

Simulation of Flow Through a Y-Splitter

Chaitanya Apte
Veermata Jijabai Technological Institute

1. Abstract

This case study aims to perform a simulation of flow through a Y-splitter using OpenFOAM. The dimensions of the Y-splitter are as mentioned in the figure below. Transient solver pimpleFoam is used for the simulation. The k-epsilon turbulence model is applied. Similar study conducted at SASTRA University using Ansys Fluent is used as reference.

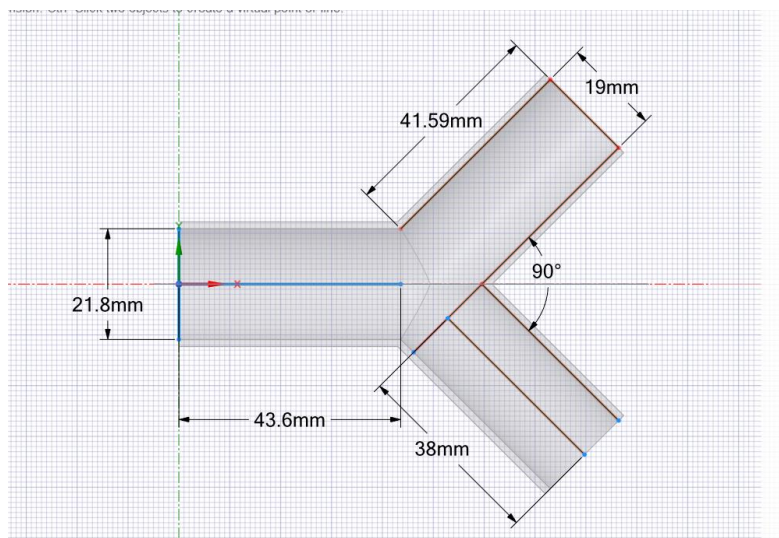


Figure 1:Geometry

2. Introduction

The geometry mentioned in Figure 1 has one inlet and two outlets both with different cross-sectional diameter. Fluid which is air enters with velocity of 1.56 m/s and leaves at the bifurcated outlets. Flow is incompressible since velocity is very low. In the referred Fluent Study, the exact geometry is used to monitor mass flow-rates at the bifurcated outlets to confirm if splitting is uniform. The results of that study are compared with the simulation results performed using OpenFOAM in this project. Additionally, mesh independence study is done using 4 meshes of 6k, 35k, 70k and 1.5L cells respectively.

3. Governing Equations and Models

The Navier-Stokes equations for single-phase flows govern the simulation. The 2-equation based k-epsilon turbulence model is used to capture turbulence in the flow. The governing continuity and momentum equations are given by:

$$\nabla \cdot \mathbf{u} = 0$$

$$\nabla \cdot (\mathbf{u} \otimes \mathbf{u}) - \nabla \cdot \mathbf{R} = -\nabla p + S_u$$

The K-epsilon model is one of the most common turbulence models. It is a two-equation model, as it includes two extra transport equations to represent the turbulent properties of the flow. The first transported variable is turbulent kinetic energy, k . The second transported variable in this case is the turbulent dissipation, ϵ . It is the variable that determines the scale of the turbulence, whereas the first variable, k , determines the energy in the turbulence.

The turbulent kinetic energy equation, k :

$$\frac{D}{Dt}(\rho k) = \nabla \cdot (\rho D_k \nabla k) + P - \rho \epsilon$$

where

- k = Turbulent kinetic energy [m²s⁻²]
- D_k = Effective diffusivity for k [-]
- P = Turbulent kinetic energy production rate [m²s⁻³]
- ϵ = Turbulent kinetic energy dissipation rate [m²s⁻³]

