

FOSSEE–CFD Learning Program

CFD Analysis of Shear-Thinning Fluid Flow in a Converging–Diverging Channel

Samuel Sagar Ambati¹, Evan Fernandes²

¹Department of Chemical Engineering, Rajiv Gandhi University of Knowledge
Technologies–RK Valley, Andhra Pradesh, India

²FOSSEE Project, Indian Institute of Technology Bombay, Mumbai, Maharashtra, India

Abstract

This study investigates the flow behaviour of a shear-thinning non-Newtonian fluid through a two-dimensional converging-diverging channel using Computational Fluid Dynamics (CFD). OpenFOAM v2012 and the nonNewtonianIcoFoam solver are used to simulate transient, incompressible, laminar flow. A dilute aqueous partially hydrolyzed polyacrylamide (HPAM) polymer solution is represented using a power-law viscosity model with $k = 0.02$ and $n = 0.6$, where $n < 1$ describes shear-thinning behaviour. The computational domain is generated using blockMesh and consists of a converging section, a constant-area throat, and a diverging section. A mesh-independence study is performed using five structured mesh levels ranging from 17,496 to 437,400 cells, and area-averaged throat pressure is used to assess mesh sensitivity. The Very Fine mesh containing 194,400 cells is selected for the subsequent geometry study because the change in throat pressure relative to the Very Very Fine mesh is approximately 0.78%. Four geometries are then investigated by keeping the converging angle fixed at 11° and varying the diverging angle from 4° to 7° . Area-averaged velocity, pressure, and dynamic viscosity are evaluated at selected cross-sections from the inlet to the outlet. The results show acceleration toward the throat, pressure reduction followed by downstream recovery, and shear-dependent viscosity variation. The study demonstrates the combined influence of mesh resolution, channel geometry, and non-Newtonian rheology on the predicted flow behaviour.

Keywords: Computational Fluid Dynamics (CFD), Non-Newtonian Fluid, Shear-Thinning Fluid, Power-Law Model, Converging-Diverging Channel, OpenFOAM, Mesh Independence, Pressure and Velocity Distribution.

Contents

Abstract	1
1 Introduction	6
1.1 Background of the Topic	6
1.2 Motivation of the Study	6
1.3 Problem Statement	7
2 Governing Equations	7
2.1 Continuity Equation	7
2.2 Momentum Equation	7
2.3 Constitutive Equation for the Power-Law Fluid	8
2.4 Apparent Kinematic and Dynamic Viscosity	8
2.5 Shear-Rate Definition	8
3 Flow Setup	9
4 Computational Domain	9
4.1 Geometry and Mesh Generation	9
4.2 Mesh Independence Study	10
4.2.1 Pressure-based mesh independence	11
4.2.2 Selection of the mesh	12
4.3 Boundary and Initial Conditions	12
4.4 Solver Selection	12
4.5 Discretization Schemes	13
5 Implementation in OpenFOAM	13
5.1 OpenFOAM Case Structure	13
5.2 Simulation Procedure and Post-Processing	14
6 Results	15
6.1 Flow field visualization	15
6.1.1 Velocity	16
6.1.2 Pressure	18
6.1.3 Dynamic Viscosity	19
7 Conclusion	20
8 Limitations of the Study	21
9 References	22

10 Appendix	23
A blockMeshDict: Geometry and Mesh Definition	23

List of Tables

Table 1	Flow setup used in the simulations.	9
Table 2	Mesh levels used in the mesh-independence study.	11
Table 3	Predicted throat pressure for each mesh level.	11
Table 4	Geometrical parameters and area-averaged velocity characteristics for different diverging angles.	17
Table 5	Geometrical parameters and area-averaged pressure characteristics for different diverging angles.	18
Table 6	Dynamic viscosity at selected locations for different diverging angles. . . .	19

List of Figures

Figure 1	Schematic of the two-dimensional converging-diverging channel with the principal geometric dimensions.	10
Figure 2	Structured hexahedral mesh generated using blockMesh.	10
Figure 3	Variation of area-averaged throat pressure with mesh type.	11
Figure 4	Residual convergence history for U_x , U_y , and p	13
Figure 5	OpenFOAM case folder structure used in the present study.	14
Figure 6	Simulation workflow.	15
Figure 7	Velocity contour of the converging-diverging channel.	16
Figure 8	Area-averaged velocity and pressure along the channel for the four diverging-angle cases, shown using a dual-axis representation.	17
Figure 9	Pressure contour of the converging-diverging channel.	18
Figure 10	Variation of dynamic viscosity with equivalent shear rate.	20

1 Introduction

1.1 Background of the Topic

Non-Newtonian fluids don't follow Newton's law of viscosity — their apparent viscosity changes with shear rate instead of staying constant. Such fluids show up everywhere, from chemical and polymer processing to food processing and biomedical systems, and their behaviour becomes especially important in channels with varying cross-section, where velocity, pressure, and shear rate all shift along the flow path.

A converging-diverging channel consists of a converging section, a narrow throat, and a diverging section. As the fluid enters the converging region, the reducing area speeds it up and drops the pressure, reaching peak velocity and minimum pressure at the throat. Beyond that, in the diverging section, the flow slows down and pressure recovers. For a shear-thinning fluid, these velocity and shear-rate changes also affect the apparent viscosity — behaviour typically captured using the power-law model.

CFD offers a practical way to study these flow characteristics without relying purely on experiments. In this study, OpenFOAM v2012 was used to simulate incompressible, laminar flow of a shear-thinning non-Newtonian fluid through a two-dimensional converging-diverging channel. In this study, a dilute aqueous HPAM polymer solution is considered as the representative shear-thinning working fluid.

1.2 Motivation of the Study

Flow through a converging-diverging channel depends strongly on both channel geometry and fluid rheology. For non-Newtonian fluids, changes in local shear rate alter the effective viscosity, which in turn affects the pressure and velocity fields — making this interaction essential to understand for accurate flow prediction.

