by Sir Geoffrey Ingram Taylor in his paper [1] “Stability of a viscous liquid contained between two rotating cylinders”. The Taylor-Couette flow consists of a viscous fluid confined in the gap between two rotating cylinders. For low angular velocities, which is measured by Reynolds number Re, the flow is steady and azimuthal.

In this project, the fluid is assumed to be Newtonian and the outer cylinder as fixed. The variation of velocity with radial distance and wall shear stresses are to obtained from simulating the steady state problem.

2. Problem Statement

A 2D Taylor-Couette flow with inner radius 1m with angular velocity 6 rad/s and outer radius 2m is given. The dynamic viscosity is 0.1 Pa.s and density 1 Kg/m^3.A graph is to be plotted for azimuthal velocity for laminar and turbulent case .It is to be compared with analytical solution. The shear stress at walls of both cylinders is to be compared with analytical results.

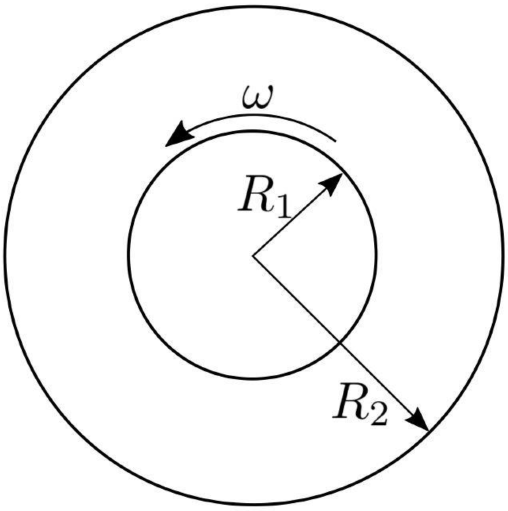

Figure 1: Schematic drawing of the Taylor Couette Flow.
(R_1 = 1 m, R_2 = 2 m and ω = 6 rad/s)

3. Governing Equations

The governing differential equations of fluid flow are as follows:

- Conservation of mass

$$\frac{\partial \rho}{\partial t} + \nabla.\left(\rho \vec{U}\right) = 0 \qquad 3.1.a$$

- Conservation of momentum

$$\frac{\partial \rho \vec{U}}{\partial t} + \nabla.\left(\rho\left(\vec{U} X \vec{U}\right)\right) = -\nabla p + \mu \nabla.\nabla\left(\vec{U}\right) + \rho \vec{a} \qquad 3.1.b$$

Where X is cross product as in vector product and vector a is acceleration.

4. Simulation Procedure

4.1 Analytical solution

There will be no axial motion in 2D Taylor Couette flow. This gives

$$\frac{1}{r}\frac{\partial}{\partial r}(r\mathrm{v_r}) + \frac{1}{r}\frac{\partial v_\theta}{\partial \theta} = 0$$

Due to circular symmetry azimuthal velocity varies only in radial direction. As radial velocity is zero at inner and outer wall, it is zero for whole case according to

$$\frac{1}{r}\frac{d}{dr}(rv_r) = 0$$

The theta momentum equation gives

$$\rho(U.\nabla)v_\theta + \frac{\rho v_r v_\theta}{r} = \frac{-1}{r}\frac{\partial p}{\partial \theta} + \rho g_\theta + \mu(\nabla^2 v_\theta - \frac{v_\theta}{r^2})$$

Only the last term exists in the present problem.

$$\nabla^2 v_\theta - \frac{v_\theta}{r^2} = 0$$

The solution to linear second order differential gives

$$v_\theta = c_1 r + \frac{c_2}{r}$$

The boundary conditions are

$$v_\theta = 0 \; at\ outer\ wall\ radius$$

$$v_\theta = \omega r_{inner} \; at\ inner\ wall\ radius$$

The final solution gives

$$v_\theta = \omega r_i\left(\frac{\frac{r_o}{r} - \frac{r}{r_o}}{\frac{r_o}{r_i} - \frac{r_i}{r_o}}\right)$$

Where r_i is inner wall radius and r_o is outer wall radius.