The turbulent kinetic energy dissipation rate equation, ϵ :

$$\frac{D}{Dt}(\rho \epsilon) = \nabla \cdot (\rho D_\epsilon \nabla \epsilon) + \frac{C_1 \epsilon}{k} \left(P + C_3 \frac{2}{3} k \nabla \cdot \mathbf{u} \right) - C_2 \rho \frac{\epsilon^2}{k}$$

where

- D_ϵ = Effective diffusivity for ϵ
- C_1 = Model coefficient
- C_2 = Model coefficient

The turbulent viscosity equation, ν_t

$$\nu_t = C_\mu \frac{k^2}{\epsilon}$$

where

C_μ = Model coefficient for the turbulent viscosity

The model coefficients are $C_\mu = 0.09$; $C_1 = 1.44$; $C_2 = 1.92$

4. Simulation Procedure

4.1 Mesh

The geometry is meshed using `snappyHexMesh` utility available in OpenFOAM. The structured grid consists of 70k cells comprising of hexahedra, prisms and polyhedra. Mesh non-orthogonality is around 9 with max skewness of 1.

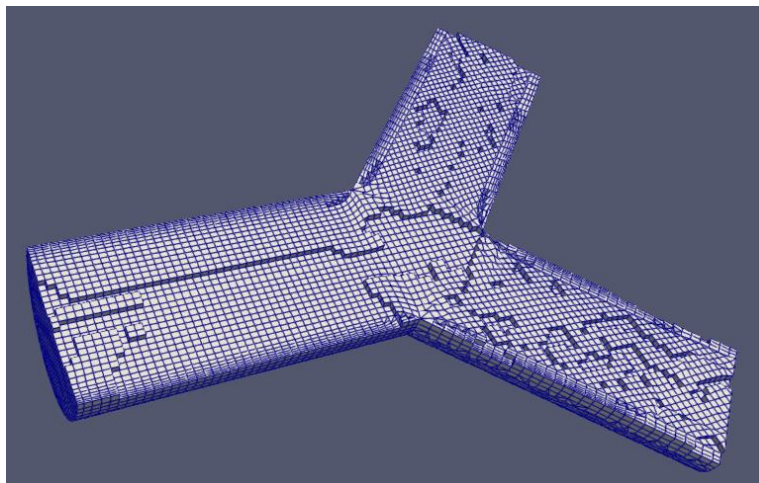


Figure 2: Mesh

4.2 Initial and Boundary Conditions

There are three boundaries that required boundary conditions: An inlet, outlets and walls. At the inlet and outlets, u , p , κ and ϵ are specified, as shown in Table 1. At the walls, wall-function approach used for κ and ϵ . The turbulent viscosity(ν_t) value was calculated based on other parameters hence, kept calculated.

Table 1: Boundary Conditions

	Initial	Inlet	Outlets	walls
$U[m^2 s^{-1}]$	0	1.56	zeroGradient	noSlip
$P[m^2 s^{-2}]$	0	zeroGradient	1632.65	zeroGradient
$k[m^2 s^{-1}]$	0.009126	0.009126	zeroGradient	kqRWallFunction
$\epsilon [m^2 s^{-1}]$	0.01432	0.01432	zeroGradient	epsilonWallFunction
$\nu_t [m^2 s^{-1}]$	0	calculated	calculated	calculated

The turbulent kinetic energy and dissipation rate is calculated by:

$$k = \frac{3}{2} (I |\mathbf{u}_{ref}|)^2$$

where, I = Turbulence intensity[%] and u_{ref} = reference flow speed[ms^{-1}]

$$\epsilon = \frac{C_\mu^{0.75} k^{1.5}}{L} \quad \text{where } C_\mu = \text{model constant equal to } 0.09 \text{ and } L = \text{reference length}$$

4.3 Solver

A transient, incompressible, turbulent-flow solver pimpleFoam is used to solve the governing equations. It uses the PIMPLE algorithm which is a combination of the PISO(Pressure Implicit with Splitting of Operator) and SIMPLE(Semi-Implicit Method for Pressure Linked Equations) algorithms. The simulation runs for 10sec with time step of 0.001sec. Below figure shows the convergence of residuals.

