FOSSEE, IIT Bombay
OpenFOAM Case Study Project
February 16, 2020

Study of flow pattern and vortices in a backward stepped flow with two inlets

Shouvik Ghorui
B.E., Jadavpur University, Kolkata

Abstract

The aim of this study is to examine the flow pattern and behavior of vortices in a backward-stepped flow with two inlets in order to get a basic understanding about how mixing occurs within a micro scale combustor along with it the best suitable injection position for flame stability. For simplicity and only understanding the flow physics, single phase problem is simulated. Open-source CFD package, **OpenFOAM®-v7** is used and *pimpleFoam* solver is used for the case study. Here for the case study, a channel type geometry (micro scale combustor) is observed and the secondary inlet is added. The vortices generated are studied by changing the position of the secondary inlet, injection velocity. The second inlet may break the vortex near the step and increase the mixing.

keyword: CFD, OpenFOAM®, BFS, micro scale combustor

1. Introduction

Nowadays many researches going on towards the miniaturization of daily usage devices have resulted in the need for micro power generation devices with low weight, low recharge time and long life. Hence, a combustor with a transverse injection in a backward-facing step is one of the common designs to enhance mixing and flame stabilization. Backward Facing Step (BFS) problem is one of the intriguing problems in the field of Turbulence Fluid Dynamics. In this problem, the flow, after entering from the inlet, passes through a constant area duct and exits into another duct having a cross-sectional area larger than the inlet duct. As this occurs, a recirculation zone occurs at the 'step' of the duct having a larger cross-sectional area. Even though it is a simple and common problem but it has a wide scope which makes it a building block for a variety of applications. This recirculation depends on the Reynolds number of the flow, size of the step, expansion ratio, etc. This paper dives into one such problem of BFS computationally using OpenFOAM®.

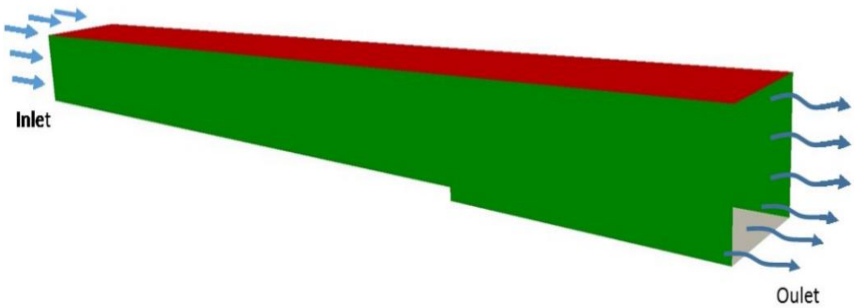

Fig 1: 3D geometry of backward facing step

Flow over backward facing step is a very common problem. This case study includes secondary inlet for better mixing of fluid. A similar study has not been done before. Some common study of backward facing step are referenced in the reference section. [3, 4]

Entrance Length and Fully Developed Flow:

Fully developed flow occurs when the viscous effects due to the shear stress between the fluid particles and plate wall create a fully developed velocity profile. In order for this to occur, the fluid must travel through a particular length. In addition, the velocity of the fluid for a fully developed flow will be maximum at the middle of the two plates.

At first, the flow is not fully developed when it enters two parallel plates. Instead, the fluid has to travel a certain distance undisturbed before it becomes fully developed. This is because the plates will disrupt the velocity profile of the fluid. As a result, it will need to travel a certain distance through the parallel plates to become fully developed again.

The entrance length can be expressed with the dimensionless **Entrance Length Number** as

$$El = l_e / d$$

Where,

El = Entrance Length number
l_e = length to fully developed velocity profile (m, ft)
d = tube or duct diameter (m, ft)

Entrance Length Number for Laminar Flow

The Entrance length number correlates with the Reynolds Number and for laminar flow the relation can be expressed as:

$$El_{laminar} = 0.06*Re$$

Where,
Re = Reynolds Number

In this case, flow through the parallel plate's simulation approaches are considered for incompressible transient flow using pimpleFoam solver in OpenFOAM-v7 [1].

