

## ME6151/ ME21M009-AM21S082

### SOLVING FLOW FIELD OVER A BACKWARD-FACING STEP USING SIMPLE ALGORITHM USING A COLLOCATED GRID

**Allu Sai Nandan Reddy**

Thermodynamics & Combustion Engineering Lab  
Department of Mechanical Engineering  
Indian Institute of Technology Madras  
Chennai, India  
me21m009@smail.iitm.ac.in

**Shouvik Ghorui \***

Multiphase Flow Physics Laboratory  
Department of Applied Mechanics  
Indian Institute of Technology Madras  
Chennai, India  
am21s082@smail.iitm.ac.in

#### ABSTRACT

*In this report, we are going to numerically find the velocity and the pressure field for the flow over a backward-facing step in the channel. The SIMPLE algorithm with a staggered grid will be used to solve NSE. The investigation will be done at different Reynolds numbers trying to capture the variation of flow near the step. Experimental results from the literature will be used to validate the results obtained in this report.*

#### NOMENCLATURE

|                |                                  |
|----------------|----------------------------------|
| $\vec{V}$      | Velocity Vector                  |
| $\vec{u}$      | Velocity in $x$ direction        |
| $\vec{v}$      | Velocity in $y$ direction        |
| $p$            | Scalar pressure                  |
| $\rho$         | Density                          |
| $Re$           | Reynolds number                  |
| $a_{nb}$       | Neighbor coefficient             |
| $\vec{u}_b$    | Boundary velocity                |
| $\vec{u}_{nb}$ | Neighbor velocity                |
| $\alpha_p$     | Pressure correction factor       |
| $\alpha_v$     | Velocity correction factor       |
| $\mu$          | Dynamic viscosity                |
| $\delta x$     | Horizontal discretization length |
| $\delta y$     | Vertical discretization length   |
| $u^*$          | Guess velocity                   |

#### INTRODUCTION

The backward Facing Step (BFS) problem is one of the classical problems in the field of Computational Fluid Dynamics which is used to validate the computational methods. In this problem, the flow that enters from the inlet passes through a constant area duct and exits into another duct having a cross-sectional area that larger than the inlet duct. This paper is mainly concentrated on to solve the flowfield over a backward-facing step using SIMPLE algorithm using a Collocated grid.

As can be seen from a vast review articles, progress in CFD, viewed initially as a research topic, has clearly been rapid, with much work done on basic discretization techniques, efficient numerical algorithms, and grid generation methods. In parallel, the instantaneous increase in computing power has provided momentum for extensive research into the mathematical modeling of turbulence, nonNewtonian phenomena, chemical reaction, and multi-phase processes.

In fundamental fluid dynamics, The study of backward-facing step flows is very important. Among the numerous studies that have been done on the topic, the works of Armaly *et al.* [1] stand apart. They presented a detailed experimental and numerical investigation in a backwards-facing geometry for an expansion ratio of  $H/h = 2$ , an aspect ratio  $W/h = 36$  and Reynolds numbers upto  $Re_D = 8000$ . They reported multiple recirculation zones Kim and Moin [3] computed flow over a backward-facing step using Fractional Step method, which is second-order accurate in both space and time. They investigated the reattachment

---

\* Address all correspondence to this author.

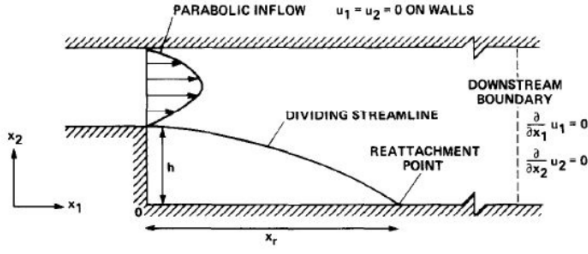


FIGURE 1. Physics of backward facing step

length changing with Reynolds number downstream of the step.

