FOSSEE, IIT Bombay
OpenFOAM Case Study Project
June 26, 2020

CFD Simulation of Tesla Valve

Vignesh.S.P

M.E. Energy Engineering

College of Engineering Guindy, Anna University

Abstract

A Tesla valve is a one way valve with no moving parts. The objective of the present project is to simulate the behavior of Tesla Valve in forward and reverse flow conditions for different flow rates. The flow is a two dimensional, laminar, steady flow, so simpleFoam solver is used for the simulation. The aim is to observe the change in diodicity of the tesla valve for increasing flow rates.

1. Introduction

A tesla valve is a passive device similar to a gate valve in purpose but with no moving parts. The performance of a tesla valve is determined by the parameter Diodicity *D* which is the ratio between pressure drop in reverse flow Δp_r and pressure drop in forward flow Δp_f for the same flow rate.

$$D = \frac{\Delta p_r}{\Delta p_f}$$

2. Problem Statement

A T45A tesla valve is considered for the simulation, with flow rates 500 μl/min, 750 μl/min, 1000 μl/min, 1250 μl/min and 1500 μl/min. The flow is steady and laminar so simpleFoam solver is being used.

3. Governing Equations

The computational model solves system of Navier-Stokes equations for incompressible fluid flow (implemented in OpenFOAM) is as follows

Continuity Equation: $$\nabla . U = 0$$

Momentum Equation: $$(U . \nabla) U + \frac{1}{\rho} \nabla . P = \nu \nabla^2 U + g$$

Where U is the velocity vector, ϱ is the density, P is the pressure, **ν** is the kinematic viscosity and g is the gravitational field

4. Simulation Procedure

4.1 Geometry

The general geometry of the tesla valve is given in Figure 1. The design parameters which are used are as in table 1.

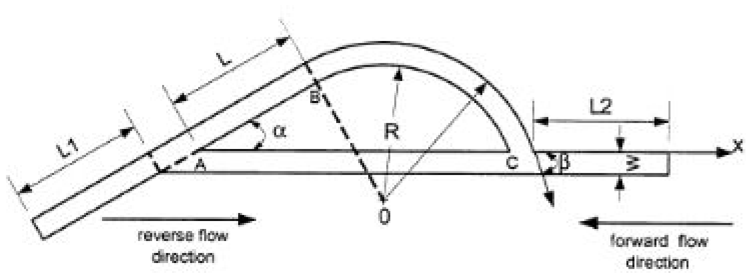

Figure 1 Geometry of Tesla valve [1]

Where, W is the channel width, R is the inner curve radius, L1 and L2 are entry and exit length respectively.

Dimensional Parameter	Value	Unit
L1, L2	600	µm
W	100	µm
R	228	µm
L	235	µm
α	45^{O}	-

Table 1 Design parameters

4.2 Mesh

Modelling and meshing is done using Salome-Meca, A Hexahedral structured mesh is generated with one axis having a single cell to represent 2D. The mesh can be seen in Figure 2.

