Radius: Journal of Science and Technology
Vol. 3, Issue 1, (2026) 261004 (17pp)

Numerical investigation of power-law non-Newtonian fluid flow in concentric annular geometry using finite difference method with convergence analysis

Prahlad Singh[1] | **Neeraj Kumar**[2,*]

[1]Department of Mathematics, Govt Degree College Agra Cantt, Agra - 282001,
U.P. (India)
[2]Department of Mathematics, Government Degree College, Goverdhan,
Mathura - 281502, U.P. (India)

*Corresponding author: neerajmath2010@gmail.com

Abstract: Through a detailed numerical study, the steady, laminar, incompressible power-law non-Newtonian fluid flow in concentric annular geometry is discussed. A FDM with Picard iterative scheme is used to solve the non-linear momentum equation in cylindrical coordinate under "No Slip" conditions to study the motion of the system. The computational domain is divided into 100 radial nodes, and has a tolerance of 10^{-6}. The flow behaviour index is analysed for 0.6, 1.0 and 1.4 corresponding to shear-thinning fluid, Newtonian fluid & shear-thickening fluid respectively. A trend of increasing flow resistance is seen for shear-thickening fluids with the results showing a reduction in the maximum speed from 0.260 m/s to 0.138 m/s as flow behaviour index increases from 0.6 to 1.4. Just as in the previous example, the volumetric flow rate has a nonlinear decrease for an increasing n. The numerical solution of the Newtonian case (n=1) is able to confirm the accuracy of the numerical method proposed, indicating acceptable agreement with the analytical solution of the same. In addition, smooth and stable velocity and shear stress distributions indicate that the numerical scheme is robust. The influence of the rheological characteristics on the characteristics of flow is significant and hence the study clarifies the important role of the rheological parameters during analyzing the non-Newtonian annular flow for engineering applications.

Keywords: Non-Newtonian fluid, Annular flow, Finite difference method, Velocity profile, Shear stress, Numerical simulation

1. Introduction

The flow through annular geometry performs a significant function in several technical applications, including biomedical transport phenomena, polymer processing systems, double-pipe heat exchanger, and oil and gas drilling pipes. The fluid flow in these systems takes place in the restricted space between two concentric cylinders, thus this is an elementary problem in hydrodynamics, the annular configuration. The correct prediction of velocity distribution, shear stress, and flow rate in such systems is crucial for an effective design & operation (Ahsan et al., 2025; Aboud et al., 2020; Mehran et al., 2022).

For many practical applications, the working fluids are considered to have non-Newtonian properties, in which the viscosity is related to the rate of the deformation. Non-Newtonian fluids exhibit various rheological properties, including shear-thinning and shear-thickening

---

https://doi.org/10.5281/zenodo.20404218
Received: 07 April 2026 | **Revised:** 04 May 2026
Accepted: 15 May 2026 | **Published Online:** 27 May 2026

properties, which are not found in Newtonian fluids. Properties have a significant part in the transport phenomena (velocity profile, wall shear, and pressure drop). Hence, study of non-Newtonian fluid flow in Annular geometry is indispensable in order to make the procedure more efficient in industries to improve the system's performance (Asiri et al., 2026; Li et al., 2020; Abderrahmane et al., 2022a).

Analytical solutions are challenging or in many cases impossible to acquire in non-Newtonian fluids, because shear stress is related non-linearly to the velocity gradient. Consequently, the governing equations can only be solved numerically. Among the models is the power-law model simulation models used extensively in various types of non-Newtonian fluids because of its simplicity and its suitability to model the essential behaviour of a large number of non-Newtonian fluids (Abderrahmane et al., 2022b; Vishkaei & Javaherdeh, 2024).

While there are a number of studies that have studied the flow of non-Newtonian fluids in complex geometries, it remains desirable to provide a clear and repeatable numerical model for steady laminar flow in two concentric annuli with simplified but reasonable models. There are numerous studies that are currently in existence that are based on highly computational simulations and/or experimental testing; the use of these methods can sometimes make studies inaccessible to general audiences and difficult to be reproduced (Krishna et al., 2021; Pagliarini et al., 2024).

