Modeling and Simulation of Newtonian Fluid Flow through Two-Dimensional Backward-Facing Step Channel with Finite Element’s Technique

Abid Ali Memon1, Hisam-uddin Shaikh1, Baqir Ali Shah2, Muhammad Afzal Soomro3, Abdul Ghafoor Shaikh3,4 and A.H. Sheikh3

1Department of Mathematics, Sukkur IBA University, Sukkur, Sindh, Pakistan; abid.ali@iba-suk.edu.pk
2Department of Mathematics, Shah Abdul Latif University Khairpur Mirs, Khairpur, Sindh, Pakistan; hisam.shaikh@salu.edu.pk, baqirshah@salu.edu.pk
3Department of Mathematics and Statistics, QUEST, Nawabshah, Sindh, Pakistan; m.a.soomro@quest.edu.pk, ah.sheikh@quest.edu.pk
4Department of BS&RS, QUEST, Nawabshah, Sindh, Pakistan; agshaikh@quest.edu.pk

Abstract

Objectives: In this study, the behavior of laminar, incompressible and Newtonian fluid flow through the two dimensional backward facing step channels with the expansion ratio 1:2 is studied. Methods: The governing system of non-linear partial differential equations is worked out with the Galerkin least square finite element method. For computation purpose, the finite element based package COMSOL Multi-Physics 5.0 is used. Findings: The streamline pattern of the velocity field and pressure isobars is conferred at different Reynolds numbers. The minimum and maximum velocity magnitudes as well as pressure at the inlet and outlet including their differences are measured. Moreover, the reattachment length of the vortex organized at the corner of downstream of observed geometry is calculated and contrasted with experimental investigation of Armalyetal and CFD investigation of. Applications: Such fluid dynamics has frequently been used in industries including polymers, petro refineries, paints, biological systems etcetera.

Keywords: Finite Element’s Technique, Newtonian Fluid Flow, Non-Linear Partial Differential Equations

1. Introduction

A definite quantity of numerical investigations has been made to discourse the recirculation flow rates and growth in length of the vortex formed in the silent corner of the two dimensional backward facing step channels. The latter accommodates classical grandness in the computational fluid dynamics due to the downrightness of the geometry and flow symptomatic technique.

Recently1 studied the laminar as well as turbulence effect of flow through 2D backward facing step channel taking expect ratio 1:2, using Ansys Fluent. It was established that the vortex length increase with the increase in inlet velocity up to a certain range and then falls when velocity increases. Using the CFD package Open FOAM2 carried out a numerous simulation exploration of the fluid flow through backward facing step channel with growth ratios (1.48, 2.00 and 3.27). The author initiates that due to variation of the maximum turbulence intensity there is an incurable difference in the reattachment length at the separation and for both Newtonian and Laponite fluid flows, there are no major differences are recovered between the mean and turbulent flows3, attain an experimental probe of the turbulent fluid flow through the rapid channel in observance of expansion and aspect ratio 1.43 and 13.3 respectively. About 1.5 Laponite solutions, shear thinning and thixotropic liquid was reserved to flow and for comparison purposes the two

*Author for correspondence
Newtonian fluids were also experienced. He found that due to variation of the maximum turbulence intensity there is an incurable difference in the reattachment length at the separation and for both Newtonian and Laponite fluid flows, no major distinction is found between the mean and turbulent flows. In\textsuperscript{4}, plumbed the results of separation and reattachment lengths of the upper and lower wall due to Newtonian fluid flow through the backward facing step channel with expansion ratios of 1.17 and 2.0. The whole experiment was performed using negligibly spaced, multi- element hot-film sensor arrays for \(Re < 43000\). First-class results were observed and compared to available literature and determined that the Lees numerical results provide a good effort to study the unsteady turbulent flow using the multi- element hot-films. In\textsuperscript{5}, achieved the numerical solution of the Navier Stokes equation to testify the simulation of laminar and Newtonian fluid flows through the two-dimensional backward facing step channel with high Reynolds numbers up to 3000 by applying the very efficacious finite difference scheme. He found that the length of vortex found in the corner of the backward facing step channel increases in a linear manner with that of increasing in the Reynolds number.

In the present study, we will see the numerical examination of the stream of fluid through two-dimensional backward facing step channel with observance of 1:2 as development proportion. The finite element technique is used to acquire various simulations. Impacts in the fluid regime such as pressure distribution, velocity orientations on three horizontal lines in particular Reynolds number and vortex enhancement with increasing Reynolds numbers are discussed for the down- stream of the geometry.