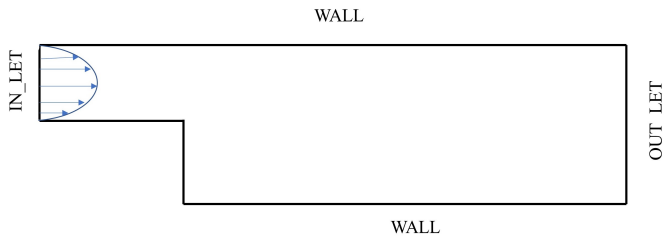


FIGURE 2. Flow domain

## PROBLEM DESCRIPTION

This case considers the **air** flow through a duct. The fluid enters through the inlet with a **parabolic velocity profile** of  $u$  m/s. In this case, flow through the duct simulation approaches is considered for incompressible, isothermal, and laminar. In this paper our concentration to see the effect of varying injection velocity on the flow. The geometrical parameters and flow conditions are shown as in Table 1 and Table 2 respectively:

TABLE 1. Geometry details

| Parameter                 | Value  |
|---------------------------|--------|
| Total length of the plate | 0.11 m |
| Inlet                     | 0.01 m |
| Outlet                    | 0.02 m |
| Step to outlet            | 0.1 m  |

### 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

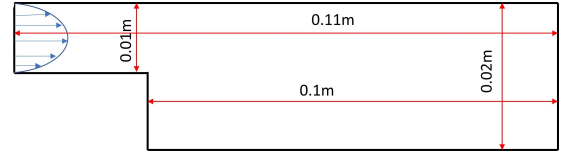


FIGURE 3. Flow geometry

TABLE 2. Details of fluids property

| Fluid property                            | Value         |
|---|---------------|
| Dynamic viscosity, fluid ( $\mu$ ), Pa.s  | $1.8110^{-5}$ |
| Density of the fluid ( $\rho$ ), $kg/m^3$ | 1.2           |

fully developed velocity profile. For this to happen, the fluid must travel through a certain length. In addition, the velocity of the fluid for a fully developed flow will be maximum in the middle of the duct.

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

$$El_{laminar} = 0.06Re$$

In this case, at the inlet of the duct parabolic velocity boundary condition is considered to get a fully-developed velocity boundary condition at the inlet

## Boundary Conditions

In this case Velocity inlet and Pressure outlet (assumed to be zero) boundary condition is used. For the walls no slip boundary condition is applied. On the wall and inlet boundary layer approximation is used for pressure (pressure gradient normal to wall is far less than tangential one) at outlet velocity gradient in the flow direction is taken to be zero.

## Finite Volume Method

In general, finite difference methods have some weaknesses for highly complicated domains. On the other hand, finite volume schemes do not have such limitations. That is because the independent variables are integrated directly into the physical domain and, therefore, grid smoothness is no longer a significant issue [2]

## GOVERNING EQUATIONS

2-D, steady, laminar, incompressible flows of a Newtonian fluid is considered. It can be described by the conservation laws for mass and momentum equation in a Cartesian coordinate system.

### Continuity equation

$$\frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} = 0 \quad (1)$$

### X-momentum equation

$$\frac{\partial \rho u u}{\partial x} + \frac{\partial \rho u v}{\partial y} = -\frac{\partial P}{\partial x} + \nabla(\mu \nabla \cdot u) \quad (2)$$

### Y-momentum equation

$$\frac{\partial \rho u v}{\partial x} + \frac{\partial \rho v v}{\partial y} = -\frac{\partial P}{\partial y} + \nabla(\mu \nabla \cdot u) \quad (3)$$

### Grid Generation

The discretization of the flow domain and the relevant transport equations are the first step to start the finite volume method. First, we need to decide where to store the velocities. For that collocated mesh is used to store velocities, pressure and All variables at the identical location cell center there is only one set of control volumes as shown in Figure.

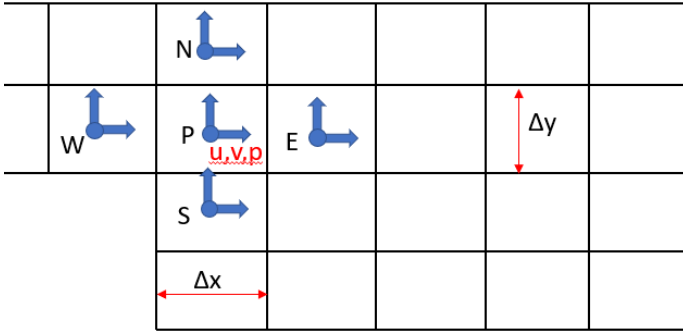


FIGURE 4. Collocated grid arrangement

To avoid velocity and pressure “checkerboard” oscillations we derive face velocities using momentum equations and average them over the adjacent cells instead of directly averaging over adjacent velocities which actually leads to checkerboard oscillations. Staggered grid cannot be used for unstructured meshes whereas co-located can be used irrespective of mesh.