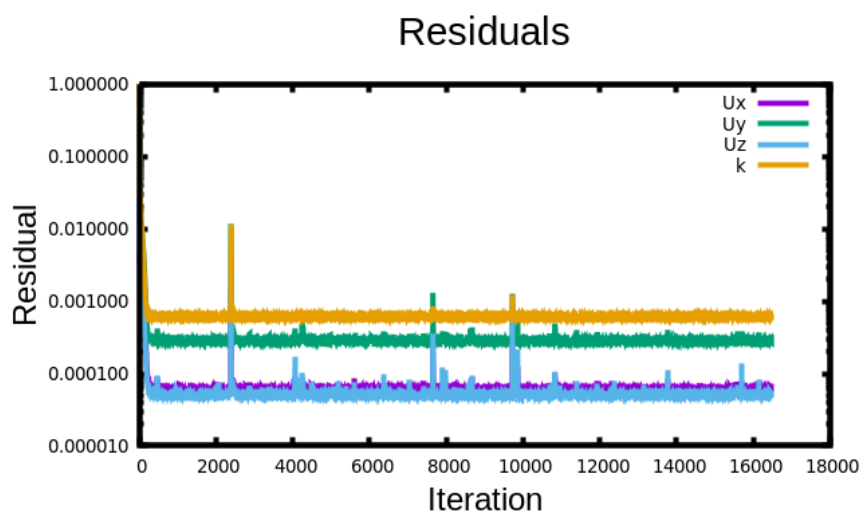


Figure 3: Residuals Convergence

5. Results

The velocity and flowrate values were found for the four different meshes for the mesh independence study. Table 2 shows the outlet flowrates and velocities for each mesh.

Table 2: Comparison of results of different meshes

	M1 6k mesh	M2 35k mesh	M3 70k mesh	M4 1.5L mesh
Outlet1 flowrate	0.000353527	0.000354928	0.000354676	0.000354824
Outlet2 flowrate	0.000353209	0.000354402	0.000354436	0.000354754
Outlet1 velocity	(7.425566e-01 - 3.898796e-04 - 7.329100e-01)	(7.485583e-01 - 2.679062e-03 - 7.093675e-01)	(0.754903 2.49026e-05 -0.702698)	(7.526913e-01 2.296143e-04 - 7.019036e-01)
	1.03486	1.0324	1.031338	1.03086
Outlet2 velocity	(-0.741140 1.957551e-04 - 0.736445)	(-0.7548165 - 2.565011e-03 - 0.70607)	(-0.754046 1.78786e-03 - 0.702435)	(-0.7561935 - 1.042967e-03 - 0.7041932)
	1.03486	1.03112	1.0305	1.03044

Further, flow distribution ratio, which is the ratio of outlet1 flowrate to outlet2 flowrate was calculated and compared with M1. The percentage change was found to be less than 0.1%. It was noticed that the outlet velocity components of the M1 differed slightly when compared to other meshes. This was because the mesh was too coarse (6k cells) which caused errors in approximation of the governing differential equations. So, it was concluded that the mesh comprising of 70k cells is sufficient for the simulation.

Table 3: Mesh Independence

	Flow Distribution ratio	% change with respect to M1
M1 6k cells	1.000901386	0
M2 35k cells	1.001483194	-0.05812846
M3 70k cells	1.000676724	0.02244599
M4 2.5L cells	1.000197517	0.070323478

Figures 4-8 show the variation of pressure, velocity, turbulent kinetic energy, viscosity and dissipation along the splitter geometry.