This case study demonstrates how to do the following:

- Set up a problem case;
- Create the mesh file using blockMesh;
- Set up the properties of the fluids;
- Initialize the flow;
- Consider the laminar model for laminar flow regime;
- Post-processing the case for results using the ParaView & Tecplot 360;

2. Problem statement

This case considers the water flow between two parallel plates. The fluid enters through the inlet_1 with a velocity of 1m/s. In this case, flow through the parallel plates simulation approaches is considered for incompressible, isothermal, and transient flow. First fluid enters through the inlet_1 and an injection port is there at the lower plate. The same type of fluid is injected through the injection port (inlet_2). We want to see the effect of the vortex at the step by varying the injection port position and injection velocity. The geometrical parameters and flow conditions are shown as in Table 1 and Table 2 respectively:

Parameters	Value
Total length of the plate (L), m	0.6
Distance between two parallel plates at entry position (D1), m	0.0052
Distance between two parallel plates at exit position (D2)	0.0101
Height of step (H), m	0.0049
Distance between inlet_1 and step (l), m	0.4

Table 1: Geometry details

Fluid property	Value
Kinetic viscosity, fluid (μ), Pa.s	$6.5e^{-6}$
Density of the fluid (ρ), kg/m^3	1000

Table 2: Details of fluids property

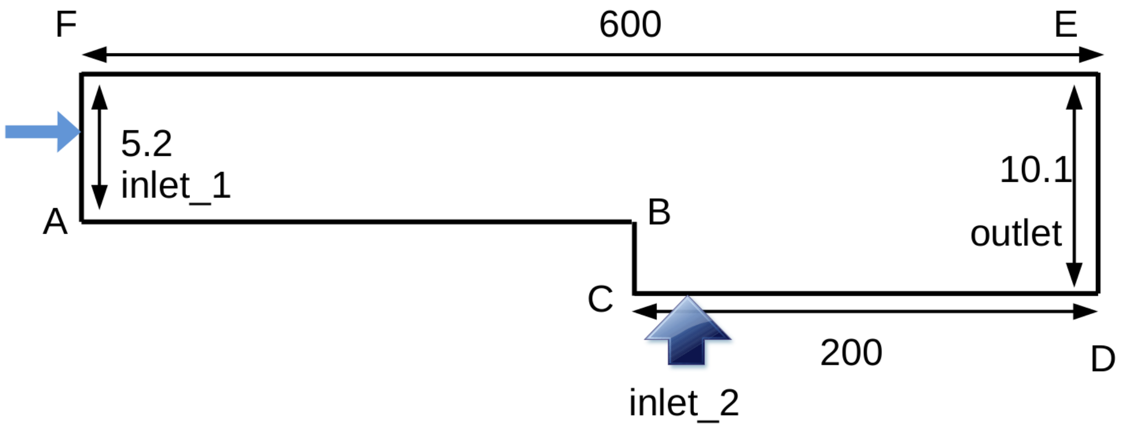

Fig 2: 2D view of backward facing step

The velocity ratio, ε, is defined as the ratio of the injected stream velocity to the free stream velocity, and is expressed as

$$\varepsilon = \frac{V_f}{U_\infty}$$

where, U_{∞} is the free stream velocity which enters at inlet_1, and
V_f is the injection velocity.

3. Mathematical modeling

PimpleFoam (solver) solves the Navier-Stokes equations for isothermal and incompressible flow.

3.1 Continuity equation:

The continuity equation (constant-density) is defined as:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

3.2 Momentum equation:

x-Momentum:

$$\frac{\partial u}{\partial t} + \frac{\partial (uu)}{\partial x} + \frac{\partial (uv)}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \boldsymbol{\vartheta}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right)$$

y-Momentum:

$$\frac{\partial v}{\partial t} + \frac{\partial (uv)}{\partial x} + \frac{\partial (vv)}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \vartheta\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right)$$

4. Simulation procedure:

The case deals with 2D laminar flow simulation of liquid between parallel plates having a step on the lower plate. First, copy a **pimpleFoam** tutorial case from the tutorial folder and paste the folder where we want to keep our simulation file. Then we need to set all require input parameters before starting the simulation. Also, we need to write the blockMeshDict file as our requirement. This case study is considered with the transient flow. And we use **pimpleFoam** solver in this case.

4.1 Creating geometry and mesh

- ✔ Mesh files are generated using ‘blockMesh’ utilities in OpenFOAM
- ✔ Figure 2 shows the front view of the generated mesh using ‘blockMesh’ utility