### Solution Method

1. Guess the pressure and velocity field  $p^*, u^*, v^*$ .
2. Solve the  $u$  and  $v$  momentum equations using the prevailing pressure field  $p^*$  to obtain  $u^*$  and  $v^*$  at cell centroids.

$$a_P^u u_P^* = \sum_{nb} a_{nb}^u u_{nb}^* + b_P^u + \Delta y \frac{(p_W^* - p_E^*)}{2} \quad (4)$$

$$a_P^v v_P^* = \sum_{nb} a_{nb}^v v_{nb}^* + b_P^v + \Delta y \frac{(p_W^* - p_E^*)}{2} \quad (5)$$

3. Compute the face mass flow rates  $F^*$  using momentum interpolation to obtain face velocities.

$$u_e^* = \hat{u}_e + d_e (p_P^* - p_E^*) \quad (6)$$

$$v_n^* = \hat{v}_n + d_n (p_P^* - p_N^*) \quad (7)$$

$$\hat{u}_e = \frac{\hat{u}_P + \hat{u}_E}{2} \quad (8)$$

$$d_e = \frac{d_P^u + d_E^u}{2} \quad (9)$$

$$\begin{aligned} d_P^u &= \frac{\Delta y}{a_P^u} \\ d_E^u &= \frac{\Delta y}{a_E^u} \end{aligned} \quad (10)$$

4. Solve the  $p'$  equation.

$$a_P p_P' = \sum_{nb} a_{nb} p_{nb}' + b \quad (11)$$

5. Correct the face flow rates cell-centered velocities  $u_P^*, v_P^*$ .
6. Correct the cell pressure using underrelaxation factor
7. Check for convergence. If converged, stop. Else go to 2.

RESULTS & DISCUSSIONS

In this report, the main aim is to see how reattachment length and flow physics change with increasing the Reynolds number. For the analysis, four different Reynolds numbers are used. Reynolds numbers are obtained by changing the velocity of the inlet. Re number and corresponding inlet velocity are shown in the table.

TABLE 3. Reynolds number and velocity

| No.    | Reynolds number | Velocity |
|--------|-----------------|----------|
| Case 1 | 10              | 0.015    |
| Case 2 | 50              | 0.075    |
| Case 3 | 100             | 0.15     |
| Case 4 | 200             | 0.3      |

In this case from the below figure it can be shown the circulation near the step.And reattachment length can be calculated from the diagram.