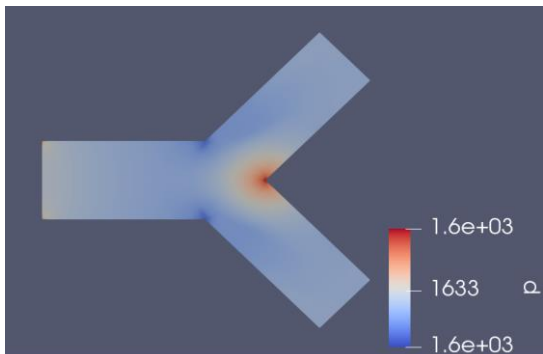


Figure 4 : Pressure

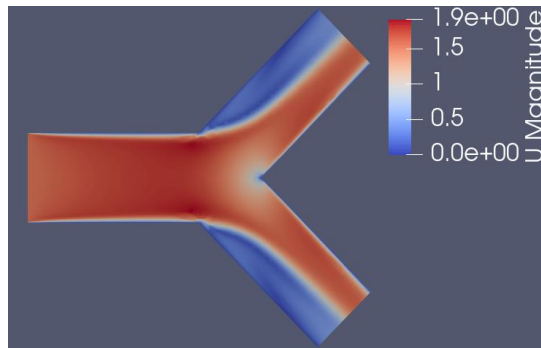


Figure 5: Velocity

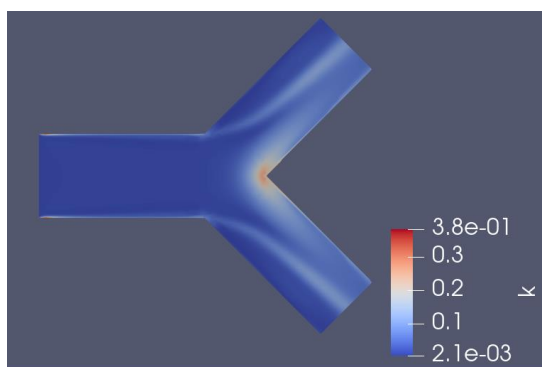


Figure 6: k

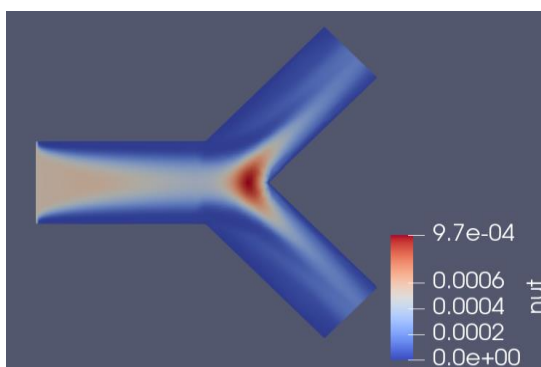


Figure 7: nut

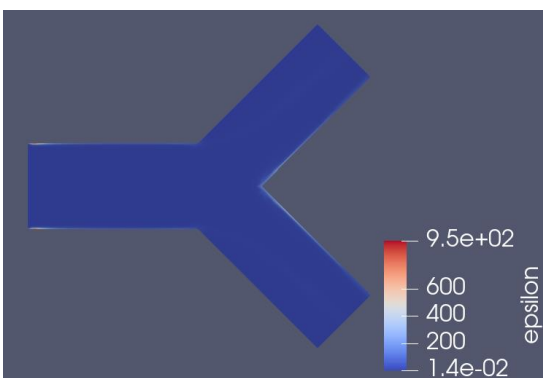


Figure 8: epsilon

Velocities and mass flow-rates at both the outlets were found. Table 4 shows the comparison of OpenFOAM and Fluent results.

Table 4: Results

	Openfoam	Fluent	% difference
Outlet1 flowrate	-0.000354587	-0.0003551312	0.154
Outlet2 flowrate	-0.000354254	-0.000354142	0.03
Outlet1 velocity	1.0329	1.052745	1.88
Outlet2 velocity	1.0321	1.051624	1.85

6. References

[1] Fluid dynamics of a bifurcation: <https://doi.org/10.1016/j.ijheatfluidflow.2019.108483>

[2] Transient flow of evaporating fluid in multiple parallel pipes, theory and experiments: <https://doi.org/10.1016/j.ijheatmasstransfer.2020.120287>