CFD analysis of lubricant flow in a Planetary Gearbox

Parth Tawarawala

Indian Institute of Technology Bombay

Abstract

The idea of the present study is to simulate the fluid flow of lubricant inside a planetary gearbox. OpenFOAM solver simpleFoam will be used for the same, since we can reasonably assume steady-state in a gearbox of a car in motion, and simpleFoam provides the best options for the same. The geometry is relatively complex, with a curved square pipe with rotating walls and a rotating cylinder placed within the flow. As part of an initial analysis, the geometry has been simplified to a 2D geometry. Since the geometry may not be exact, the emphasis will not be on exact flow parameter values but rather on visualisation of flow and look for any accumulation of fluid in the current complex geometry and recording it. This study is flexible because it can be expanded further to study the heat transfer within such geometries since the flow parameters and pattern will be well established.

1. Introduction

This study aims to study, visualise, and analyse the lubricant flow within a planetary gearbox, a device commonly used in industry. It consists of a rotating sun gear in the centre, 3-4 planet gears rotating and revolving around the sun gear. A fixed ring gear surrounds this assembly. All contacting surfaces have teeth to ensure that they are rolling with respect to each other. The current study will aim to find the maximum local velocities and their location for given physical properties and initial conditions. Since gears have complex geometry, they can be simplified to rolling cylinders with pitch diameters Instead of getting exact values, this analysis focuses on observing the general flow trends and plot velocity vectors and streamlines. Another important activity will be to study any undesired fluid accumulation. Accumulation and concentration of the lubricant, in the case of gearboxes, can be detrimental, as it can lead to corrosion, undesired temperature increases, and flow obstruction, leading to improper lubrication in the system.

2. Problem Statement

The goal is to do the following:

1. Plot the velocity vectors and streamlines to visualise and study the fluid flow
2. Plot the spatial variation of velocity, and find the location and value of the highest velocity.
3. Look for any accumulation in the gearbox flow pattern.

3. Governing Equations

Continuity equation(incompressible flow):

$$\boldsymbol{\nabla} . \boldsymbol{V} = \boldsymbol{0}$$

Momentum equation:

$$\frac{\boldsymbol{dV}}{\boldsymbol{dt}} + \boldsymbol{V}(\boldsymbol{\nabla} . \boldsymbol{V}) = -\boldsymbol{\nabla P} + \frac{\boldsymbol{\mu}}{\boldsymbol{\rho}}(\boldsymbol{\nabla}^2 \boldsymbol{V})$$

4. Simulation Procedure

4.1 Geometry and Mesh

The planetary gearbox has 3 components: a fixed ring gear, a rotating sun gear, and rotating and revolving planet gears. The sun gear has a diameter of 4 cm. The outer ring gear has a diameter of 16 cm. Accordingly, the planet gear is supposed to have a diameter of $\frac{16-4}{2} = 6$ cm. However, some space needs to be kept between the touching solids for fluid flow. Therefore the planet is kept with a diameter of 5 cm. This is a slight approximation, however, since we are looking at the nature of flow, this will suffice. The thickness of the domain will be equal to the thickness of a typical domain and is kept at 2 cm. The thickness of the planet gear is a little lower, at 1.8 cm.

Exploiting the symmetry of the problem, we can take one sector and apply a cyclic boundary condition at left and right patches to imply that the flow leaves from left and enters the right. However, in this case, we perform simulations in the frame of reference of planet gear's centre, keeping it at rest. This will significantly simplify the problem, from revolving planets, to stationary planters. Keeping the planets at rest will imply that the ring gear and sun gear will rotate in opposite sense, and the planet gear will rotate such that the contact velocities are equal. The sun and ring gears are modelled as rotating walls. The planet gear is inserted in between, with its cylindrical and circular face rotating in the same sense as the ring gear. The exact values of the velocities are shared in Section 4.2.