Building on this, the present study aims to develop a systematic CFD methodology for examining how pressure, velocity, and viscosity vary in a shear-thinning fluid through a converging-diverging channel, and how channel geometry influences these quantities — with particular focus on the throat region, where flow velocity and pressure change most significantly.

Converging-diverging channels are common in engineering systems, as changes in cross-sectional area cause significant variations in fluid velocity and pressure. These effects become more complex for non-Newtonian fluids, whose apparent viscosity depends on local shear rate. Shear-thinning power-law fluids, in particular, show decreasing viscosity with increasing shear rate, setting them apart from Newtonian fluids. Prior studies have shown that both fluid rheology and channel geometry influence pressure and velocity distributions in such flows.

1.3 Problem Statement

This study focuses on a 2D converging-diverging channel carrying a shear-thinning power-law fluid with $n = 0.6$. Since mesh resolution directly affects both accuracy and computational cost, five mesh levels are examined to establish mesh independence before further analysis. Different diverging-channel angles are then studied to assess their effect on flow behaviour, with pressure, velocity, and effective viscosity evaluated at multiple locations from the inlet through the converging section, throat, diverging section, and outlet.

In summary, this study aims to determine how mesh resolution and diverging angle influence the pressure, velocity, and effective-viscosity behaviour of a shear-thinning power-law fluid, and to establish a reliable CFD methodology for such analysis using OpenFOAM v2012. The working fluid is represented by a dilute aqueous HPAM polymer solution, selected as a representative polymeric shear-thinning fluid. The study therefore focuses on the hydrodynamic behaviour of this polymer solution rather than on a specific commercial formulation.

2 Governing Equations

This study considers two-dimensional, incompressible, laminar flow of a shear-thinning non-Newtonian fluid through a converging-diverging channel. Simulations are carried out using the nonNewtonianIcoFoam solver in OpenFOAM, which is designed for transient, incompressible, laminar non-Newtonian flows.

2.1 Continuity Equation

Since the fluid is treated as incompressible, conservation of mass reduces to

$$\nabla \cdot \mathbf{U} = 0 \quad (1)$$

where $\mathbf{U}$ is the velocity vector. This simply ensures that no mass is created or lost anywhere in the computational domain.

2.2 Momentum Equation

Momentum conservation for the incompressible fluid is written as

$$\rho \left(\frac{\partial \mathbf{U}}{\partial t} + \mathbf{U} \cdot \nabla \mathbf{U} \right) = -\nabla p + \nabla \cdot \boldsymbol{\tau} \quad (2)$$

where ρ is the fluid density, p is pressure, and $\boldsymbol{\tau}$ is the viscous stress tensor. Even though the flow here is laminar, this equation still accounts for transient acceleration, convective effects, pressure forces, and viscous stresses acting on the fluid.

2.3 Constitutive Equation for the Power-Law Fluid

The shear-thinning behaviour of the dilute aqueous HPAM polymer solution is represented using the Ostwald–de Waele power-law model. In conventional rheological notation, the constitutive relation is expressed in terms of the consistency index and flow behaviour index. For the present CFD implementation, the OpenFOAM powerLaw model uses a kinematic-viscosity formulation. The parameters used in the simulation are $k = 0.02$ and $n = 0.6$. Since $n < 1$, the fluid exhibits shear-thinning behaviour, meaning that its apparent viscosity decreases with increasing shear rate.

2.4 Apparent Kinematic and Dynamic Viscosity

The OpenFOAM power-law model calculates the local kinematic viscosity as a function of shear rate. The resulting kinematic viscosity field is denoted by ν . To obtain dynamic viscosity for physical interpretation and reporting, the kinematic viscosity is converted using the density of the dilute aqueous HPAM solution:

$$\mu = \rho \nu \quad (3)$$

where μ is the dynamic viscosity in Pa·s, ρ is the fluid density in kg/m³, and ν is the kinematic viscosity in m²/s.

A density of $\rho = 998.2$ kg/m³ is used as the representative density of the dilute aqueous polymer solution at approximately room temperature. Because the solution is dilute, its density is treated as approximately equal to that of water for the present incompressible-flow analysis.

For a power-law fluid, apparent kinematic viscosity is given by

$$\nu_{app} = k \dot{\gamma}^{n-1} \quad (4)$$

With $n = 0.6$, this becomes

$$\nu_{app} = 0.02 \dot{\gamma}^{-0.4} \quad (5)$$

This means that in regions of higher shear rate — particularly near the converging section and throat — the apparent viscosity drops noticeably. This shear-thinning response is one of the key behaviours examined in this study.

2.5 Shear-Rate Definition

The shear rate is derived from the rate-of-deformation tensor:

$$\dot{\gamma} = \sqrt{2 \mathbf{D} : \mathbf{D}} \quad (6)$$

where

$$\mathbf{D} = \frac{1}{2} [\nabla \mathbf{U} + (\nabla \mathbf{U})^T] \quad (7)$$

Since shear rate depends directly on the velocity gradient, any change in channel area ends up affecting both the local velocity and the local viscosity. The calculated local shear rate determines the spatial variation of the kinematic viscosity, while the corresponding dynamic viscosity is obtained subsequently from $\mu = \rho \nu$ using the constant solution density.

3 Flow Setup

The simulations consider two-dimensional, incompressible, laminar flow of a shear-thinning non-Newtonian fluid. The nonNewtonianIcoFoam solver is used with a power-law rheological model. The principal flow and numerical parameters are summarized in Table 1.

Table 1: Flow setup used in the simulations.