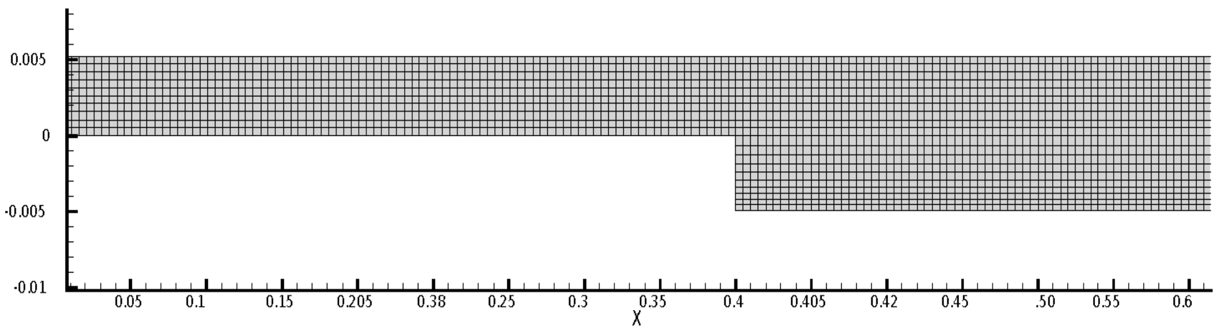

Fig 3: 2D mesh structure

4.2 Setting boundary conditions (BC)/Initial conditions (IC):

There are two files in the 0 folder. One is 'U' and the other one is 'p'. Boundaries are assigned and nine boundaries are added of the present case in two the files, i.e. 'U' & 'p'. Details of the boundary conditions are listed in Table 4.

Boundary	U	P
INLET1	fixedValue	zeroGradient
INLET2	fixedValue	zeroGradient
OUTLET	zeroGradient	fixedValue
LPATE	noSlip	zeroGradient
UPLATE	noSlip	zeroGradient

Table 4: Details of boundary conditions

4.3 Solver details

In this case, we need to analyze Turbulent, Transient flow for an incompressible fluid, we have used pimpleFoam solver. Here, there is no need to solve the energy equation as we are interested mainly in the velocity field and pressure field. PimpleFoam is a Transient solver for incompressible, turbulent flow of Newtonian fluids, with optional mesh motion and mesh topology changes. The PIMPLE Algorithm is a combination of PISO (Pressure Implicit with Splitting of Operator) and SIMPLE (Semi- Implicit Method for Pressure-Linked Equations). All these algorithms are iterative solvers but PISO and PIMPLE are both used for transient cases whereas SIMPLE is used for steady-state cases.

4.4 Post-processing

The ParaView software can be used to visualize the simulations results in OpenFOAM. This can be run by typing 'paraFoam' command line in the terminal to open the ParaView software and upload the case. We have also used **Tecplot 360** for post-processing.

5. Results and discussion:

In a backward stepped flow with only one inlet, vortices are formed mainly near the step region and consists of two main eddies- the primary eddy and the secondary eddy as shown.

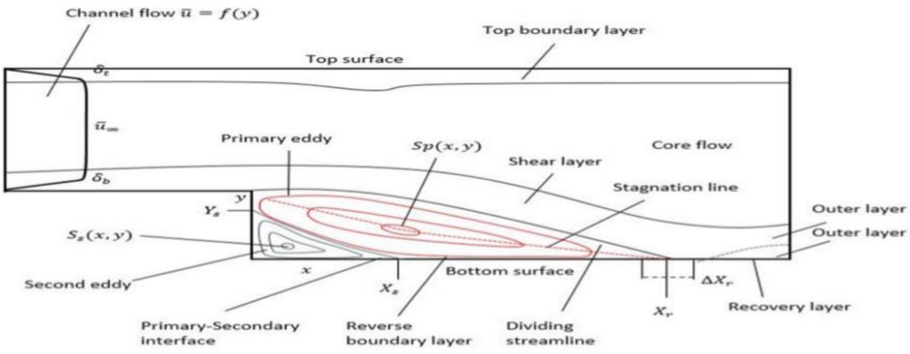

Fig 4: vortices and flow pattern in a backward stepped flow with one inlet

Source: Al-Jelawy, H., Kaczmarczyk, S., Alkhafaji, D., Mirhadizadeh, S., Lewis, R., & Cross, M.R. (2016). A Computational Investigation of a Turbulent Flow over a Backward Facing Step with OpenFOAM. 2016 9th International Conference on Developments in eSystems Engineering (DeSE), 301-307.

Simulations are performed using OpenFOAM to investigate the flow over a backward step with 2 inlets. Results are plotted with the help of 'paraView' (_/case/paraFoam). Simulation results are analyzed with the help of paraFoam software.

The velocity ratio, ε, is defined as the ratio of the injected stream velocity to the free stream velocity, and is expressed as

$$\varepsilon = \frac{V_f}{U_\infty}$$

We will see the effect of change of streamlines and vortex patterns by changing the velocity ratio ε and by varying the position of inlet_2 along the length of the lower_wall.