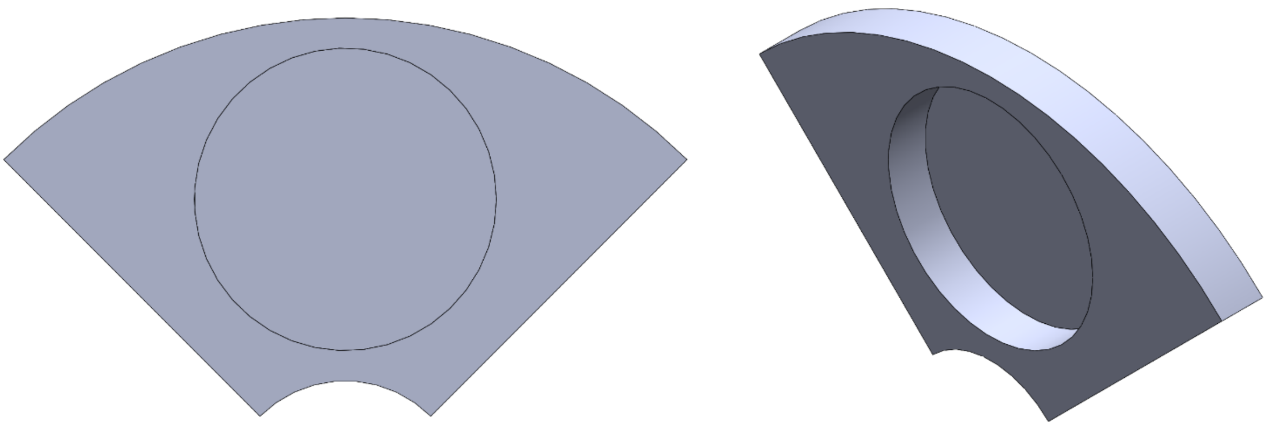

Fig 1: Isometric cross-sectional view of the CAD of the fluid domain

Initially, snappyHexMesh was tried to create a mesh, with the planet wall being rotated as well. However, the mesh requirements were complicated, and the method did not offer much flexibility. Since OpenFOAM allows mesh generated from other software to be inserted in it, Ansys Mesh was chosen as it offered much more flexibility and can also give a much refined and reasonable mesh. Finally, a fluid with lubricant properties is inserted, initially at rest, and the simulation is run till steady state is achieved.

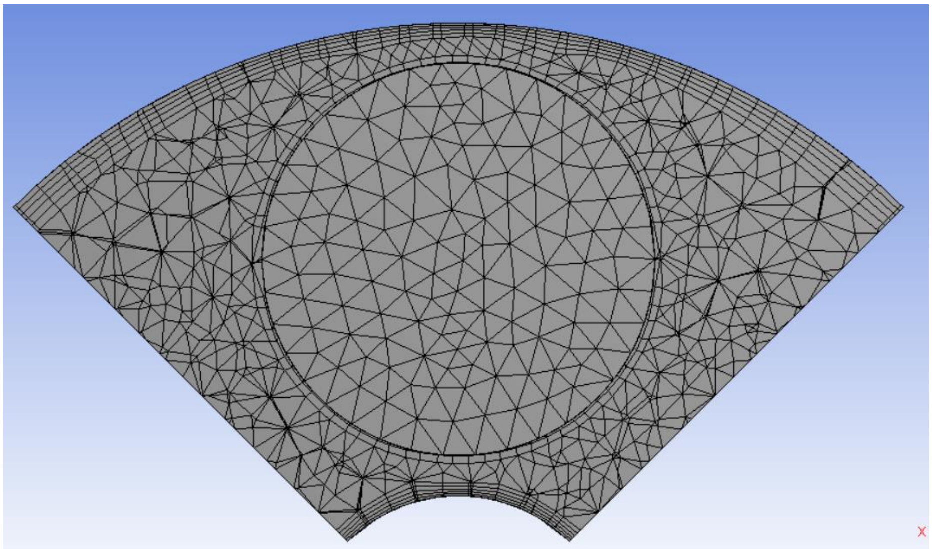

Fig 2: Cross-sectional view of the mesh of the initial 3D fluid domain

However, after running the simulation, there were convergence issues. The solution was diverging, and there was no explanation. Therefore, it was decided to shift to a 2D case, with a much simpler geometry, and conduct ana analysis on it. Once we have a basic idea of the results, we could shift to a 3D case for further complicated cases.

