FOSSEE, IIT Bombay

OpenFOAM Research Migration Project

April 20, 2022

STUDY OF PRESSURE DROP IN MICROCHANNEL VARYING TEMPERATURE WITH CONSTANT BIFURCATION OF 90°

Mahima Bunker

MTech, IIT Bombay

Synopsis

This research migration project aims to study and simulate water flow with the constant bifurcation of 90° in symmetric three-dimensional microchannel using OpenFOAM-9. The geometry and mesh were defined using blockMesh utility. A steady-state, SIMPLE algorithm-based simpleFoam solver was used in the simulation. The analysis executed by Samy et. al. [1] using commercial CFD code Fluent was taken as a reference.

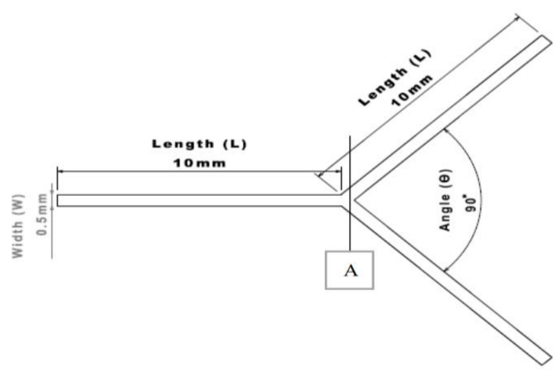

Figure 1: Geometry and Dimensions

The dimensions of the geometry stated in figure 1 are: Length(L) = 10mm, Width(W)= 0.5mm and Bifurcation angle= 90◦. Flowing fluid is entering from inlet with velocity of 0.251m/s and exiting from outlet.

References

https://soar.wichita.edu/bitstream/handle/10057/14480/t17020_Mudiki.pdf?isAllowed=y&sequence=215

CONTENTS

1. Introduction

The scaling down of circuits and to increase their performance, is leading to excess heat generation and affecting longevity and reliability over time. It Need to microscale and compact heat exchanger (high surface area to volume ratio). The research [1] is done on various methods and designs with different coolants fluid that is Water and Air. Our work is determining the variation of pressure drop with temperature on a surface of a microchannel with a 90° bifurcation angle. The fluid used in this study is water.

2. Governing Equations

Continuity Equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$$

For the standard cartesian coordinate system, the velocity component in x, y, and z-direction are represented by the notations u, v, and w respectively.

Momentum Equation:

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z} = -\frac{\partial p}{\partial x} + \frac{1}{Re}\left[\frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z}\right]$$

$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} + \frac{\partial(\rho vw)}{\partial z} = -\frac{\partial p}{\partial y} + \frac{1}{Re}\left[\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z}\right]$$

$$\frac{\partial(\rho w)}{\partial t} + \frac{\partial(\rho uw)}{\partial x} + \frac{\partial(\rho vw)}{\partial y} + \frac{\partial(\rho w^2)}{\partial z} = -\frac{\partial p}{\partial z} + \frac{1}{Re}\left[\frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z}\right]$$

The equations above represent the x, y, and z-momentum respectively and together with continuity equation are known as Navier-stokes equations.

3. Methodology

3.1. Geometry and Mesh

The current study is focused on studying the effects of temperature on pressure drop in micro-channels for Newtonian fluid - water. A 3-dimensional geometry has been considered in this study. The microchannel geometry as presented in fig 1 is of aspect ratio 1 and hydraulic diameter 500 microns investigated for a fully developed laminar flow bifurcated after the flow reached its hydrodynamic developed length with bifurcation angle of 90◦.

Using blockMesh utility, Geometry and mesh are defined with six blocks. The total number of cells is 410000. The meshing of the geometry is shown in fig. 2.