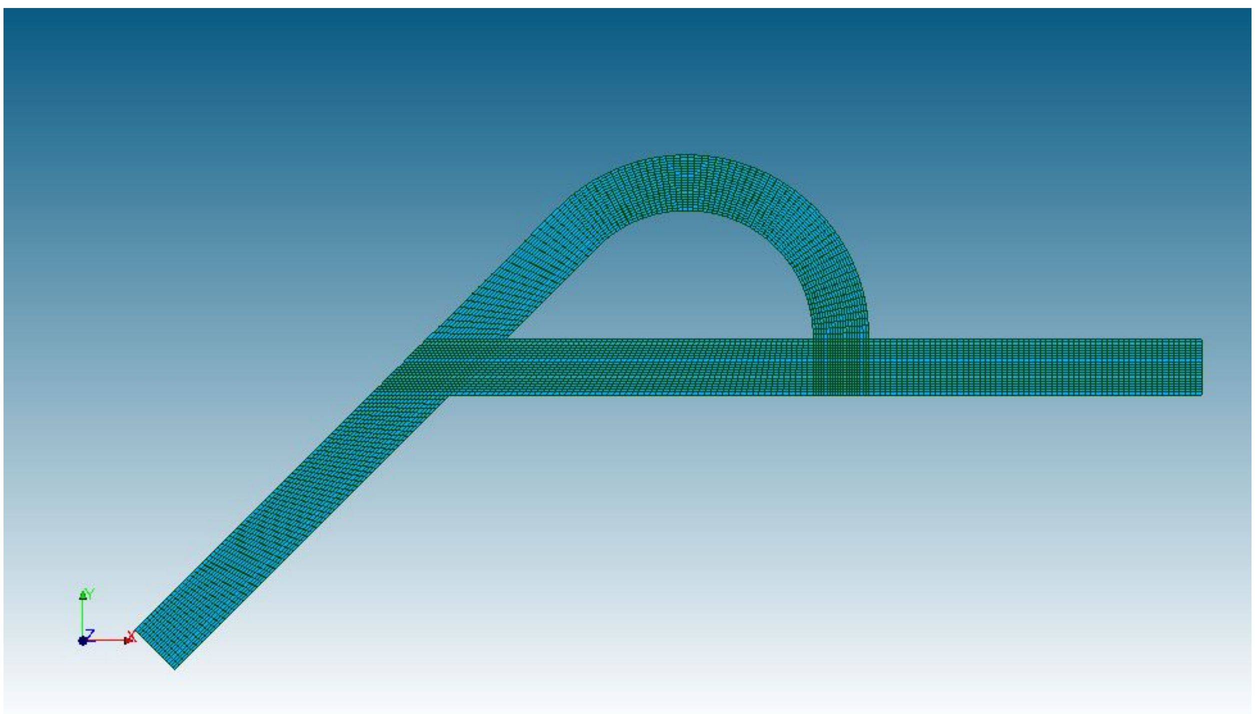

Figure 2 Mesh

4.3 Initial and Boundary Condition

The Boundary conditions for reverse flow used for the simulation are as given in the tabel 2, The boundary condition of inlet and outlet patch is swapped for forward flow.

Boundary Patch	U	p
inlet	flowRateInletVelocity	zeroGradient
outlet	zeroGradient	fixedValue
walls	noSlip	zeroGradient
frontAndBack	empty	empty

Tabel 2 Boundary conditions for reverse flow condition

The flowrate at the inlet patch is given as 8.33e-7 kg/s, 12.5e-7 kg/s, 16.66e-7 kg/s, 20.83 kg/s, 25e-7 kg/s.

4.4 Solver

The solver used is **simpleFoam**, a steady-state solver for incompressible, turbulent flows. The simulation is continued until convergence and using the **pressureDifferencePatch** tool, the difference in pressure is calculated after convergence.

5 Results and Discussions

In figure 3, the pressure contour for a flow rate of 500 µl/min is given, the top is reverse flow and the bottom is the forward flow. Since the flow rate is low, there is not much difference in pressure between each flow condition. The didocity is calculated and found out to be 1.11257 which is very near to 1 so that under this flow rate, reverse flow has very little increase in pressure.

In figure 4, the pressure seems for a flow rate of 1250 µl/min, calculating didocity yields 1.36134. From the didocity value, in reverse flow condition there is 36% increase in pressure with respect to forward flow condition. This increase in pressure is due to the mixing of two separated flows in reverse flow condition.

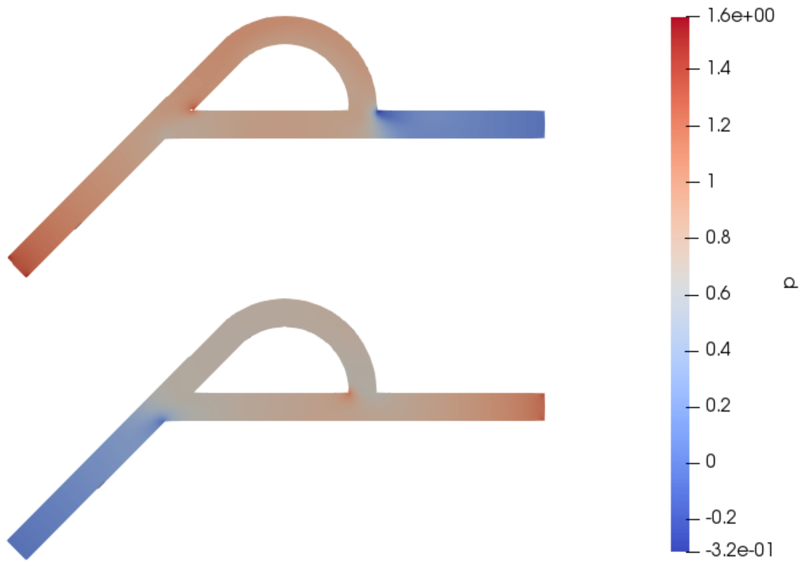

Figure 3 Pressure Contour for flow rate 500 µl/min, top(reverse), bottom(forward)

In forward flow the flow is not separated by the curved profile so the flow is undisturbed, but in reverse flow, the flow get separated at point A as in Figure 1 and the separated flow rejoins the main flow at point C as in Figure 1, due to this mixing the flow is retarded and due to which the pressure increases before the point C.

In figure 5, the plot flow rate against didocity gives the trend of increase in didocity valve with increase in flow rate. As the flow rate increases the pressure which retards the flow increases.