Parameter	Specification
Flow regime	Laminar
Fluid	Incompressible, non-Newtonian
Solver	nonNewtonianIcoFoam
Rheological model	Power-law
Flow behaviour index	$n = 0.6$ (shear-thinning)
Dimensionality	Two-dimensional
Flow type	Transient

4 Computational Domain

4.1 Geometry and Mesh Generation

The computational domain is a two-dimensional converging-diverging channel designed to study the flow of a shear-thinning non-Newtonian fluid through a varying cross-section. The channel is symmetric about the centreline $y = 0$ and extends from $x = 0$ to $x = 140$. The converging section has an angle of 11° , the throat has a height of 20 units and a length of 20 units, and the diverging section has an angle of 4° for the reference geometry shown in Fig. 1. The inlet and outlet heights are 41.04 and 29.216 units, respectively. The geometry is divided at $x = 20, 40, 54.12, 74.12$, and 110 to form six structured hexahedral blocks.

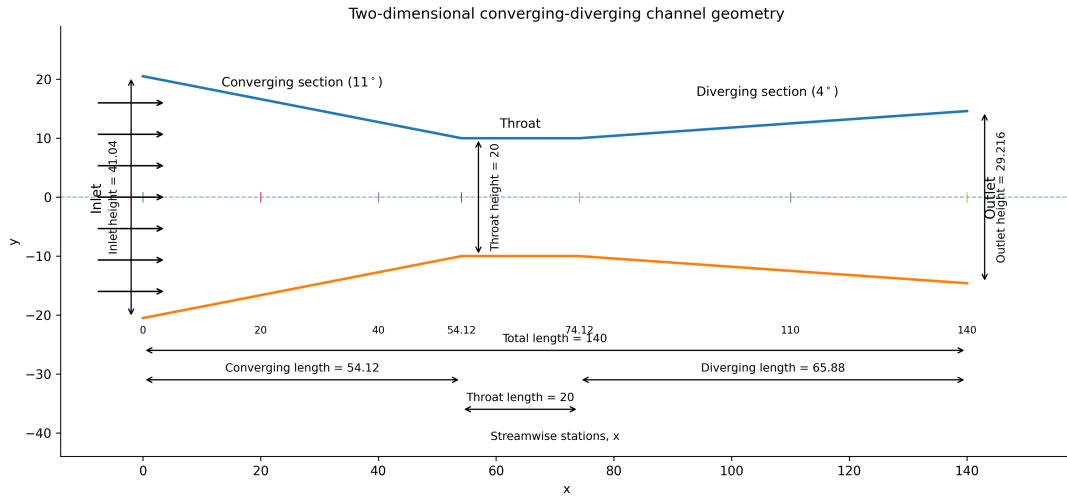


Figure 1: Schematic of the two-dimensional converging-diverging channel with the principal geometric dimensions.

The throat extends from $x = 54.12$ to $x = 74.12$, giving a length of 20 units. The domain has a thickness of one unit in the z -direction with one cell through the thickness, and the front and back faces are treated as empty. The six-block arrangement provides a continuous structured mesh along the streamwise direction.

The mesh is generated using OpenFOAM v2012's `blockMesh` utility. For the Very Fine mesh used in the subsequent geometry study, each block contains $180 \times 180 \times 1$ cells, giving a total of 194,400 cells. A uniform distribution is applied within each block using `simpleGrading` (1 1 1).

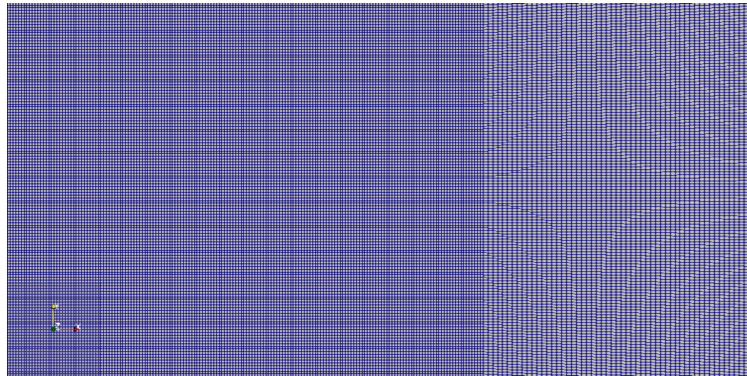


Figure 2: Structured hexahedral mesh generated using `blockMesh`.

4.2 Mesh Independence Study

A mesh independence study was carried out to determine the effect of spatial discretization on the predicted flow characteristics. The same geometry and six-block topology were retained at every mesh level, while only the cell count was increased. Five meshes containing 17,496, 38,400, 86,400, 194,400, and 437,400 cells were investigated, as summarized in Table 2.

Table 2: Mesh levels used in the mesh-independence study.

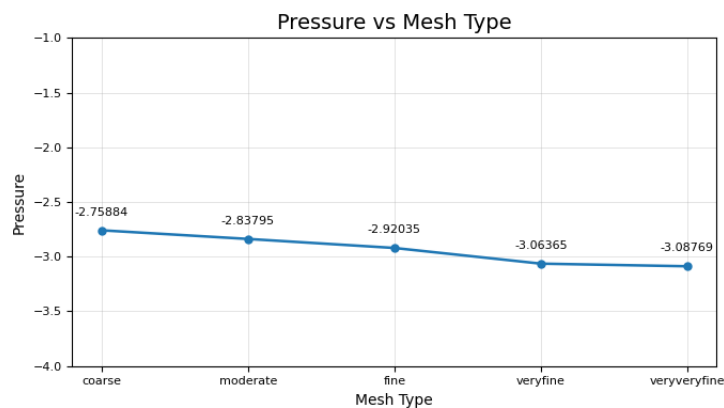
Mesh level	Cells per block	Total cells
Coarse	$54 \times 54 \times 1$	17,496
Moderate	$80 \times 80 \times 1$	38,400
Fine	$120 \times 120 \times 1$	86,400
Very Fine	$180 \times 180 \times 1$	194,400
Very Very Fine	$270 \times 270 \times 1$	437,400

4.2.1 Pressure-based mesh independence

Area-averaged throat pressure was used to assess mesh sensitivity. The predicted pressure changed from -2.75884 Pa on the Coarse mesh to -3.08769 Pa on the Very Very Fine mesh. The change between the Fine and Very Fine meshes remained noticeable, whereas refinement from Very Fine to Very Very Fine changed the pressure from -3.06365 to -3.08769 Pa, corresponding to approximately 0.78%.