In this case azimuthal velocity is given by

$$v_\theta = -2r + \frac{8}{r}$$

4.2 Geometry and Mesh

The geometry was divided into 4 regions for meshing as the region had to be convex. The geometry was created in openFOAM itself. The grid generation was done using blockMesh tool. It was made using hexahedral blocks with simplegrading 1 in all direction and number of

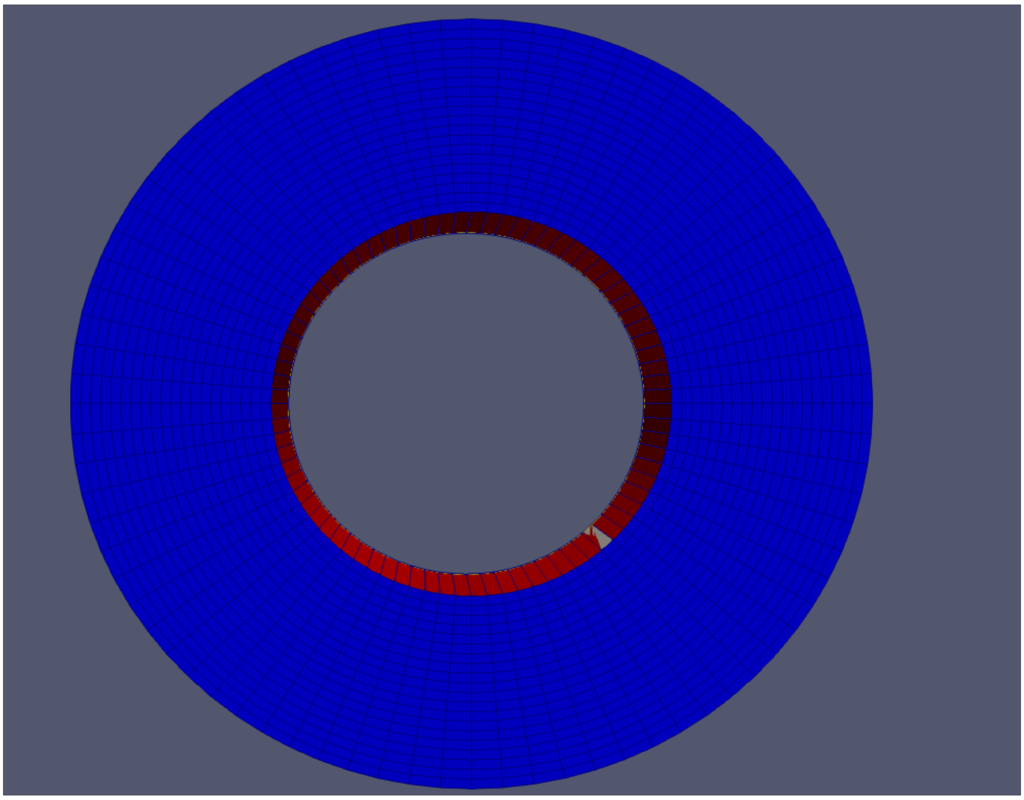

Figure 2:Geometry after grid generation

blocks in x, y and z direction as 20, 20 and 1.The z direction needed only one block as it is not used for calculation in 2D case. There are 1600 hexahedra cells in the mesh.

4.3 Initial and Boundary Conditions

The initial and boundary conditions are included in the 0 folder as p and U.

	Velocity U (m/s)	Pressure p (m^2/s^2)
rotatingWall	rotatingWallVelocity - omega 6	fixedValue - uniform 0
fixedWalls	noSlip	zeroGradient
Top	empty	empty
Bottom	empty	empty

Table 1:Boundary conditions

The internal fields were kept as uniform 0 for pressure and uniform 0 in all directions for velocity field.

4.4 Constant Properties of the system

The constant properties of the system defines various characteristics defining mesh information, transport properties and turbulence properties in each of their own folder in constant folder as follows:

• **polyMesh**

The polyMesh folder contains the mesh information which is created using blockMesh command.

• **transportProperties**

This file contains the information about the physical properties of the working fluid such as density and dynamic viscosity. These properties are provided at problem statement.