- **For three different position of the injection**

1. Injection at the point x=0.396 to 0.398

❖ **(**ε=1.2, U_∞=1, Re=800**) t=7sec**

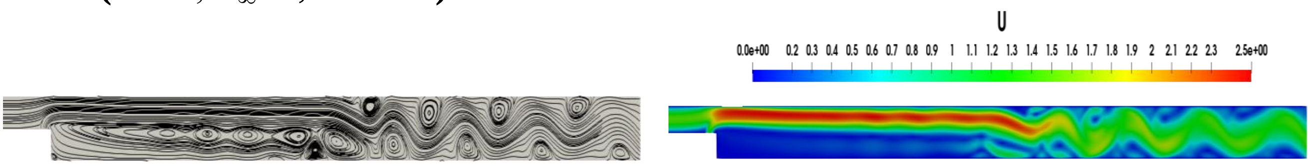

Fig 5: Streamline & contours for secondary inlet x=0.396, ε=1.2

Fig 6: contribution of injection in the flow field, x=0.369, ε=1.2

❖ **(**ε=0.3, U_∞=1, Re=800**) t=7sec**

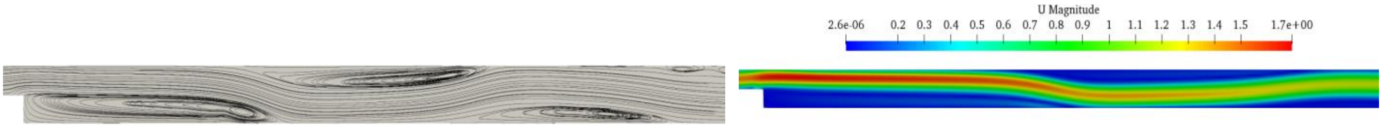

Fig 7: Streamline & contours for secondary inlet x=0.396, ε=0.3

Fig 8: contribution of injection in the flow field, x=0.369, ε=0.3

For these cases it is evident from the velocity contour, streamline and contribution of fluid-stream from inlet_2 to the main flow, that by increasing the ε quality of mixing is increasing. Hence the stability of the flame near the step should increase.

2. Injection at the point x=0.427 to 0.429

❖ **(**ε=1.2, U_∞=1, Re=800**) t=7sec**

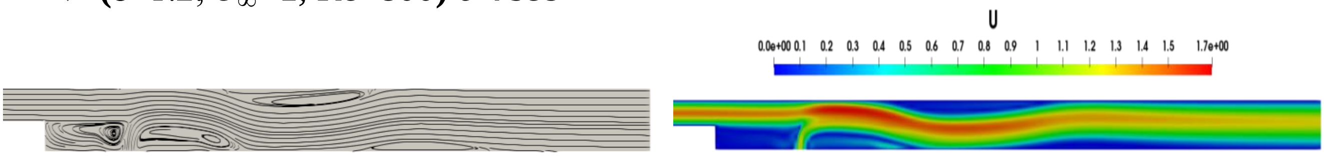

Fig 9: Streamline & contours for secondary inlet x=0.427, ε=1.2

Fig 10: contribution of injection in the flow field, x=0.427, ε=1.2

It is evident from the streamlines that the fluid stream from the inlet_2 consists a major part of the vortex formed at the left side of the inlet_2 and hence it can be said that for this case the quality of mixing should be good at the left of the inlet_2

❖ **(**ε=0.3, U_∞=1, Re=800**) t=7sec**

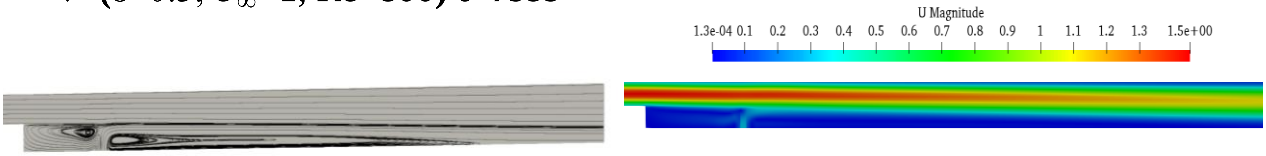

Fig 11: Streamline & contours for secondary inlet, x=0.427, ε=0.3

Fig 12: contribution of injection in the flow field, x=0.427, ε=0.3

It is evident from the streamlines that the fluid stream from the inlet_2 consists a major part of the vortex formed at the right side of the inlet_2 and hence it can be said that for this case the quality of mixing should be good at the right of the inlet_2