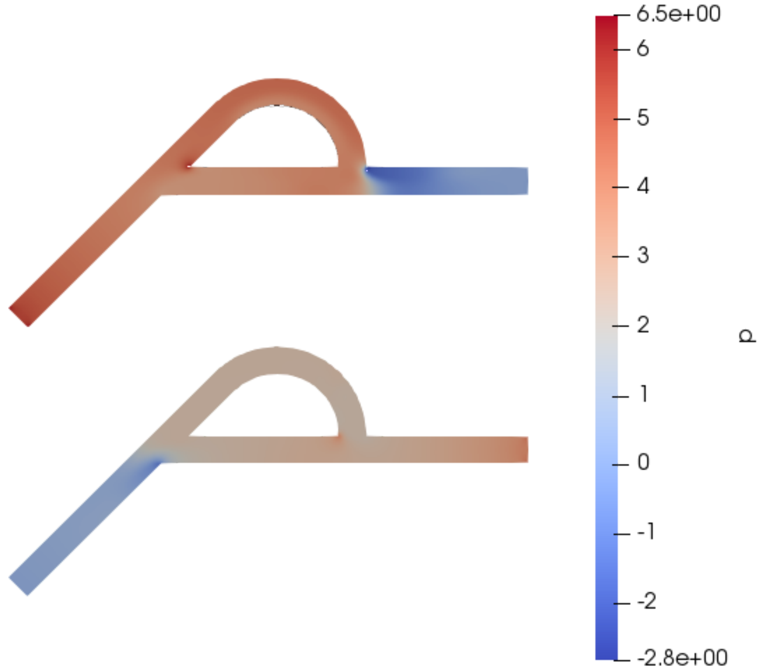

Figure 4 Pressure Contour for flow rate 1250 μl/min, top(reverse), bottom(forward)

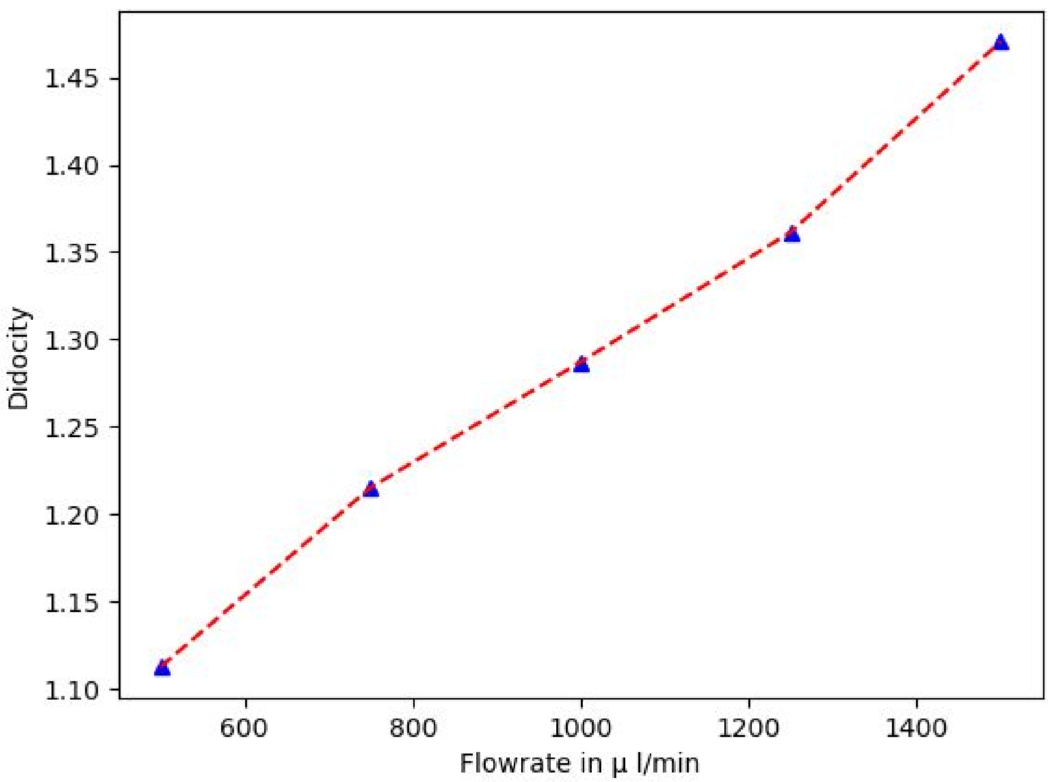

Figure 5 Flow Rate vs Didocity

6. Conclusion

This case study results provides a better understanding of the behaviour of tesla valves under different flow rates. The increasing trend of didocity of tesla valves for increasing flow rate, as found in [1], was verified in this OpenFOAM study.

REFERENCES

1. Truong, T.Q. and Nguyen, N.T., 2003. Simulation and optimization of tesla valves. Nanotech,1, pp.178-181.
2. OpenFOAM v1906 User Guide