Table 3: Predicted throat pressure for each mesh level.

Mesh	Pressure (Pa)
Coarse	-2.75884
Moderate	-2.83795
Fine	-2.92035
Very Fine	-3.06365
Very Very Fine	-3.08769

**Figure 3:** Variation of area-averaged throat pressure with mesh type.

4.2.2 Selection of the mesh

Although the Very Very Fine mesh provides the highest resolution, it contains 437,400 cells, more than twice the 194,400 cells of the Very Fine mesh. Increasing from Very Fine to Very Very Fine therefore increases the cell count by approximately 125%, while the throat-pressure prediction changes by only about 0.78%. Considering the small change in the predicted pressure relative to the additional computational cost, the Very Fine mesh with 194,400 cells was selected for the subsequent geometry simulations.

4.3 Boundary and Initial Conditions

The inlet velocity was prescribed using a fixedValue condition. A value of $U_{in} = 1.15$ m/s was used for the mesh-independence study, while $U_{in} = 2.0$ m/s was used for the final geometry cases. At the outlet, velocity was allowed to develop using a zeroGradient condition, while noSlip was applied at the channel walls. The front and back faces were assigned empty because the computational domain is two-dimensional.

Pressure was assigned zeroGradient at the inlet, outlet, and walls, with empty on the front and back faces. The working fluid was represented by a dilute aqueous HPAM polymer solution using the power-law model with $k = 0.02$ and $n = 0.6$, together with $v_{min} = 1 \times 10^{-5}$ m²/s, $v_{max} = 1 \times 10^{-1}$ m²/s, and $\rho = 998.2$ kg/m³. The local dynamic viscosity was obtained from $\mu = \rho \nu$.

4.4 Solver Selection

Simulations were carried out using the nonNewtonianIcoFoam solver, which provides the transient incompressible laminar formulation required for the present non-Newtonian flow problem. The solver couples the continuity and momentum equations while allowing viscosity to vary with the local flow conditions through the power-law model. A PISO-based pressure–velocity coupling procedure was used for the transient solution.

Numerical convergence was monitored using the solverInfo function object, which records initial and final residuals, solver iterations, and convergence status for the velocity and pressure fields. The resulting residual histories were used to assess numerical stability and convergence during the simulations.

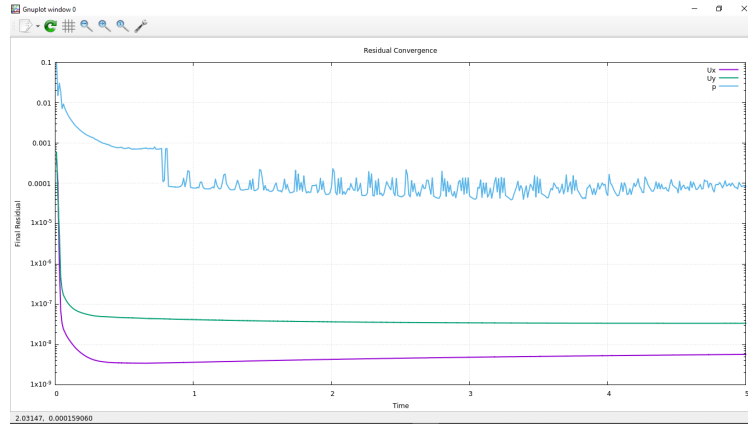


Figure 4: Residual convergence history for U_x , U_y , and p .

4.5 Discretization Schemes

The governing equations were discretized using the finite volume method in OpenFOAM v2012. The transient term was treated using the first-order Euler scheme, gradients using Gauss linear, and the convective term using Gauss upwind. Diffusion and viscous terms used Gauss linear corrected, while face interpolation used linear interpolation and surface-normal gradients used the corrected scheme. The same discretization settings were retained across the mesh-independence cases.

5 Implementation in OpenFOAM

5.1 OpenFOAM Case Structure

The OpenFOAM case follows a standard directory structure. The initial and boundary conditions are stored in the 0/ directory, while the fluid properties and computational mesh are stored under constant/. The system/ directory contains the mesh, numerical, solver, and simulation-control dictionaries. The postProcessing/ directory contains the files generated during post-processing. The Allrun and Allclean scripts are used to run the case and remove generated files, respectively. The structure of the case used in the present study is shown in Fig. 5.

|- - - Allrun
|- - - Allclean

```
0/: folder
```

$$\begin{array}{l} | - - - U \\ | - - - p \end{array}$$

constant/: folder

```
|- - - transportProperties
|- - - turbulenceProperties
|- - - polyMesh/: folder
                |- - - - - boundary
                  - faces
                  - faceZones
                  - neighbour
                  - owner
                  - points
```

system/: folder

```
| - - - blockMeshDict
| - - - topoSetDict
| - - - controlDict
| - - - fvSchemes
| - - - fvSolution
```

postProcessing/: folder

```
|- - - solverInfo/: folder
|- - - surfaceFieldValue/: folder
```

Figure 5: OpenFOAM case folder structure used in the present study.

5.2 Simulation Procedure and Post-Processing

The simulations were executed in serial mode for each mesh and channel-geometry configuration. The workflow consisted of preparing the case, generating the mesh with `blockMesh`, checking the mesh with `checkMesh`, creating sampling face zones using `topoSet`, and running `nonNewtonianIcoFoam`. During the simulation, `solverInfo` monitored the numerical solution, while `surfaceFieldValue` with the `areaAverage` operation extracted pressure and

velocity at the inlet, intermediate locations, throat inlet, throat outlet, and outlet.

The main time-control settings were an end time of 5 s, adaptive time stepping with a maximum Courant number of 0.3, and output at the specified write interval. The extracted area-averaged data were subsequently used to construct the mesh-independence and geometry-comparison plots. This procedure was repeated consistently for all five mesh levels and for the four final channel geometries.