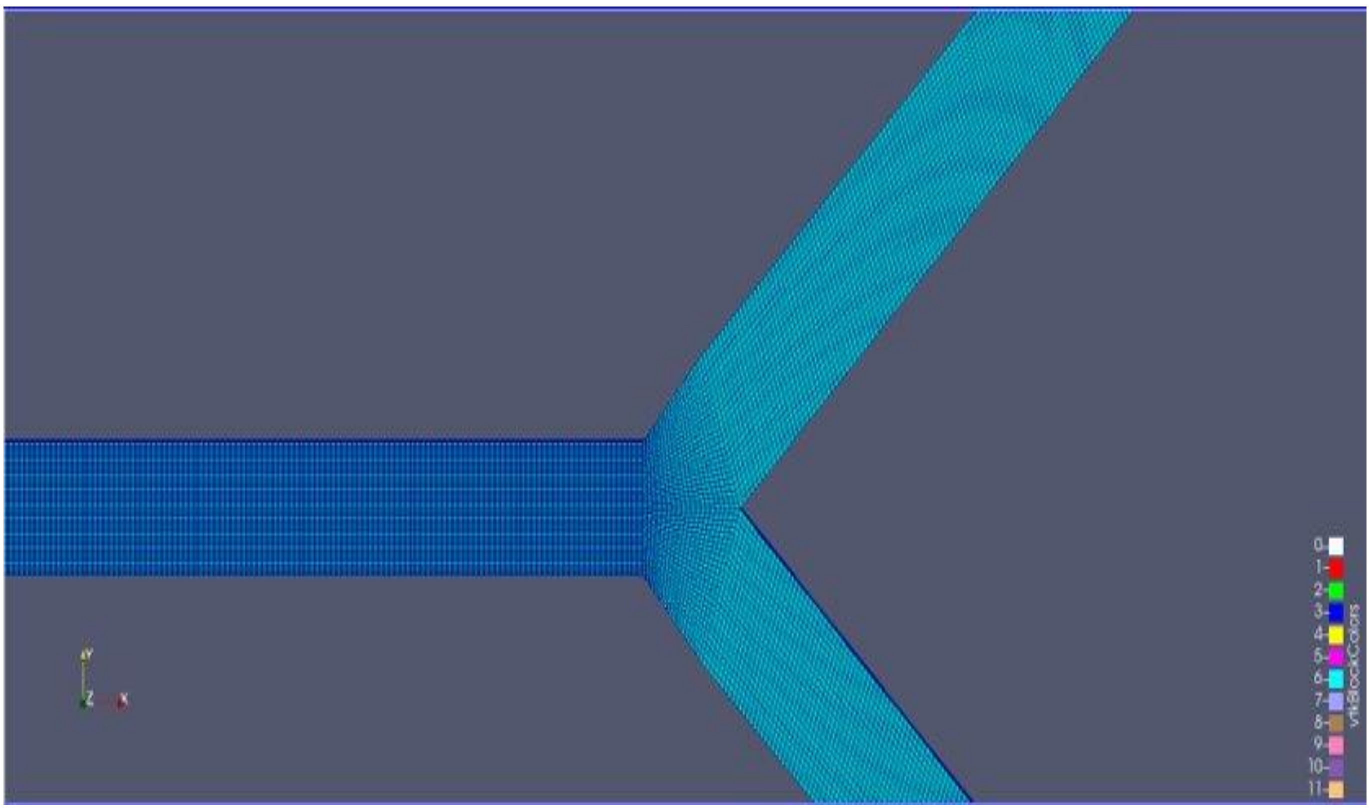

Figure 2: Computational Grid(Stretched View)

3.2 Initial and Boundary Conditions

The boundary conditions are set such that, for a given aspect ratio and fluid, the inlet Reynolds number is the same and stays in the laminar regime. The inlet velocity for water is set to 0.251 m/s for aspect ratio 1. The outlet condition is set as constant pressure condition.

The walls are assigned a no-slip boundary equation and the top wall is assigned a constant temperature boundary condition depending on each analysis.

3.3. Solver

SimpleFoam solver is used as it is applied for a steady-state incompressible to run the governing equations. It uses SIMPLE) algorithm to evaluate Navier Stroke equations. The SIMPLE algorithm is a numerical procedure to solve the Navier-Stroke equations. Simple is an acronym of (Semi-Implicit Method for Pressure Linked Equations).

The solution is converging at the 2000 iterations with the convergence criteria of 0.001.

