Proceedings of Computational Heat & Fluid Flow
IIT Madras, Jan-May 2022
April 30, 2021, Chennai, India, , ,

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 and expansion ratios 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
ρ Density
Re Reynolds number
a_{nb} Neighbor coefficient
$\vec{u_b}$ Boundary velocity
$\vec{u_{nb}}$ Neighbor velocity
α_p Pressure correction factor
α_v Velocity correction factor
μ Dynamic Viscosity
δx Horizontal discretization length
δy 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 slove the flowfied 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 up to ReD=8000. They reported multiple recirculation zones Kim and Moin [3] computed flow over a backward-facing

*Address all correspondence to this author.

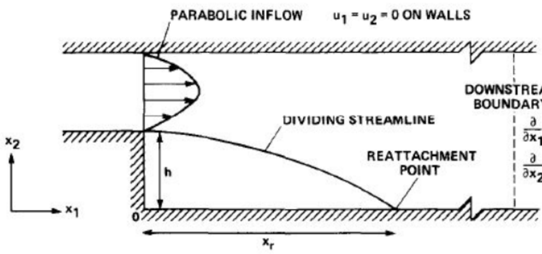

FIGURE 1. Physics of backward Facing Step

step using Fractional Step method, which is second-order accurate in both space and time. They investigated the reattachment length changing with Reynolds number. downstream of the step.

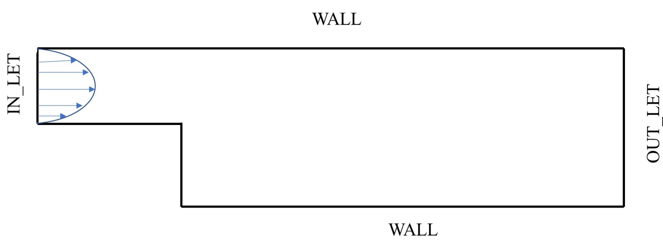

FIGURE 2. Flow Domain

PROBLEM DESCRIPTION

This case considers the **water** flow through a duct. The fluid enters through the inlet with a velocity 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	0.1m
Inlet	0.01m
Outlet	0.02m
Step to outlet	0.08m

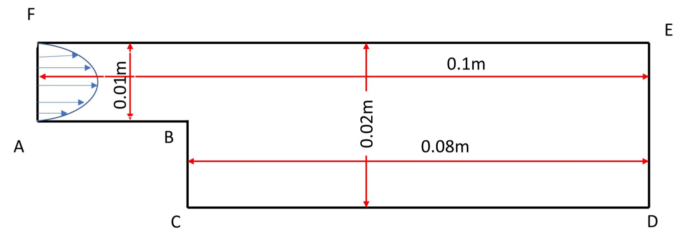

FIGURE 3. Flow Geometry

TABLE 2. Details of fluids property

Fluid Property	value
Kinetic viscosity, fluid (μ), Pa.s	0.1
Density of the fluid (ρ), kg/m^3	0.01

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. 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:

$$\boxed{El_{laminar} = 0.06Re}$$

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

$$\frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} = 0 \tag{1}$$

X-momentum Equation

$$\frac{\partial \rho uu}{\partial x} + \frac{\partial \rho uv}{\partial y} = -\frac{\partial P}{\partial x} + \nabla(\mu \nabla .u) \tag{2}$$

Y-momentum

$$\frac{\partial \rho uv}{\partial x} + \frac{\partial \rho vv}{\partial y} = -\frac{\partial P}{\partial y} + \nabla(\mu \nabla .u) \tag{3}$$

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.

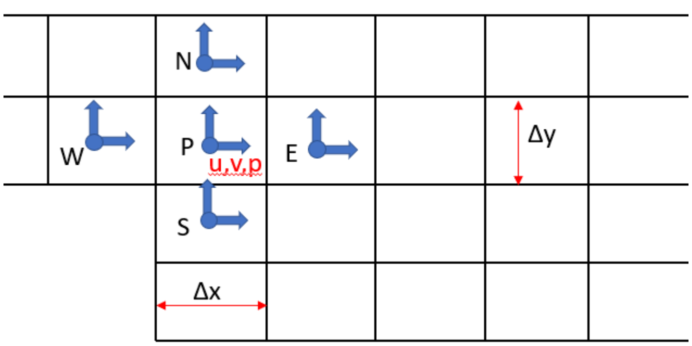

FIGURE 4. Collocated grid arrangement

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.

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

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

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

$$u_e^* = u_e^* + d_e\left(p_P^* - p_E^*\right) \tag{6}$$

$$v_n^* = v_n^* + d_n\left(p_P^* - p_N^*\right) \tag{7}$$

$$u_e^* = \frac{u_P^* + u_E^*}{2} \tag{8}$$

$$d_e = \frac{d_P^u + d_E^u}{2} \tag{9}$$

$$d_P^u = \frac{\Delta y}{a_P^u} \qquad d_E^u = \frac{\Delta y}{a_E^u} \tag{10}$$

4. Solve the p' equation.

$$a_P p'_P = \sum_{\text{nb}} a_{\text{nb}} p'_{\text{nb}} + b \tag{11}$$

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

RESULTS & DISCUSSIONS

we are getting results,But The solution is not converging.

ACKNOWLEDGMENT

We would like to thank our course instructor **Dr.Kameswarrao Anupindi** for excellent lecture and constant support in completing this project.

CITING REFERENCES

first cite [1]

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.

[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. URL: https://www.sciencedirect.com/science/article/pii/0021999185901482.