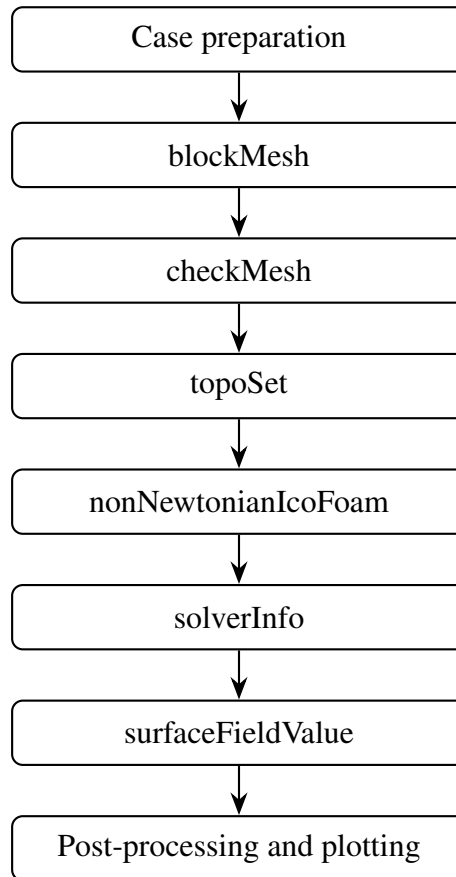


Figure 6: Simulation workflow.

6 Results

6.1 Flow field visualization

The flow field inside the converging-diverging channel was studied using contour plots generated from the numerical results obtained in OpenFOAM v2012. These visualizations helped examine how velocity, pressure, and dynamic viscosity of the shear-thinning fluid varied spatially throughout the domain.

Special attention was given to the converging, throat, and diverging regions, since these experience the most significant changes in cross-sectional area — and consequently, in flow velocity, pressure, and local shear conditions.

While the contour plots give a good qualitative picture of the overall flow behaviour, area-averaged values were also extracted at several locations along the channel — the inlet, intermediate sections, throat inlet, throat outlet, and outlet — to allow a more quantitative look at how each variable changed along the channel length.

6.1.1 Velocity

The velocity field was examined to understand how converging-diverging channel geometry affects the flow behaviour of the shear-thinning fluid. All four geometry cases were run with an inlet velocity of 2 m/s, keeping the converging angle fixed at 11° while varying the diverging angle from 4° to 7° .

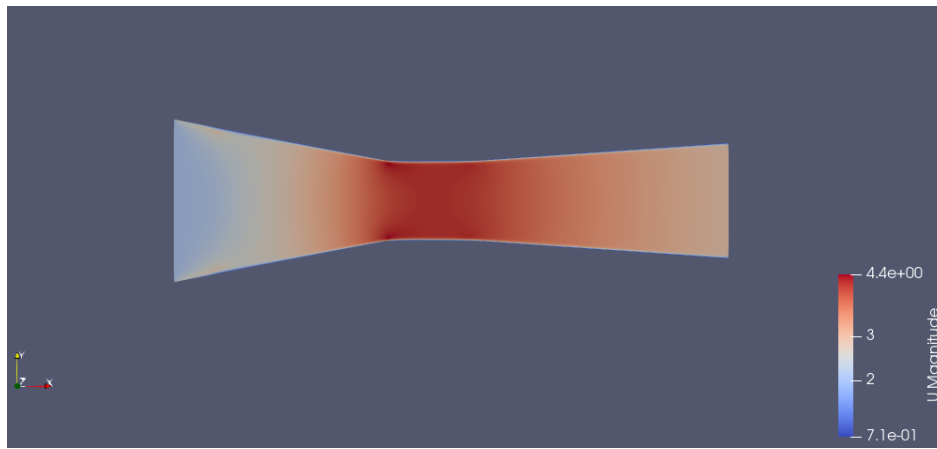


Figure 7: Velocity contour of the converging-diverging channel.

The velocity contour from OpenFOAM gives a good qualitative picture of where the flow accelerates and decelerates within the channel. As the fluid moves into the converging section, the shrinking cross-sectional area pushes it to speed up, with velocity climbing steadily toward the throat — the point of minimum flow area and, correspondingly, maximum velocity. Once past the throat, the fluid enters the diverging section, where the growing flow area causes it to slow back down, with velocity dropping progressively on the way to the outlet.

This velocity pattern isn't shaped by geometry alone — the fluid's non-Newtonian rheology plays a role too. The power-law viscosity model used here, available through OpenFOAM's generalized-Newtonian framework, means local viscosity responds to the shear rate as the flow accelerates and decelerates.

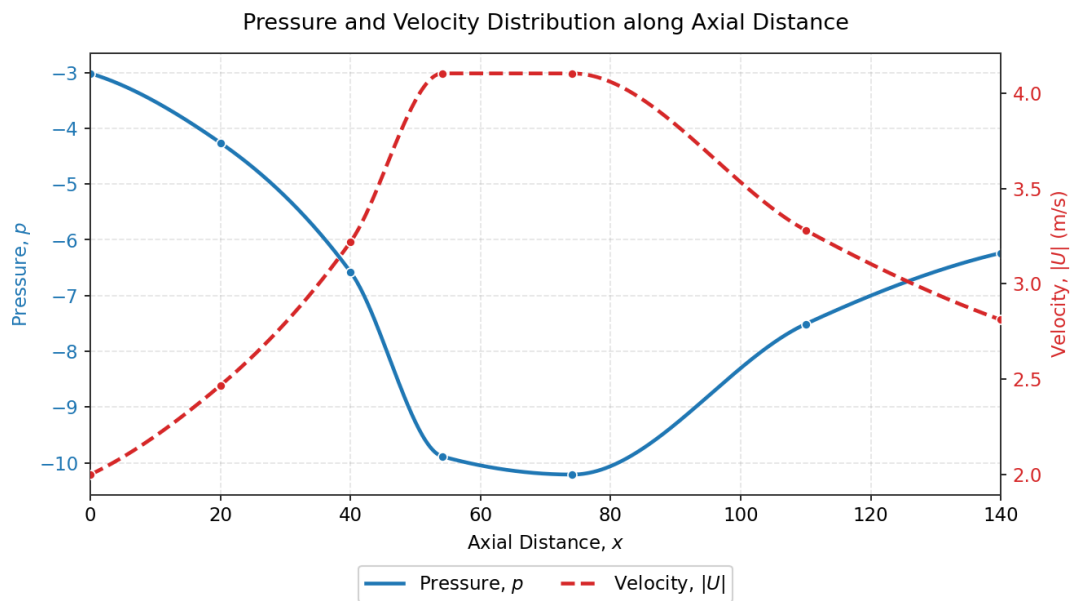
Quantitative velocity results. To put numbers to this, area-averaged velocities were extracted at the inlet, throat inlet, throat outlet, and outlet for all four diverging-angle configurations, summarized in Table 4.