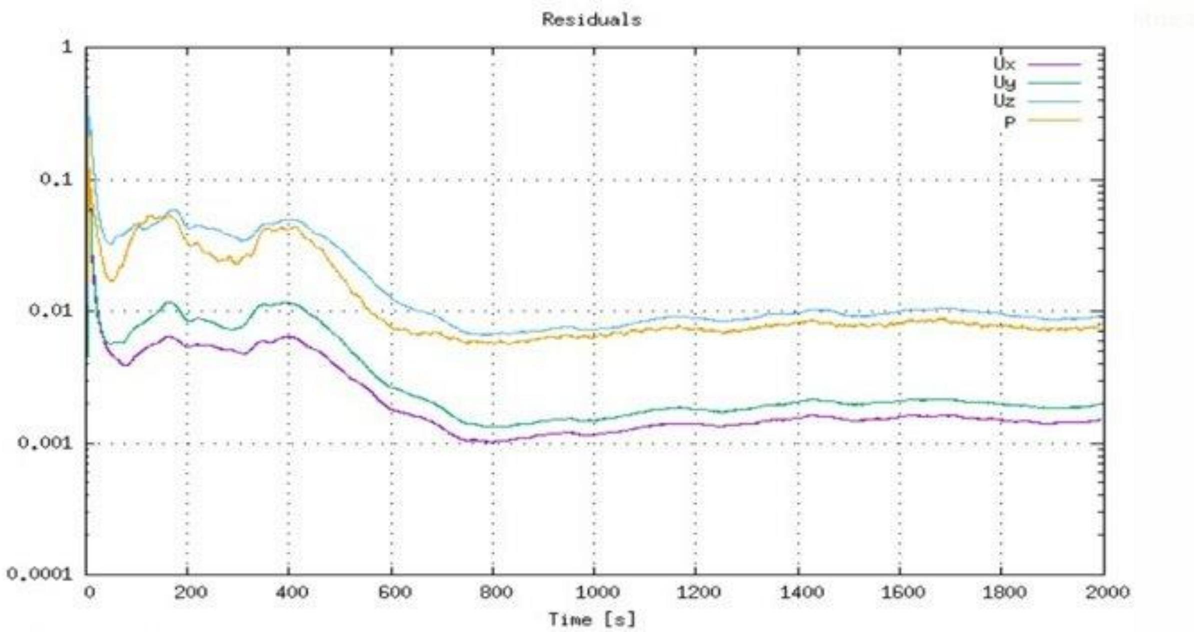

Figure 3: Residual Convergence

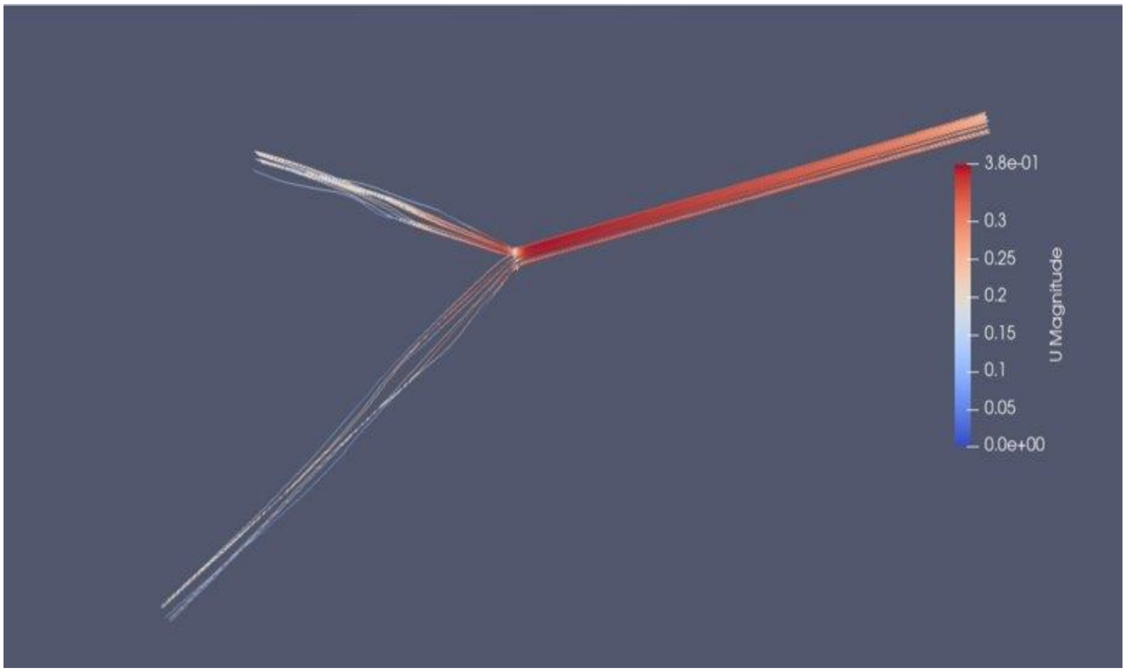

Figure 4: Streamline tracing

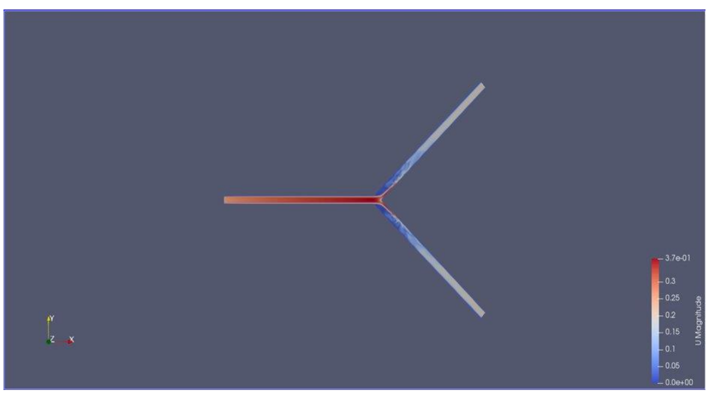

Figure 5: Velocity Contour

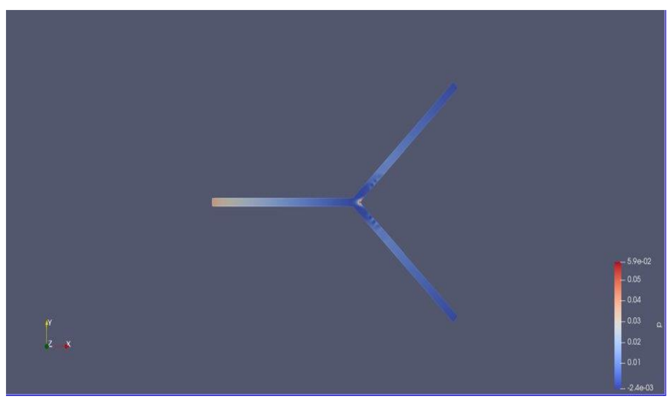

Figure 6: Pressure Contour

4. Validation

The current study is validated with [1] and the following results have been shown below:

1. Velocity Contours:

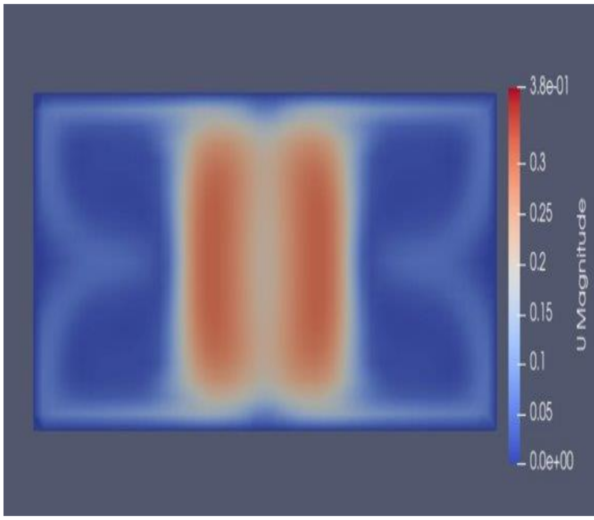

Figure 7

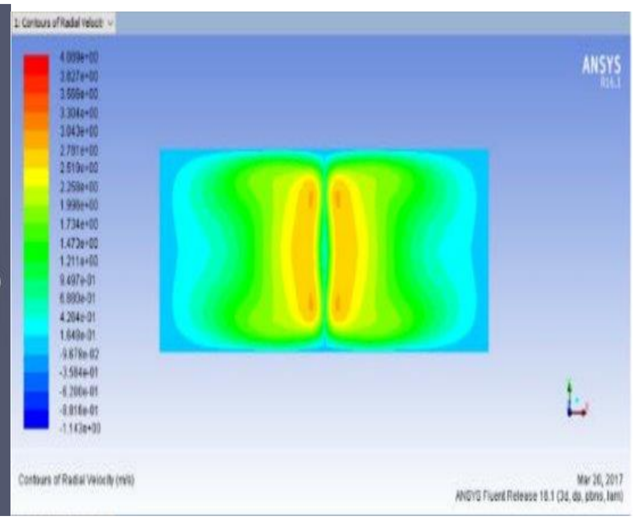

Figure 8

2. Pressure Contours:

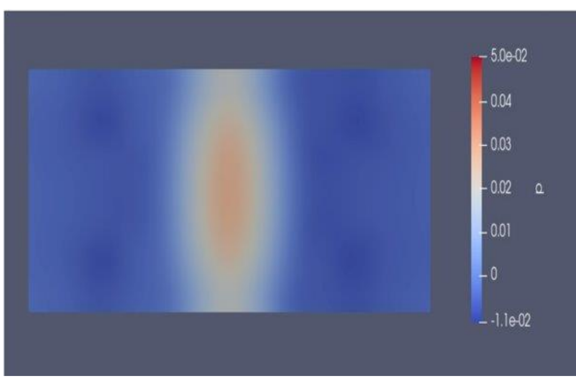

Figure 9

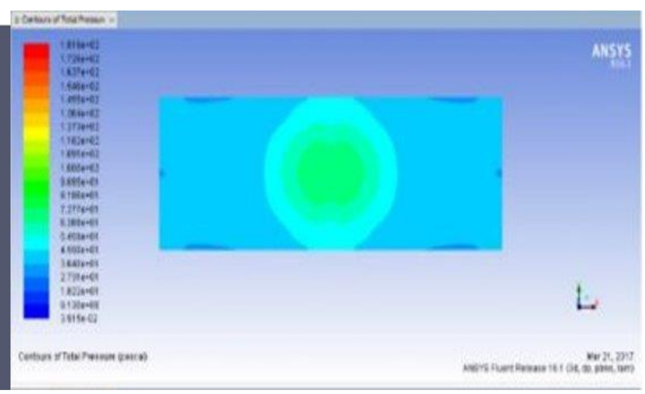

Figure 10

Table 1: Pressure drop for aspect Ratio 1

Temperature (K)	Paper Pressure Difference	Study Pressure Difference	Error Precent
ΔP 1/318K	478.36562	388.64	18.75
ΔP 1/333K	463.64424	383.58	17.26
ΔP 1/348K	462.54667	381.27	17.57

Figure 11: Pressure Difference between Inlet and Outlet at 333K

5. Results and Discussion

The effect of 90◦ bifurcation angle for different temperature on pressure difference on inlet and outlet of a microchannel has been studied. Our simulation showed profile very similar to the velocity and Pressure profiles of the paper which we are validating from the paper [1]. The residual convergence of 0.001 has been achieved as shown in fig. 3. As it is evident from table 1 that our simulation produced the pressure difference at different temperatures of the same scale as the paper we are validating for. The error has induced between our result and paper due to the lesser meshing in z-direction.

The comparison between velocity simulation of our study fig. 7 and reference paper fig. 8 is done. Similarly, the comparison between pressure simulation is done in fig. 9 and fig. 10 are done.

References

[1] Teja Swaroop Naik Mudiki. “Study of pressure drop and heat transfer in Micro-channel branch with varying Bifurcation angle, Aspect ratio, and Temperature ” https://soar.wichita.edu/bitstream/handle/10057/14480/ t17020_Mudiki.pdfisAllowed=y&sequence =215.

[2] Mahesh S. Nagargoje, Deepak k. Mishra and Raghvendra. “Pulsatile flow dynamics in symmetric and asymmetric bifurcating vessels”.

[3] Jonghwun Jung and Ahmad Hassanein. “Three phase CFD analytical modelling of blood flow”.

[4] G.B. Nimadge and S.V. Chopade. “CFD analysis of flow through T-Junction of pipe”.