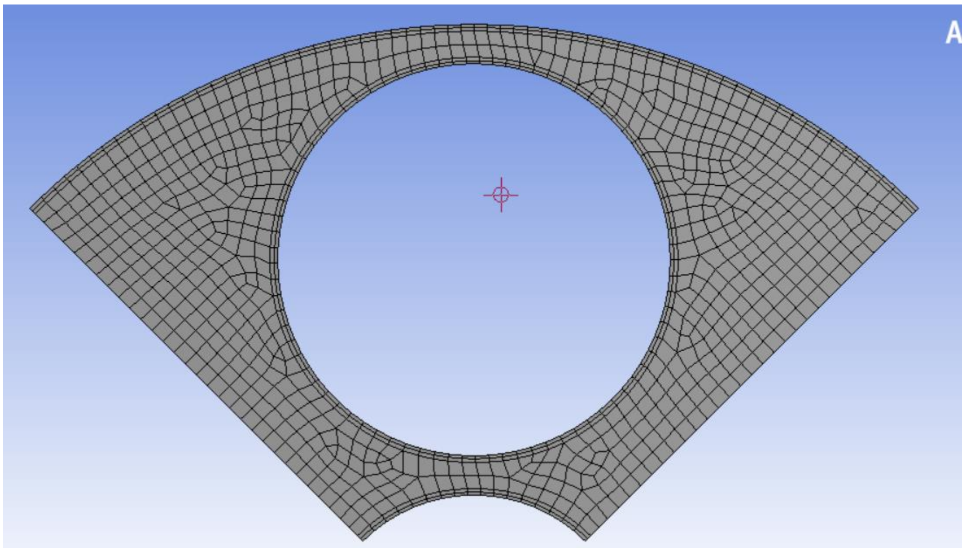

Fig 3: Cross-sectional view of the mesh of the final 2D fluid domain

4.2 Initial and Boundary Conditions

Initial Condition:

The following are the initial conditions:

a. Velocity: $\vec{\boldsymbol{v}} = \mathbf{0}$
b. Pressure: $\boldsymbol{P} = \mathbf{0}$

Boundary conditions:

a. Front and Back walls: Since this is a 2D case, these are fixed walls with empty boundary condition for Velocity. For pressure, the boundary condition is zeroGradient.

b. The top wall is the ring gear. It is a moving wall, with $\omega = 1250\, rpm = 130.899\, rad/s$. The outer layer of velocity will have noSlip with respect to the wall. The flow being incompressible, the pressure can be approximated to have zeroGradient.

c. The bottom wall is the sun gear. It is also a rotating wall, with $\omega = 5000\, rpm = 523.599\, rad/s$, in a direction opposite to that of the ring gear. Again, the outer layer of velocity will have a noSlip boundary condition, and the pressure will be zeroGradient.

d. The planet gear has 1 cylindrical surface. It will have a rotating wall boundary condition: $\omega = 4000\, rpm = 418.879\, rad/s$, in the same sense as that of the ring gear. Velocities will have noSlip on this, and pressures will be zeroGradient.

e. The left and right walls are dubbed inlet and outlet, and both of them have a cyclicAMI boundary condition. This is done to signify our assumptions of symmetry, and that fluid leaving from the right side will be identical to the fluid entering the left side.

These angular velocities are very particular, obtained from the fact that these are mutually rolling, and the contact point will have equal velocities.