---

3. Injection at the point x=0.4, y = -0.0019 to y = -0.0039

- **(**ε=1.2, U_∞=1, Re=800**) t=7sec**

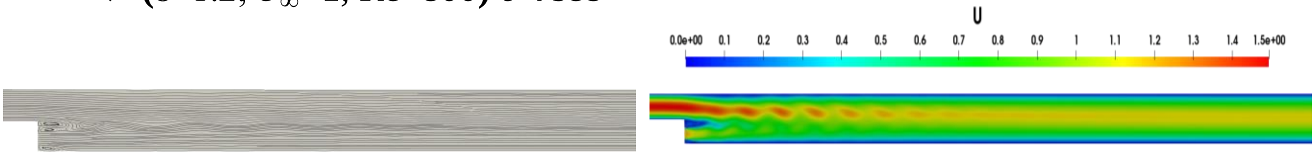

Fig 13: Streamline & contours for secondary inlet x=0.4,ε=1.2

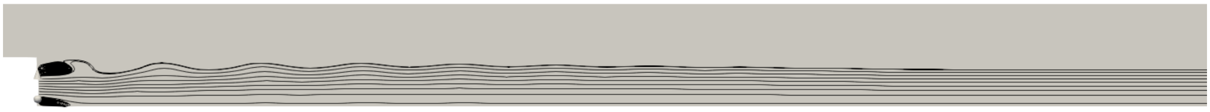

Fig 14: contribution of injection in the flow field, x=0.4, ε=1.2

- **(**ε=0.3, U_∞=1, Re=800**) t=7sec**

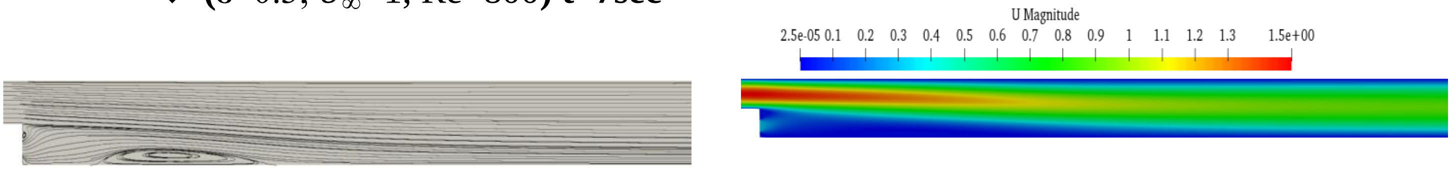

Fig 15: Streamline & contours for secondary inlet x=0.4, ε=0.3

Fig 16: contribution of injection in the flow field, x=0.4, ε=0.3

It is evident from the streamline that there is no significant vortex and the injected fluid stream from inlet_2 moves past forward instead of mixing with the free-stream flow. Hence for this case, it can be commented that the mixing quality should be very poor and hence stability of flame will be very less for this injection port.

Conclusion:

The present geometry can be used for a micro scale combustor application where for reacting flow, mixing is very important.

Here one of the inlets can be used as air inlet and the other one can be used as fuel inlet and by simulating the non-reacting cases, we make an estimation of the quality of mixing based on the intuition that if fluid from two inlets form a bigger vortex then they get enough time to mix and the mixing is better.

After an extensive study of all the above cases, an overall conclusion can be made as to which inlet should be used if the apparatus (geometry) is to be used as a combustor. The case with the inlet_2 at x=0.427 is most suitable and it can be used with both ε=0.3 and 1.2 as each one has its own advantage as explained above. However, the case with inlet_2 at x=0.3966 can be used but an ε of 1.2 should be used. Hence inlet should not be placed here for proper mixing in a combustor, at least for our considered Reynolds number (Re=100). Future works include simulation of multi-component fluids and getting a proper validation of our intuition on mixing.

Acknowledgement:

I would like to express my sincere gratitude to Prof. Sourav Sarkar of Jadavpur University for providing me the opportunity to work on this project. I deeply thank my friend Mr. Riddhideep Biswas for helping me out with this project.

References:

1. C. J. Greenshields, OpenFOAM: The OpenFOAM Foundation. User Guide Version 6. CFD Direct Limited, July. 2018.

2. Numerical Heat Transfer and Fluid Flow by SUHAS.PATANKAR.

3. IIT Bombay spoken tutorial.

4. https://www.researchgate.net/publication/232005183_Experimental_and_Theoretical_Investigation_of_Backward-Facing_Step_Flow

5. https://aip.scitation.org/doi/pdf/10.1063/1.5141565?download=true

6. Al-Jelawy, H., Kaczmarczyk, S., Alkhafaji, D., Mirhadizadeh, S., Lewis, R., & Cross, M.R. (2016). A Computational Investigation of a Turbulent Flow over a Backward Facing Step with OpenFOAM. 2016 9th International Conference on Developments in eSystems Engineering (DeSE), 301-307.