Table 4: Geometrical parameters and area-averaged velocity characteristics for different diverging angles.

Case	Conv. angle (°)	Div. angle (°)	Inlet height	Outlet height	Inlet vel. (m/s)	Throat-in vel. (m/s)	Throat-out / Outlet vel. (m/s)
Case 1	11	4	41.04	29.216	2	4.103195	4.102923 / 2.811417
Case 2	11	5	41.04	31.526	2	4.103195	4.102664 / 2.644476
Case 3	11	6	41.04	33.857	2	4.103195	4.102399 / 2.464140
Case 4	11	7	41.04	36.222	2	4.103195	4.102150 / 2.305928

As expected, inlet velocity holds steady at 2 m/s across all cases, while velocity rises sharply through the converging section to reach roughly 4.10 m/s at the throat. The throat inlet and outlet velocities stay very close to one another in every case, confirming that peak velocity is concentrated right around the throat region.

Where things get more interesting is at the outlet, which shows a clear dependence on diverging angle. At 4° , the outlet velocity comes out to 2.811417 m/s; by 7° , it has dropped to 2.305928 m/s. This drop tracks directly with the increase in outlet height as the diverging angle widens — from 29.216 in Case 1 up to 36.222 in Case 4. In other words, a larger diverging angle opens up the flow passage more, and the average outlet velocity falls accordingly.

**Figure 8:** Area-averaged velocity and pressure along the channel for the four diverging-angle cases, shown using a dual-axis representation.

All four cases follow the same basic pattern — velocity rises from the inlet as the channel converges, peaks near the throat, then falls off as the channel expands again. Where they differ is downstream of the throat: the larger the diverging angle, the more pronounced the drop in velocity toward the outlet. Case 1 (4°) ends up with the highest outlet velocity of the group, while Case 4 (7°) has the lowest.

6.1.2 Pressure

The pressure distribution inside the converging-diverging channel was studied to understand how channel geometry affects the pressure behaviour of the shear-thinning fluid. As with the velocity study, all four cases used a constant inlet velocity of 2 m/s, with the converging angle fixed at 11° and the diverging angle varied from 4° to 7° .

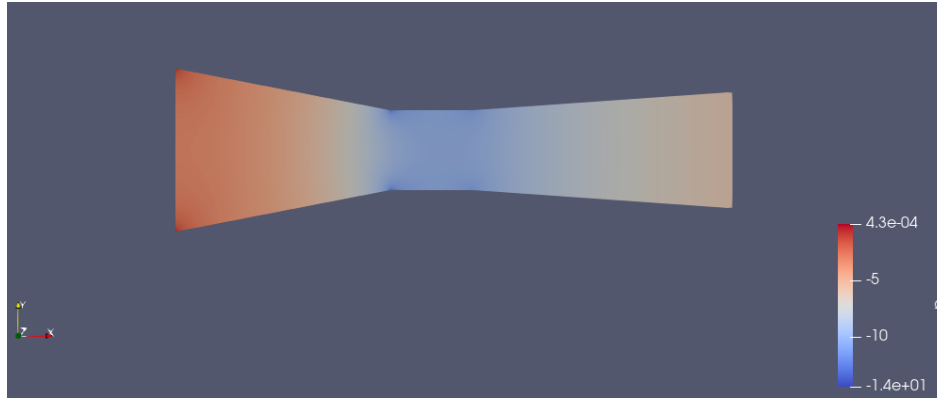


Figure 9: Pressure contour of the converging-diverging channel.

As the fluid accelerates through the converging section, pressure drops steadily in the downstream direction, bottoming out around the throat — which makes sense, since this is also where velocity peaks. Beyond the throat, as the channel starts to diverge, the flow area increases and the fluid decelerates — and with that deceleration comes pressure recovery, building back up toward the outlet.

Table 5: Geometrical parameters and area-averaged pressure characteristics for different diverging angles.

Case	Div. angle ($^\circ$)	Outlet height	Inlet pressure (Pa)	Throat-in pressure (Pa)	Throat-out pressure (Pa)	Outlet pressure (Pa)
Case 1	4	29.216	-3.014463	-9.887261	-10.21174	-6.236355
Case 2	5	31.526	-3.014456	-9.887287	-10.20297	-5.645865
Case 3	6	33.857	-3.014463	-9.887259	-10.19510	-5.170311
Case 4	7	36.222	-3.014466	-9.887260	-10.18870	-4.781753

Inlet pressure stays essentially constant at around -3.014 Pa across all four cases, then drops sharply to about -9.887 Pa at the throat inlet. The throat outlet pressure dips slightly further still, ranging from -10.212 Pa down to -10.189 Pa.

Interestingly, the lowest throat outlet pressure shows up in Case 1 (4° diverging angle), and this pressure becomes progressively less negative as the diverging angle increases — a first hint that the diverging geometry is already influencing pressure behaviour right at the throat exit. The effect becomes much more pronounced at the outlet, where pressure rises from -6.236355 Pa in Case 1 to -4.781753 Pa in Case 4. So widening the diverging angle from 4° to 7° clearly improves pressure recovery, leaving a less negative outlet pressure.