• **momentumTransport**

This file contains the information about the turbulence properties of the system. For laminar and turbulent case simulation type was kept laminar. For Turbulent case Reynolds' Averaged Simulation model was kept as k-ω SST with turbulence on.

4.5 Setting the runtime conditions and output control

The simulation control files, solution schemes, solution controls, probing files and graph plotting files are defined in the system folder.

• **controlDict**

The number of iterations were kept as 10000 with time step write control .To save storage purge write was kept as 1 and deltaT as set 1 as it is a steady case. Shear stress and Gradient of velocity was to be found by adding them in the functions. The probes function files were included to probe the gradient of velocity as sometimes the functions to find shear stress will give larger error at the rotating wall boundary patch. The singleGraph function file was added to find the velocity profile along radial direction of numerical results and compare with analytical results.

• **fvSchemes**

The finite volume scheme was kept steady state for ddt scheme, gradient and divergent schemes as gauss linear, lapacian scheme as gauss linear orthogonal with interpolation scheme as linear and surface normal gradient scheme as orthogonal. For turbulent case wall distance method was kept as meshWave.

• **fvSolution**

The pressure was to be solved by GAMG solver with GaussSeidel smoother. All other parameters were to be solved by smoothSolver with SymGausseidel smoother .SIMPLEC algorithm was used to solve the discretized equations.

• **probes**

Suitable component of gradient is set to be probed at inner wall and outer wall in order to compute shear stress.

• **singleGraph**

Component of velocity along positive y axis is to be stored by specifying start and end locations of line using this file.

4.6 Solver

The simulation is meant to be for incompressible steady state problem. SimpleFoam was used to implement this purpose. SIMPLEC algorithm was used as it gives relatively faster convergence over SIMPLE algorithm. Residual control criteria was set as 10^{-8} for all parameters and relaxation factor was kept to be 0.7 as default.

5. Results and Discussions

The solutions were converged by residual control criteria. Figure 3 represents variation of magnitude of U in the post-processing software paraview. The analytical results of velocity profile were compared with numerical results along the negative y axis using gnuplot. Negative y axis is taken as it gives positive values of azimuthal velocity which is aligned with the positive x axis of the coordinate system. The plots were compared for both laminar and turbulent case. Numerical results are almost coincident with analytical results. This is shown in figure 4 and 5.