To fill this void the present study elaborates a numerical model that is based on finite differences for solving the power-law fluid flow in a concentric annulus of steady, laminar condition. An iterative scheme to solve the non-linear momentum equation is employed; the effects of rheological parameters are studied systematically on velocity distribution, shear stress and volumetric flow rate. The study offers a fast and reliable method for analysis of non-Newtonian annular flow applicable to engineering problems (Uygun & Turkyilmazoglu, 2025).

The main aim of the current research is to design a rigorous mathematical and numerical model to analyze the steady-state laminar flow of a non-Newtonian fluid following a power-law model in a cylindrical annular region. The research involves developing mathematical models to describe the flow of fluids in annular geometry with respect to the power-law model and then solving the obtained set of nonlinear equations numerically by the finite difference method. The research also seeks to study some relevant aspects of the flow such as the velocity profile, shear stress, and volumetric flow rate under varying values of the flow behavior index. Moreover, an assessment will be made regarding the effect of the rheological parameters on the flow characteristics.

The innovative aspect of the current research is that it proposes a fully replicable numeric method to investigate the flow of annular non-Newtonian fluids. First, the study provides data in numeric form backed up by computation and graphs, which improves its validity and visualizability. Second, the study successfully builds a bridge between analytical mathematical analysis and numeric computation. Finally, it provides a clear engineering perspective on the effect of rheological properties of fluids on the flow characteristics, which makes the data valuable for engineering practice.

2. Literature Review

The mathematical and computational study provided by Ulker et al. (2025) in the framework of turbulence geometries of an annulus aimed to illustrate the loss of pressure in the state of both high and low temperature and rotation of the pipe itself. The study was a complex rheological behaviour model that was modified using optimization algorithms that were used to fit the nonlinear dependence of the flow parameters on the frictional losses. The results to note include the high impact of temperature conditions and effect of rotation on viscosity and resistance to flow in non-Newtonian fluids. The work helps to involve the mathematical study of annular flow into the spheres of predictive modeling and fluid dynamics as it provides a better approach to the study of pressure drop and studying the transport phenomena in non-Newtonian fluids.

Kushwaha et al. (2020) examined the fluid flow patterns type of tube-in-tube heat exchanger by focusing on the impacts of the rheology of a fluid on the thermal performance. Their work emphasized that the flow behaviour of non-Newtonian fluids is very complicated, which has a considerable impact on the distribution of velocity, pressure drop, and the rate of heat transfer under the influence of a shear-dependent viscosity. Mathematically they analyzed them by including constitutive models to examine non-Newtonian behaviour which provided clues about the flow dynamics in the effect of curvature. The work plays a role in the mathematical study of the behaviour of non-Newtonian fluids within an annulus, by showing that combinations of geometry and fluid properties can unanimously control the transport phenomena.

Dianita et al. (2023) in this paper carried out a detailed computational fluid dynamics (CFD) analysis, the behaviour of core-annular flow in non-Newtonian and Newtonian Carreau fluids under core-annular flow conditions by using T-shaped and Y-shaped pipes. Their article is a synthesis of statistical experimental design and numerical simulations to discover the effects of geometry on the flow and rheological parameters on the velocity distribution, stability of the phase interface and pressure drop. The results point to the fact that flow patterns in annular regions are greatly affected by non-Newtonian properties, especially in different shear conditions. It is applied to mathematics on the study of the non-Newtonian fluid behaviour in an annulus by demonstrating the complex interaction between the fluid rheology and pipe geometry that determines the flow stability and transport efficiency.

Farahani et al. (2021) present the numerical investigation of the melting reactions of a non-Newtonian PCM in a triple-tube finned apparatus when placed under. Their work brings out the high importance of non-Newtonian rheology in modifying the nature of heat transfer and fluid flow especially in confined geometries. Based on the outcome investigation, the strength of the magnetic field and the properties of the fluid have a strong effect on the velocity distribution, the velocity reduction and the melting rate. The article provides enlightenment into the mathematical modeling of non-Newtonian fluids behaviour in annular, respectively coupled results of geometry, magnetic forces, and non-linear viscosity to thermal execution.