4.3 Solver

Initially, icoFoam solver was chosen, as it gave a satisfactory steady-state, laminar solution to a preliminary problem, where there was no planet gear. However, with the current geometry, the Courant number would go on above 1. Hence, for this reason, pimpleFoam solver was chosen, as it is stable for $Co > 1$. Moreover, pimpleFoam allows an adjustable time step to be applied. Given the complex mesh being used, this feature was used to get better results. The time duration, max courant no., and deltaT were set accordingly. However, with pimpleFoam, the solution would often converge. Moreover, we can reasonably assume the fluid domain to be at steady state, for a car moving on a straight track at constant velocity. Therefore, for a steady-state, laminar solution, simpleFoam was chosen, as it is easy to use and offers all the features necessary.

5. Results and Discussions

The spatial variation of velocity of the fluid in the 2D fluid domain was plotted, and the observations are as follows:

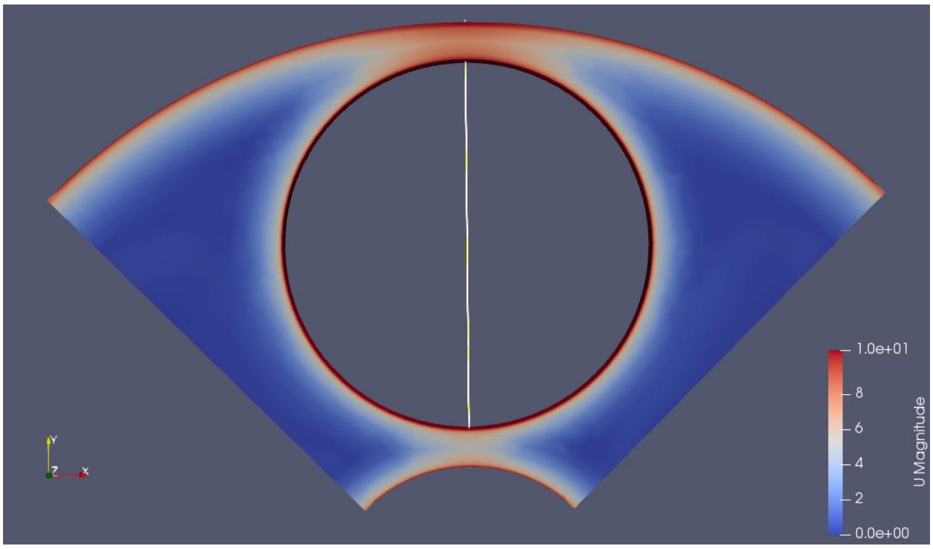

Fig 4: Spatial variation of velocity of fluid in the domain.

As evident from figure, the velocity is maximum near the rotating walls. The fluid had not been imparted any initial velocity, and any motion that arises, is due to the rotating walls. Thus, as expected, velocity is maximum near the walls, and goes down to almost 0, where there is no impact of the walls. The value of this maximum velocity is around 10 m/s, which is equal to the velocity of the walls, and can be obtained from calculations as below:

$$\omega_{ring} = 130.899\ \frac{rad}{s};\ R_{ring} = 0.08\ m$$

$$\Rightarrow v_{wall} = \omega_{ring} \times R_{ring} = 130.899 \times 0.08 \approx 10.47\ m/s$$

We can take any diameter, and observe the variation of velocity along it, to observe this value. In this case, the variation along y-axis is considered, as that is also where the planet gear is located, and we will observe the maximum gradient in velocity in this line.