2. Mathematical Modelling

Supposing the water density, the simulation and modeling is carried out by solving the two-dimensional incompressible Navier Stokes equation and giving out the numerical results for the velocity and pressure variables. The laminar fluid flow actions are being ascertained by preferring the single phase flow interface in the commercial software COMSOL Multi-physics. The governing equations which are going to be used here are given as under:

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

\[
u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = - \frac{1}{\rho} \frac{\partial P}{\partial x} + \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right)
\]

\[
u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = - \frac{1}{\rho} \frac{\partial P}{\partial y} + \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right)
\]

where \(u\) and \(v\) are \(x\) and \(y\) components of the velocity field respectively, \(\mu\) and \(\rho\) are the fluid viscosity and density respectively. To analyze the fluid behaviour it is conventional to use the non-dimensional number, i.e., Reynolds number \(Re\). Hence, by the definition

\[
Re = \frac{\rho U_{av} D_h}{\mu}
\]

where \(U_{av}\) is the average inlet velocity, which depends upon the Reynolds number, while \(D_h\) is hydraulic diameter, which we assume here equal to \(2h\).

3. Numerical Modelling

3.1 Geometry and Boundary Conditions

The Steady state, laminar and Newtonian fluid flow is studied here keeping the expansion ratio 1:2 from the height of the upstream downstream. Figure 1 exhibits the geometrical development and boundary conditions imposed on the channel respectively. To understand the fluid attitude via the backward facing step channel \(h = 5\) mm step length is supposed. A parabolic velocity profile \(U_0 = f(s)\) (\(s\) being a parameter set up by the commercial software COMSOL Multi-Physics) is imposed on the inlet of the channel.

![Figure 1](image-url)
3.2 Mesh

About 10686 triangular elements are used for meshing the whole domain of the channel. The complete mesh statistics and mesh feature are given in the Table 1 and Figure 2. To discretize the governing equation through the finite element method, here we used linear Lagrangian shape function for velocity field as well as for the pressure. The size of the triangular elements is not taken to be uniform on the whole domain, but to get the best approximation where we need typically it moves from 0.02 to 0.45.

Table 1. Mesh statistics of the geometry

| Property                  | Value |
|---------------------------|-------|
| Minimum element quality   | 0.7601|
| Average element quality   | 0.9917|
| Triangular elements       | 10686 |
| Edge elements             | 468   |
| Vertex elements           | 6     |
| Maximum element size      | 0.45  |
| Minimum element size      | 0.02  |
| Curvature factor          | 0.3   |
| Maximum element growth rate| 1.15  |

4. Results and Discussion

A velocity profile considering as a parabolic is tricked at the inlet of the channel to keep an eye on the steady state, laminar, and Newtonian fluid flow through the backward facing step channel. The fluid flow circumstances are checked out with increasing Reynolds number from 100 to 500 with equal intervals of 50. The upshots are furnished through the streamlines pattern of velocity field, pressure contours, tables and graphs. A lot of fluid frameworks can be talked about with simulation and modeling obtained through the COMSOL Multi-Physics v 5.0 but in general we concentrate here the reattachment length of the vortex formed at the left silent corner, the velocity magnitude of the fluid particles and the pressure exerted in the channel.

4.1 Validation and comparison with CFD and Experimental Investigation

The two dimensional laminar and Newtonian fluid flow is monitored numerically via the backward facing step channel using the finite element based software COMSOL multi-physics 5.0, which appliances the latest technique of finite element method of least square Galerkin's strategy. Figure 3 presents with the streamline pattern of the velocity field which evokes that a vertical structure can be seen at the left corner of the channel. This is the primary vortex whose length is increased continually with the increasing Reynolds number from 100 to 500. We validate and proved that our results are as good as result accompanied by CFD investigation and experiments done in[1,6]. Figure 4 transports that the reattachment length can be likened to that of the CFD investigation as well as experimental investigation. In Table 2 we present an extra comparison of $x_1$, where $x_1$ is reattachment length of the primary vortex created the left corner of the channel. From the both spectators and numerical results, it is apparent that our finite element method gives more precise approach with the experimental investigation available in the literature.

Table 2. Comparison table of numerical investigation with CFD investigation

| Remolds Number | Experimental Investigation ($x_1$) | CFD investigation ($x_1/h$) | Present ($x_1/h$) |
|----------------|-----------------------------------|-----------------------------|-------------------|
| 100            | 3.261                             | 3.8                         | 3.243             |
| 150            | 4.379                             | 5.4                         | 4.4966            |
| 200            | 5.376                             | 6.6                         | 5.5916            |
| 250            | 6.23                              | 7.4                         | 6.649             |
| 300            | 7.135                             | 8.3                         | 7.575             |
| 350            | 7.791                             | 9                            | 8.3728            |
| 400            | 8.448                             | 9.2                          | 9.045             |
| 500            | 9.475                             | 10.4                         | 9.8702            |
Figure 4. Comparison with experimental investigation and CFD Simulation.