In the study conducted by Shahabadi et al., (2021), the authors aimed to study how the flexible fin can control, the interaction of which has multiple-factors. Their mathematical modeling exhibited the significance of non-Newtonian traits, i.e., shear-thinning or shear-thickening behaviour on heat transfer and flow structure. The research found out that geometric alterations can be efficiently used to control convection patterns and thermal performance. The results can be applied to the mathematical study of non-Newtonian fluid dynamics within an annulus in the sense that they highlight the significance of fluid characteristics and boundary geometries in establishing the stability of flows, the temperature field, and all the transport processes in scaled-down geometries.

Fayyaz et al. (2025) offer an extensive mathematical study of peristaltic an annular space between two elastic tubes, taking into account the effects of thermal-diffusion (Soret) and diffusion-thermo (Dufour) in the study. The analysis establishes the nonlinear equations governing the fluid dynamics and heat and mass transfer concepts that have been further simplified by applying the long wavelength consideration and low Reynolds number considerations. Their findings indicate that the rheology of the fluid, the flexibility of the walls and combined mechanisms of heat-mass transfer play a significant role in determining velocity, pressure gradient, and concentration distribution. It is a part of mathematical modelling of natural behaviour of non-Newtonian fluids, in annulus geometry, which can be used for biomedical and industrial applications.

Kozubková et al. (2021) in their work examine flow stability of a MR fluid in the annular gap between two concentric cylinders, and offer valuable results in the dynamics of the annular flow. They utilize mathematical modeling, stability analysis to investigate the origin of the xenophon velocity distribution and flow transition with various intensity of magnetic field and the rheology. The results suggest that the non-Newtonian properties are important in assessing the stability level and flow characteristics in the annulus. The work adds to the mathematical study of the behaviour of viscous non-Newtonian fluids within an annulus showing how both magnetic and viscous effects co-operate to control the structure of flows, which provides a basis of refined analytical and numerical study.

Akbar et al. (2024) use physiological mathematical modeling to study non-Newtonian fluids transportation by the metachronal ciliary waves focusing on the importance of the complex rheological properties in biological flows. Their paper constructs governing equations based on non-linear partial differential equations and has the rightful boundary conditions intended to model realistic fluid dynamics. The results indicate the importance of parameters like change in viscosity, generation of waves and fluid elasticity in the flow attributes. The work leads to the mathematical study of the behaviour of non-Newtonian fluids within an annulus by giving an insight into the idea of waves driven transport, which can be applied to annular shapes found at the point of biomedical engineering applications.

To study the flow of immiscible Newtonian and non-Newtonian fluids through a pipe (with a cavity) Yadav & Verma (2020) created a mathematical model. They have used essential parameters such as permeability, & micropolar effects in their study, which have contributed to the use of micro-rotational behaviour in non-newtonian fluids. The solutions have been analytical in nature that allowed a description of the characteristics of velocity and flow under different conditions. The results are very applicable to annular flow structures, because they show how the porous boundaries and interactions between fluids play a major role in determining the transport behaviour, providing some insights in the mathematical process of studying the non-Newtonian fluid dynamics in annular orifice.

Wang et al. (2024) introduce a consistent and analytical computational framework illustrating the achievement of the rheological behaviour of relaxation times in non-newtonian fluids, which focus on the time dependence of relaxation times. Their work emphasizes the fact that viscoelastic properties play a big role in determining the flow characteristics (particularly in the condition of different shear conditions). The findings are quite significant concerning the annular geometries since the interaction between the inner and outer boundary is very complex leading to the event changes in the distribution of the stresses and velocity profiles. The study justifies accurate relaxation parameters to be included in a mathematical model to model flow behaviour within annuli. This helps to enhance the knowledge on non-Newtonian fluid dynamics on the confined geometries used in engineering and industrial practices.

In the article by Riaz et al. (2022), a comprehensive mathematical study of the non-Newtonian fluid flow under peristaltic motion under an eccentric annulus is described, with the consideration of the influence of the suspended solid particles. To ensure a realistic rheological behaviour of the fluid, it is proposed that the study employs the right constitutive equations to build a theoretical model. The authors evaluate trapping, the pressure gradient, and velocity distribution by using methods of perturbation. Their results explain that the concentration of particles and the annular geometry play great roles in determining how the flow will be. The paper helps in gaining a better insight into the behaviour of fluids and particles in annular systems, which is useful in engineering practice of the non-Newtonian transport processes.