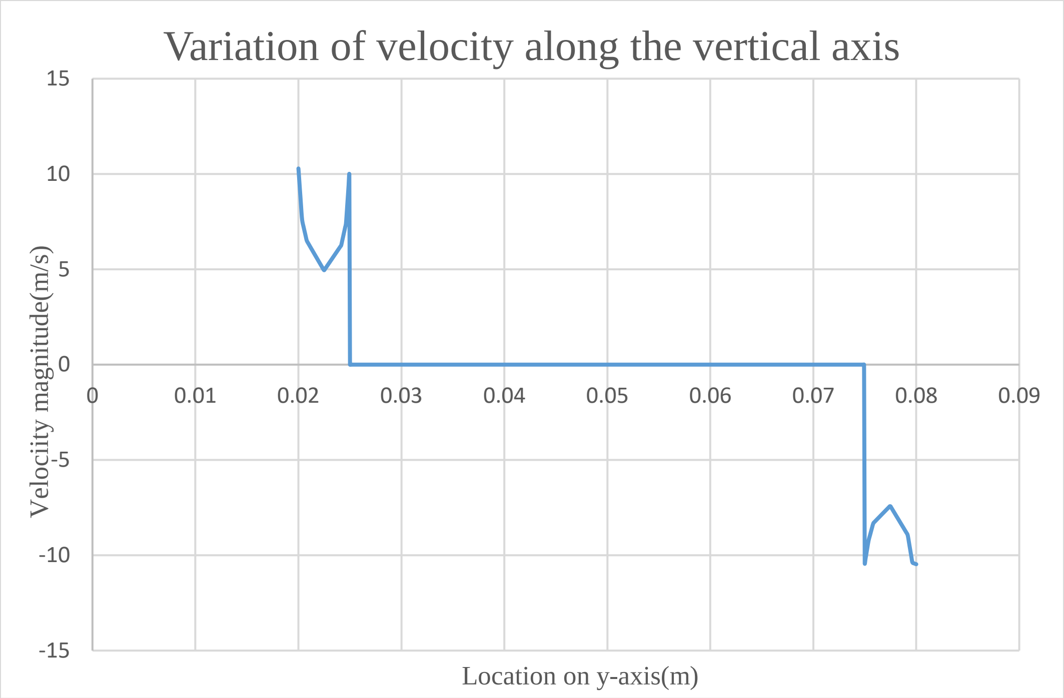

Fig 5: Variation of velocity along the vertical axis
(The discontinuity is due to the abrupt ending of the domain)

The velocity profile, looks fairly parabolic, with max magnitudes reaching around 10.46 m/s, which is very consistent with our calculations. Velocity variation along any other diameter will be similar in nature to the one shown here, with the only difference being in numeric values.

Now, we also need to look for accumulation of fluid, and for this, we need to plot the velocity streamlines, which are as follows.

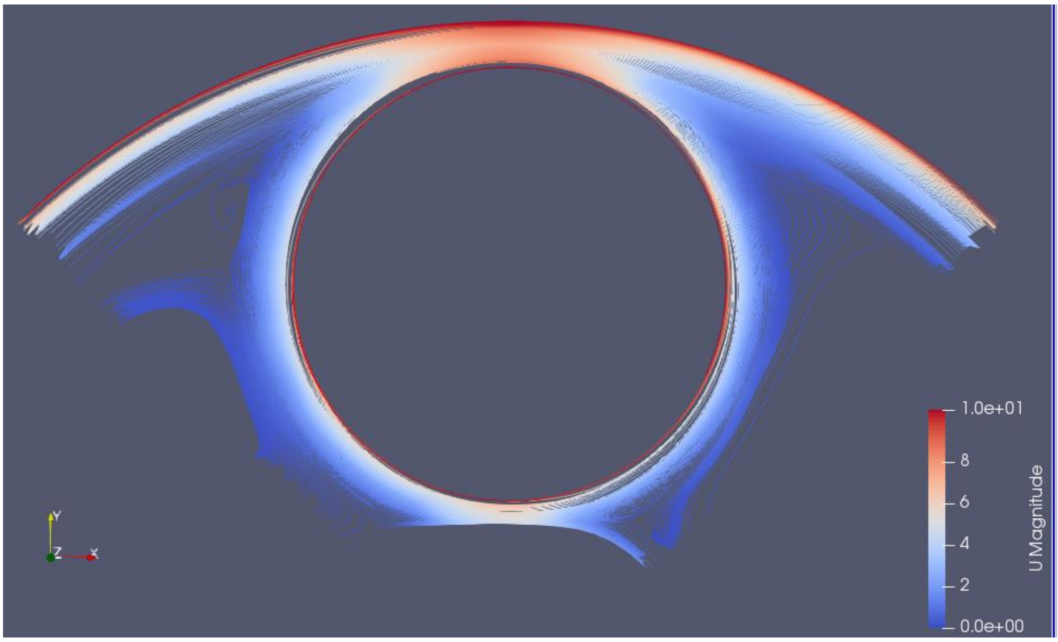

Fig 6: Streamlines plot(taken about a line inclined to the x-axis)