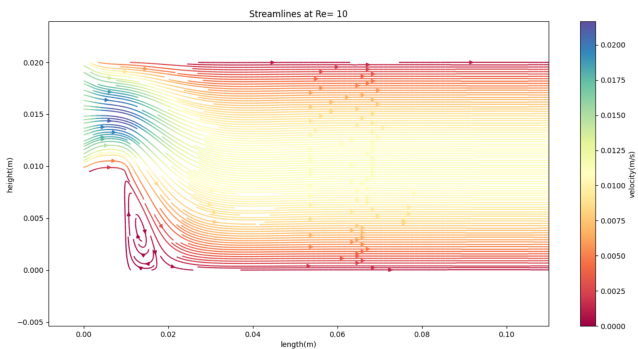


FIGURE 5. For Re 10

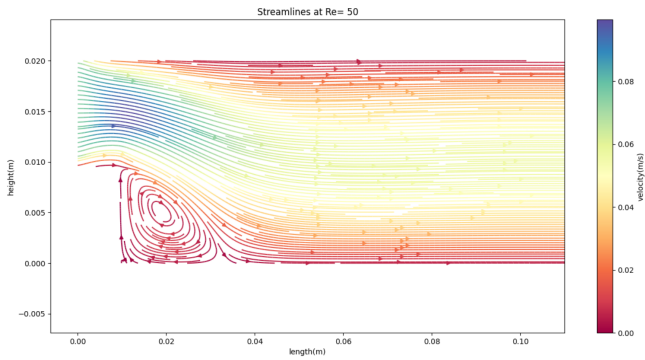


FIGURE 6. For Re 50

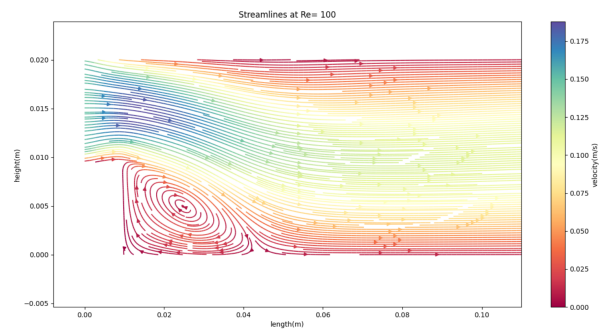


FIGURE 7. For Re 100

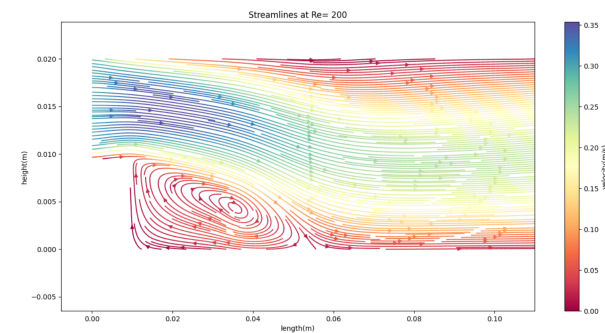


FIGURE 8. For Re 200

conclusion

From the above figures it is clear that increasing the Re number also increase the reattachment length. So to get a large circu-

lation near the step large velocity is preferred For the validation of the results B.F.Armaly et al. [1]’s paper is used.

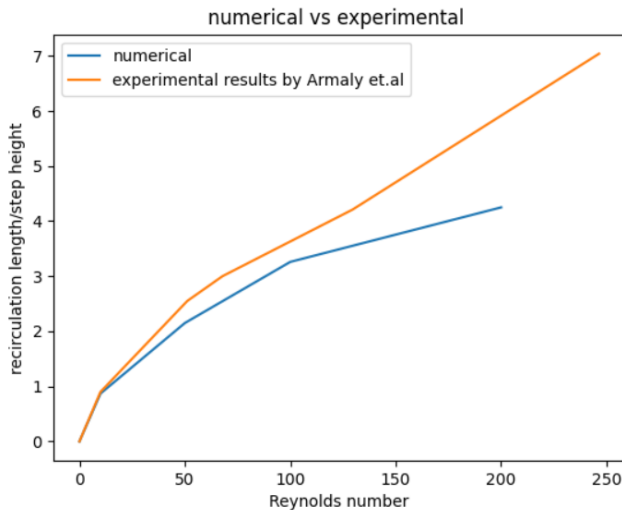


FIGURE 9. Validation

From the above graph it can be stated that at the low  $Re$  our result is matching with the experimental result. We can also observe that the computed results start to deviate from experimental results for  $Re > 150$ , because at higher Reynolds number a finer grid is to be employed to find all the fluctuations at all scales given that our code is written in high-level language and computational power of our computer is not so high we were not able to perform a grid independent test. However after  $Re > 400$  this method fails to give accurate results as reported in numerous works. This discrepancy is not attributed to the numerical error in the solution but due to the flow’s three-dimensionality, which is due to the formation of multiple flow separation zones for  $Re > 400$ , as Armaly et al. has pointed out.

## ACKNOWLEDGMENT

We would like to thank our course instructor **Dr. Kameswarrao Anupindi** for excellent lecture and constant support in completing this project and also very thankful to prof. Sandip Mazumder.

## References

- [1] B. F. Armaly et al. “Experimental and theoretical investigation of backward-facing step flow”. In: *Journal of Fluid Mechanics* 127 (1983), pp. 473–496. DOI: [10.1017/S0022112083002839](https://doi.org/10.1017/S0022112083002839).

- [2] E. Bender. “Numerical heat transfer and fluid flow. Von S. V. Patankar. Hemisphere Publishing Corporation, Washington – New York – London. McGraw Hill Book Company, New York 1980. 1. Aufl., 197 S., 76 Abb., geb., DM 71,90”. In: *Chemie Ingenieur Technik* 53.3 (1981), pp. 225–225. DOI: <https://doi.org/10.1002/cite.330530323>. eprint: <https://onlinelibrary.wiley.com/doi/pdf/10.1002/cite.330530323>. URL: <https://onlinelibrary.wiley.com/doi/abs/10.1002/cite.330530323>.
- [3] J Kim and P Moin. “Application of a fractional-step method to incompressible Navier-Stokes equations”. In: *Journal of Computational Physics* 59.2 (1985), pp. 308–323. ISSN: 0021-9991. DOI: [https://doi.org/10.1016/0021-9991\(85\)90148-2](https://doi.org/10.1016/0021-9991(85)90148-2). URL: <https://www.sciencedirect.com/science/article/pii/0021999185901482>.

4 <https://www.youtube.com/playlist?list=PLVuuXJfoPgT4gJcBAAF>