All four cases follow the same general shape: pressure falls from the inlet through the converging section, bottoms out near the throat (right where velocity is highest), then climbs back up through the diverging section as the flow decelerates and recovers. This difference in recovery lines up with the outlet height, which grows from 29.216 to 36.222 as the diverging angle increases — a larger angle means a more gradual expansion, which shapes how effectively pressure recovers.

6.1.3 Dynamic Viscosity

Dynamic viscosity was evaluated to understand how shear rate influences the rheological behaviour of the non-Newtonian fluid. As before, the power-law model was used, with a consistency index of $k = 0.02$ and a flow behaviour index of $n = 0.6$. Since $n < 1$, the fluid is shear-thinning, meaning its effective viscosity drops as shear rate increases — a behaviour OpenFOAM's power-law model captures by calculating kinematic viscosity as a function of local strain rate, bounded within specified minimum and maximum limits.

Dynamic viscosity was calculated at the inlet, throat inlet, throat outlet, and outlet, and the results are summarized in Table 6. The results show viscosity dropping sharply from the inlet to the throat in all four cases — from roughly 67.77 Pa·s at the inlet down to about 46.00 Pa·s at the throat inlet. This makes sense given the shear-thinning behaviour: as the fluid accelerates through the converging section, the rising shear rate pulls the viscosity down with it.

Table 6: Dynamic viscosity at selected locations for different diverging angles.

Case	Inlet μ (Pa·s)	Throat-in μ (Pa·s)	Throat-out μ (Pa·s)	Outlet μ (Pa·s)
Case 1 (4°)	67.7674	46.0022	70.7218	90.5106
Case 2 (5°)	67.7674	46.0028	64.9042	93.9336
Case 3 (6°)	67.7674	46.0030	60.4846	93.6056
Case 4 (7°)	67.7675	46.0035	57.0581	92.6413

Interestingly, the throat-inlet viscosity barely changes across the four cases — from 46.0022 Pa·s in Case 1 to 46.0035 Pa·s in Case 4. That is not too surprising, since the converging geometry and inlet conditions are identical for all cases; the diverging angle simply hasn't come into play yet at this point.

The real differences show up at the throat outlet. Here, viscosity falls from 70.7218 Pa·s for the 4° case down to 64.9042, 60.4846, and 57.0581 Pa·s for the 5°, 6°, and 7° cases respectively — so a larger diverging angle consistently produces a lower viscosity right at the throat exit. Moving to the outlet, viscosity climbs back up across all four cases, reaching 90.5106, 93.9336, 93.6056, and 92.6413 Pa·s for Cases 1–4. This rise reflects the lower shear rate further downstream, once the flow has decelerated through the diverging section.

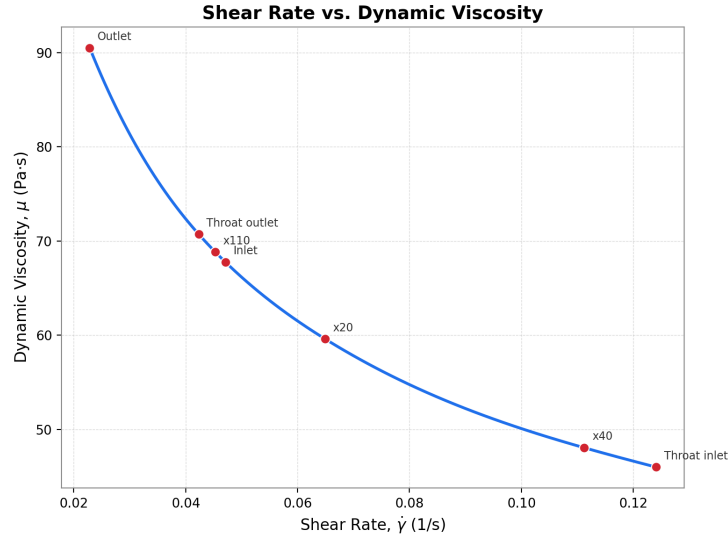


Figure 10: Variation of dynamic viscosity with equivalent shear rate.

This shear rate–viscosity plot captures the fluid’s rheological behaviour directly, showing the expected inverse relationship — viscosity is highest where shear rate is lowest, and drops as shear rate rises. This is exactly what the power-law model predicts, and it confirms the fluid’s shear-thinning nature. The highest shear rates occur near the throat inlet, which is exactly where viscosity bottoms out; as shear rate falls again downstream of the throat, viscosity climbs back up.

All four cases follow this same overall shear-thinning trend, though they diverge somewhat at the throat outlet and outlet, where the diverging angle starts to matter. The 7° case gives the lowest throat-outlet viscosity of the group, while the 4° case gives the highest. Altogether, these results make clear just how tightly dynamic viscosity is tied to local shear rate — the fluid thins out under higher shear and thickens back up as shear drops. This shear-dependent behaviour is really the defining characteristic of the non-Newtonian fluid studied here.

7 Conclusion

A two-dimensional converging-diverging channel was successfully simulated using OpenFOAM v2012 and the nonNewtonianIcoFoam solver for transient, incompressible, laminar non-Newtonian flow. The mesh-independence study considered five mesh levels and showed that the difference in throat pressure between the Very Fine and Very Very Fine meshes was approximately 0.78%. The Very Fine mesh containing 194,400 cells was therefore selected for the subsequent geometry study.

For the four final geometries, the inlet velocity was fixed at 2.0 m/s and the converging angle at 11° , while the diverging angle was varied from 4° to 7° . The area-averaged velocity increased through the converging section and reached approximately 4.10 m/s near the throat before decreasing in the diverging section. The outlet velocity decreased from 2.811417 m/s

for the 4° case to 2.305928 m/s for the 7° case.