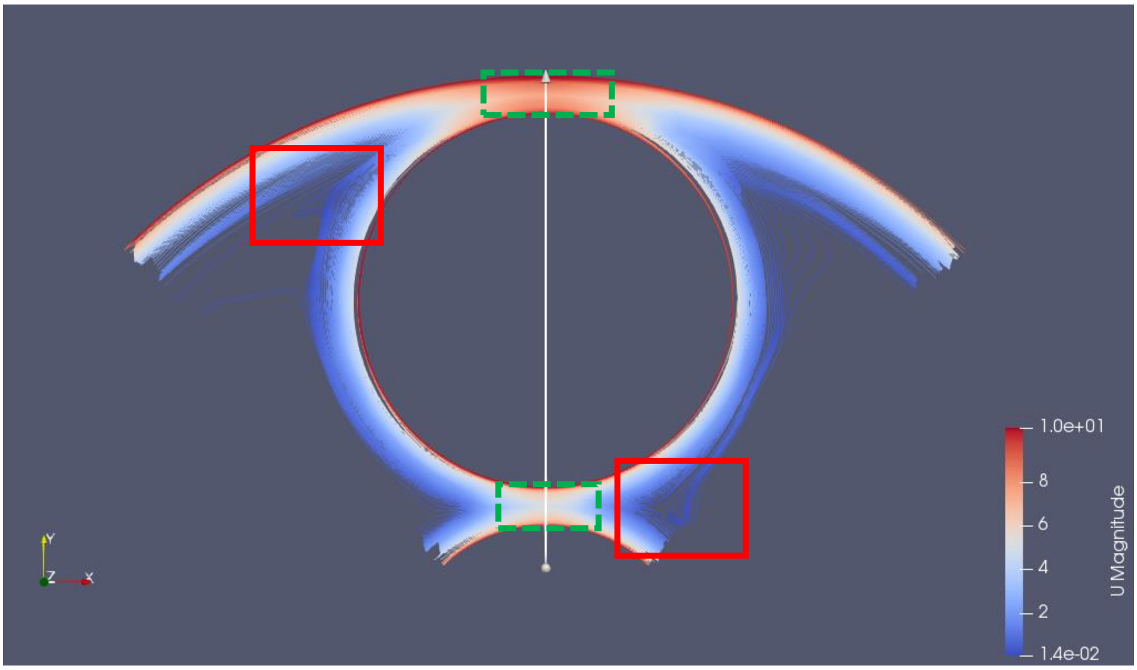

Fig 7: Streamlines plot(taken about the vertical y-axis)

As is evident from the 2 streamlines plot, there is almost no accumulation visible in the right side of the top part of the domain and left side of the bottom. However, there seems to be vortex generation on the other side of each part(highlighted by red boxes), and possible flow reversal. This can be explained, from the fact that the upper and lower parts, where the thickness of the domain is the lowest, act as pipe-like domains(shown as control volumes in green boxes), with rotating wall velocities being equivalent to the velocity of the fluid in a pipe. In such a case, there would be a low pressure region at the outlet of the pipe, and high pressure at the inlet. Near the bottom, the fluid will enter from the left, and get accelerated by the gears, witnessing an increase in the velocity. This, according to Bernoulli's principle, will result in a lowering of the pressure on the right side. A low pressure region can induce vortex generation in the flow. This can also cause possible accumulation. Similar argument can be made for the top CV. This low pressure regions, must thus be avoided, in order to obtain an uninterrupted flow.

References

1. https://www.researchgate.net/publication/279961807_Lubricant_Flow_and_Temperature_Prediction_in_a_Planetary_Gearset
2. https://journals.sagepub.com/doi/10.1177/1350650115622363