4.2 Velocity and Pressure Distribution

In this section the authors initiated attempts to analyse the fluid flow through the benchmark problem of backward facing step channel in the field of computational fluid dynamics. To stem a fully matured flow, we impose a parabolic velocity profile at the opening of the channel with function \( u = f(s) \) (where \( s \) being the perimeter preferred typically by COMSOL software). Maximum velocity magnitudes at the inlet and outlet of the channel are communicated through the Figure 5 with percentage reduction of the speed from inlet to outlet of the channel (Table 3). It discloses from the numerical consequences that with the rise in inertial values the difference from entrance to exit of the observed channel, the maximum velocity increases commonly. Moreover, the Figure 5 conveys a message that the maximum velocity at the inlet is increased in linear fashion with that of Reynolds number, but maximum velocity at the passage out is not linear.

Table 3. Maximum velocity at inlet and outlet of channel with percent change

| Re  | \( V_{inlet\ max} \) | \( V_{outlet\ max} \) | Percent decrease |
|-----|---------------------|----------------------|-----------------|
| 100 | 0.017654            | 0.0087979            | 0.88561         |
| 150 | 0.026481            | 0.013283             | 1.3198          |
| 200 | 0.035309            | 0.018072             | 1.7237          |
| 250 | 0.044136            | 0.023311             | 2.0825          |
| 300 | 0.052963            | 0.029017             | 2.3946          |
| 350 | 0.06179             | 0.035136             | 2.6654          |
| 400 | 0.070617            | 0.041626             | 2.8991          |
| 450 | 0.079444            | 0.048791             | 3.0653          |
| 500 | 0.088271            | 0.056332             | 3.1939          |

The horizontal velocity orientation is intrigued at the position, 2.5 mm, 5 mm, and 7.5 mm see Figure 6. The fluid pieces reside any position is in non-linearly manner after the downstream of the channel to turn in a stability state to move with constant speed up to the exit of the channel. Further with improving Re the range to wax the stability state for any position improved. The velocity field orientation which transits at 2.5 mm it has the forte that it passes through vortex structure which proceeded at the downstream of the channel. Thus it has about zero speed where it crosses nearly centers of the vortex.

Figure 5. Maximum velocity magnitude at the inlet and outlet of the channel with their absolute differences.

(a)
Abid Ali Memon, Hisam-uddin Shaikh, Baqir Ali Shah, Muhammad Afzal Soomro, Abdul Ghafoor Shaikh and A.H. Sheikh

Indian Journal of Science and Technology

Vol 12 (32) | August 2019 | www.indjst.org

5. Conclusion

The Steady state laminar and Newtonian fluid behavior through the backward facing step sudden expansion channel is analyzed here by using the finite element based commercial software package of CFD COMSOL Multi-Physics 5.0. About 10686 triangular elements were utilized to find the approximate solution of continuity and momentum equation. The outcomes were shown via the streamlines pattern of the velocity field and pressure isobars. The reattachment lengths were figured at each Reynolds number from 100 to 500. In addition, we conferred the maximum pressure exerted in the opening and exit of the channel (Table 4) which also discloses the truth that pressure will be altering within the domain at each location for a particular Reynolds number. The pressure for a particular Reynolds number is dropping to the middle of the channel then increasing due to decelerating of the fluid rate.
6. Acknowledgement

The authors would be delighted to thank the management of Sukkur IBA University who created a soothing environment of the research. Also, authors desire to mention special thanks to the helpful faculty of the Shah Abdul Latif University who contributes in the paper.

7. References

1. Hossain MA, Rahman MT, Ridwan S. Numerical investigation of fluid flow through a 2d backward facing step channel, International Journal of Engineering. 2013; 2(10):3700–08.
2. Choi HH, Nguyen VT, Nguyen J. Numerical investigation of backward facing step flow over various step angles, Procedia Engineering. 2016; 154:420−25. https://doi.org/10.1016/j.proeng.2016.07.508.
3. Poole R, Escudier M. Turbulent flow of non-newtonian liquids over a backward-facing step: Part II: Viscoelastic and shear-thinning liquids, Journal of Non-newtonian Fluid Mechanics. 2003; 109(2-3):193–230. https://doi.org/10.1016/S0377-0257(02)00168-4.
4. Lee T, Mateescu D. Experimental and numerical investigation of 2-d backward-facing step flow, Journal of Fluids and Structures. 1998; 12(6):703–16. https://doi.org/10.1006/jfls.1998.0166.
5. Erturk E. Numerical solutions of 2-d steady incompressible flow over a backward-facing step, part I: High Reynolds number solutions, Computers and Fluids. 2008; 37(6):633–55. https://doi.org/10.1016/j.compfluid.2007.09.003.
6. Shah B, Shaikh H, Baloch A, Ahmed Mahar J. Mathematical modelling for the fibre suspension flow passed corrugated pipes, International Journal of Computer Science and Information Security. 2016; 14(12):923–29.
7. Armaly BF, Durst F, Pereira J, Schonung B. Experimental and theoretical investigation of backward-facing step flow, Journal of Fluid Mechanics. 1983; 127:473–96. https://doi.org/10.1017/S0022112083002839.