Pressure decreased through the converging section, reached its lowest values near the throat, and recovered downstream. The outlet pressure changed from -6.236355 Pa to -4.781753 Pa as the diverging angle increased from 4° to 7° . The dynamic-viscosity results also showed the expected shear-thinning response, with viscosity decreasing from approximately 67.77 Pa·s at the inlet to about 46.00 Pa·s near the throat as the local shear rate increased. Overall, the results demonstrate the coupled influence of channel geometry and shear-dependent viscosity on the predicted pressure, velocity, and rheological behaviour of the fluid.

8 Limitations of the Study

The present study uses a two-dimensional approximation of the channel. Consequently, three-dimensional effects, secondary flows, and spanwise variations are not represented. The simulations also use a laminar formulation, so possible transition to turbulence at higher flow rates is outside the scope of the present model.

The power-law rheological model with $n = 0.6$ represents shear-thinning behaviour but may not reproduce every rheological feature of a real polymer solution over a wide shear-rate range. Only four diverging angles, from 4° to 7° , were investigated while the converging angle was fixed at 11° ; therefore, the results describe only this selected geometry range. Velocity, pressure, and viscosity were evaluated at selected cross-sections rather than at every location in the domain.

Finally, the study is numerical and does not include experimental validation. The predicted results may also be sensitive to mesh resolution, time-step control, boundary conditions, convergence criteria, and the selected rheological parameters. Future work could extend the model to three dimensions, investigate a wider range of geometries and operating conditions, evaluate additional rheological models, and compare the numerical results with experimental measurements.

9 References

1. OpenCFD Ltd., *OpenFOAM v2012: New and updated solvers and physics*, OpenFOAM, 2020. Available at: <https://www.openfoam.com/news/main-news/openfoam-v20-12/solver-and-physics>
2. OpenCFD Ltd., *OpenFOAM v2012 API Guide: powerLaw Generalized Newtonian Viscosity Model*, OpenFOAM v2012 Documentation, 2020.
3. OpenFOAM Foundation, *OpenFOAM User Guide – Power Law Model*, OpenFOAM Documentation. Available at: <https://doc.cfd.direct/openfoam/user-guide-v7/transport-rheology>
4. R. T. Balmer and J. J. Kauzlaich, “Similarity solutions for the converging or diverging steady flow of non-Newtonian elastic power law fluids with wall suction or injection. Part I: Two-dimensional channel flow,” *AIChE Journal*, vol. 17, no. 5, pp. 1181–1188, 1971. doi: 10.1002/aic.690170525.
5. Y. Mansutti and K. R. Rajagopal, “Theoretical analysis of non-Newtonian jets emanating from a converging channel,” *Computers & Fluids*, vol. 27, no. 2, pp. 255–271, 1998.
6. C. Harley, E. Momoniat, S. V. Meleshko and K. R. Rajagopal, “Reversal of flow of a non-Newtonian fluid in an expanding channel,” *International Journal of Non-Linear Mechanics*, vol. 101, pp. 44–55, 2018. doi: 10.1016/j.ijnonlinmec.2018.02.006.
7. OpenCFD Ltd., *OpenFOAM User Guide – Standard Solvers*, OpenFOAM Documentation. Available at: <https://www.openfoam.com/documentation/user-guide/a-reference/a.1-standard-solvers>
8. A. Kulkarni, A. Sawant, P. Kulkarni, and D. Nalawade, “CFD Analysis of Air Flow Through Venturi,” *Journal of Emerging Technologies and Innovative Research (JETIR)*, vol. 11, no. 5, pp. e359–e364, May 2024, Paper ID: JETIR2405442.

10 Appendix

A blockMeshDict: Geometry and Mesh Definition

The following listing contains the blockMeshDict geometry and block definitions documented in the supplied case report. It has been moved from the main body of the report so that the computational-domain discussion remains concise.

```
scale 1;

vertices
(
    // bottom line
    ( 0 -20.520 0 ) //0
    (20 -16.632 0 ) //1
    (40 -12.745 0 ) //2
    (54.12 -10.000 0 ) //3
    (74.12 -10.000 0 ) //4
    (110 -12.510 0 ) //5
    (140 -14.608 0 ) //6

    // top line
    ( 0 20.520 0 ) //7
    (20 16.632 0 ) //8
    (40 12.745 0 ) //9
    (54.12 10.000 0 ) //10
    (74.12 10.000 0 ) //11
    (110 12.510 0 ) //12
    (140 14.608 0 ) //13

    // back plane
    ( 0 -20.520 1 ) //14
    (20 -16.632 1 ) //15
    (40 -12.745 1 ) //16
    (54.12 -10.000 1 ) //17
    (74.12 -10.000 1 ) //18
    (110 -12.510 1 ) //19
    (140 -14.608 1 ) //20

    ( 0 20.520 1 ) //21
    (20 16.632 1 ) //22
    (40 12.745 1 ) //23
    (54.12 10.000 1 ) //24
    (74.12 10.000 1 ) //25
    (110 12.510 1 ) //26
    (140 14.608 1 ) //27
```

```
);  
  
blocks  
(  
  hex (0 1 8 7 14 15 22 21) (180 180 1)  
  simpleGrading (1 1 1)  
  
  hex (1 2 9 8 15 16 23 22) (180 180 1)  
  simpleGrading (1 1 1)  
  
  hex (2 3 10 9 16 17 24 23) (180 180 1)  
  simpleGrading (1 1 1)  
  
  hex (3 4 11 10 17 18 25 24) (180 180 1)  
  
  simpleGrading (1 1 1)  
  
  hex (4 5 12 11 18 19 26 25) (180 180 1)  
  simpleGrading (1 1 1)  
  
  hex (5 6 13 12 19 20 27 26) (180 180 1)  
  simpleGrading (1 1 1)  
);
```

Note: the supplied report text did not contain the remaining blockMeshDict entries, such as patch-boundary definitions, if any. Those entries should be appended from the actual case file before submission if they are part of the implemented dictionary.