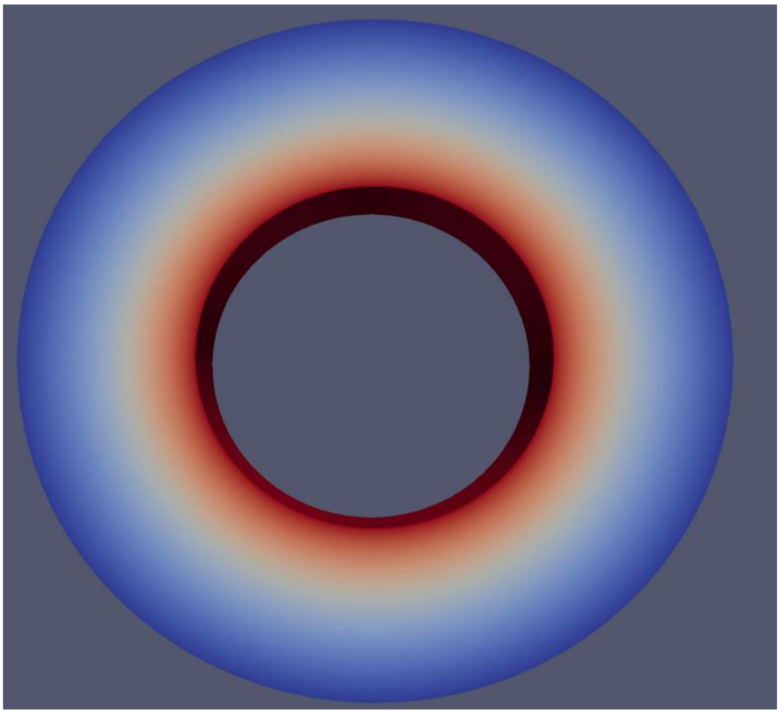

Figure 3:Variation of velocity magnitude within the geometry

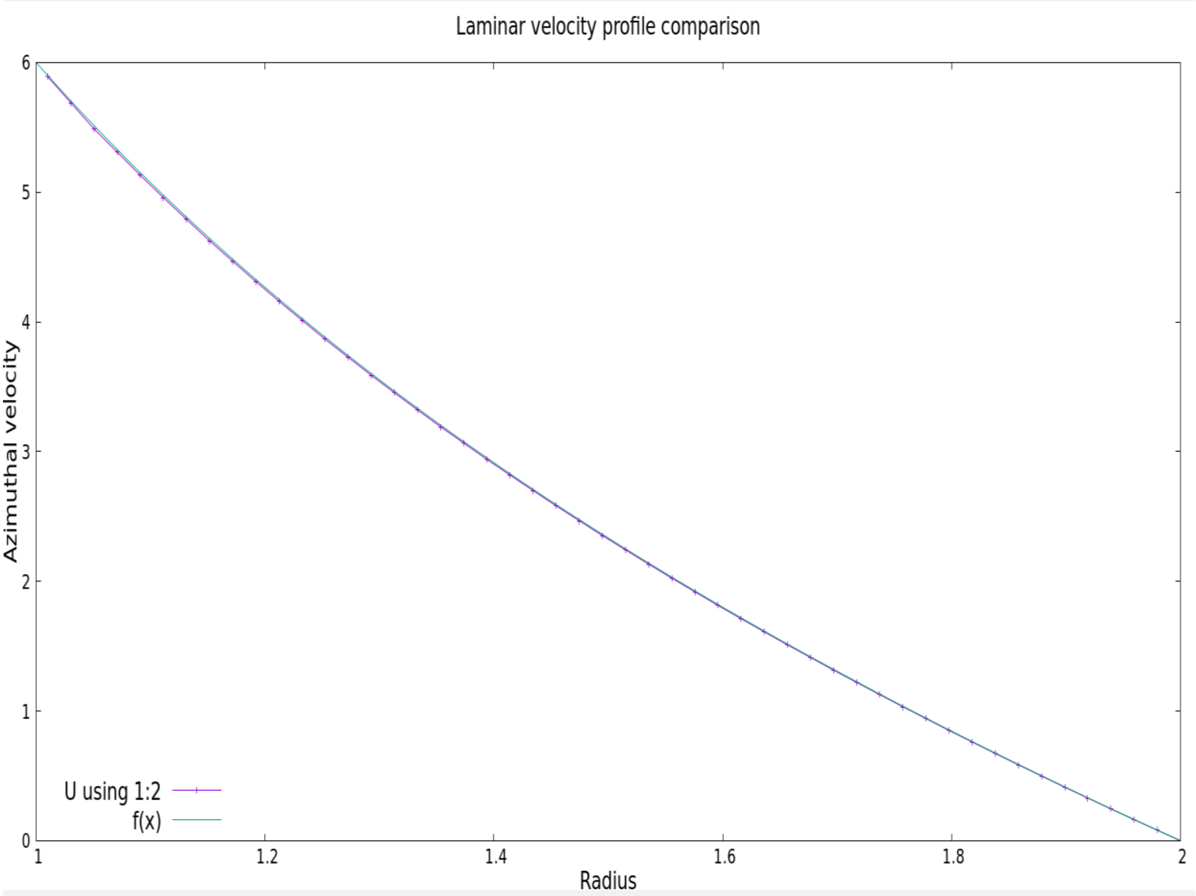

Figure 4: Velocity profile of analytical and numerical solutions in laminar case

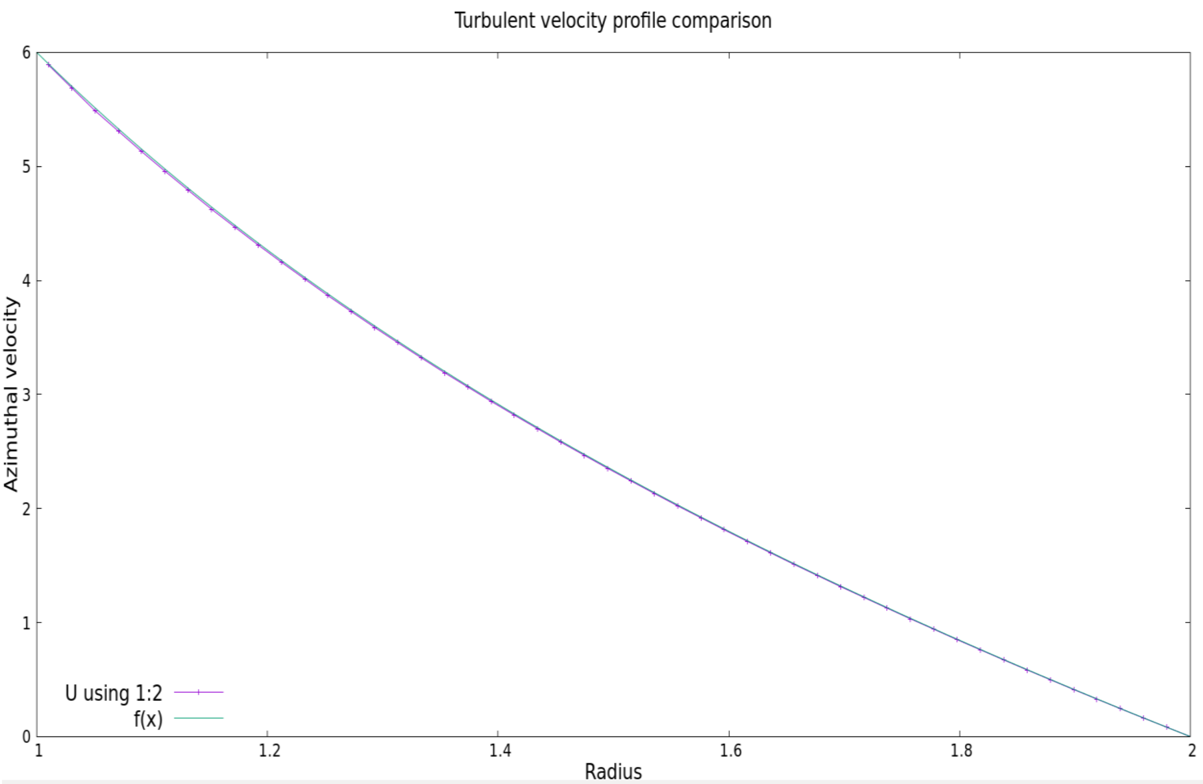

Figure 5: Velocity profile of analytical and numerical solutions in turbulent case

The shear stress found by paraview was prone to error at rotating wall boundary. So the values of velocity gradient were found from postProcessing folder in the same directory. These values obtained is used to compute the shear stress at wall boundaries. The following shows the values of shear stress computed by probing at the wall locations and value of dynamic viscosity.

	Laminar	Turbulent	Analytical
Shear stress (Rotating wall)	0.99084	0.99084	1
Shear stress (Fixed wall)	0.401266	0.401266	0.4

Table 2: Comparison of analytical and numerical results of wall shear stress

Conclusions

The numerical solution of wall shear stress is approximately equal to the analytical solution. The velocity profile is also almost coincidental with the analytical solution. The error in case of numerical results of rotating wall shear stress is 0.916 % and in case of fixed wall shear stress is 0.3165 %.

References

[1] Taylor, Geoffrey I., Stability of a Viscous Liquid Contained between Two Rotating Cylinders. VIII. *Philosophical Transactions of the Royal Society A*, (1923), 23, 289-343. (https://doi.org/10.1098/rsta.1923.0008)

[2]*CFD Direct User Guide*, 2021, Retrieved from https://cfd.direct/openfoam/user-guide/

[3]*CFD online forum*, 2021, Retrieved from www.cfd-online.com/Forums/openfoam/

[4]*FOSSEE spoken tutorial*, 2021, Retrieved from https://spoken-tutorial.org/tutorial-search/?search_foss=OpenFOAM&search_language=English

[5]*Wiki page on Taylor Couette Flow*, 22 December 2020, Retrieved from https://en.wikipedia.org/wiki/Taylor%E2%80%93Couette_flow