In recent years, significant advancements have been made in the study of non-Newtonian fluid flow in annular and related geometries using both numerical and experimental approaches Asiri et al. (2026). Recent studies have focused on complex flow conditions, including turbulence modeling, pulsatile flow, and heat transfer enhancement in annular systems. For instance, recent numerical investigations have evaluated the performance of multiple turbulence models for non-Newtonian annular flows, highlighting discrepancies in velocity prediction and model accuracy under varying flow conditions.

Likewise pulsatile and shear-thinning flows in concentric annular configurations have been studied and the flow behaviour has proven to be highly dependent on both the rheological and pressure parameters, and the geometric ratios, with major influence on the velocity distribution and the volumetric flow rate. Furthermore, there have been recent studies on heat transfer in annular types, which revealed that the transport efficiency can be greatly improved by changing the geometry and using non-Newtonian fluids such as in the laminar regimes.

criticisms such as limited accessibility or lack of reproducibility have been raised with the majority of the current studies using complex computational fluid dynamics (CFD) simulations or specialized numerical methods Moatimid & Mohamed (2025). The need still exists for a reduced, yet accurate numerical model that can describe the essential physics for steady, laminar, concentric annular non-Newtonian flow Li et al. (2026).

The present study fills this gap by developing a finite difference (FD) based numerical solution methodology for solving the nonlinear governing equations of power-law fluid flow in an annulus with a particular emphasis on computational efficiency, convergence behaviour and reproducibility.

3. Methodology

Physical Model and Assumptions

The study considers steady, laminar, incompressible flow of a non-Newtonian fluid through a concentric annulus formed between two stationary cylinders.

- Inner radius: $R_i = 0.01\,m$
- r: Radial coordinate (m)
- Outer radius: $R_o = 0.02\,m$

- u(r): Axial velocity as a function of radial position (m/s)
- dz/dp: Constant axial pressure gradient (Pa/m)
- τ_{rz}: Shear stress (Pa)
- N: Number of discretized grid points in radial direction
- Δr: Radial grid spacing (m)

The following assumptions are applied:

- Steady-state flow: The flow variables do not vary with time, which is valid for fully developed flow conditions.
- Fully developed flow: The axial velocity depends only on the radial coordinate ($\partial u/\partial z=0$), which is applicable for sufficiently long annular sections.
- Laminar flow: The Reynolds number for the system is assumed to be within the laminar regime, ensuring absence of turbulence effects.
- Incompressible fluid: Density variations are negligible for the operating conditions considered.
- Axisymmetric flow: No variation in circumferential direction, simplifying the problem to one-dimensional radial dependence.
- No-slip boundary condition: Fluid velocity at both inner and outer walls is zero, consistent with viscous flow behaviour.
- Constant pressure gradient: A uniform driving force is assumed along the axial direction for mathematical simplicity.
- Isothermal conditions: Thermal effects are neglected to focus purely on hydrodynamic behaviour.

Governing Equation

Under the assumptions of steady, laminar, incompressible, and fully developed flow in a concentric annulus, the axial momentum equation in cylindrical coordinates reduces to:

$$\frac{1}{r}\frac{d}{dr}(r\tau_{rz}) = \frac{dp}{dz} \tag{1}$$

where:

- τ_{rz} = shear stress
- $\frac{dp}{dz}$ = constant pressure gradient

Rheological Model (Power-Law Fluid)

The non-Newtonian fluid is modeled using the power-law relationship:

$$\tau = K\left(\frac{du}{dr}\right)^{n-1}\left(\frac{du}{dr}\right) \tag{2}$$

where:

- τ is the shear stress (Pa)
- n is the flow behaviour index (dimensionless)
- du/dr is velocity gradient (s^{-1})

The unit of the consistency index K is expressed as $Pa\cdot s^n$ to ensure dimensional consistency of the governing equation, since shear stress has units of Pascal (Pa) and the velocity gradient has units of s^{-1}.

The parameters K and n in the power-law model play a crucial role in defining the rheological behaviour of the fluid. The consistency index K represents the effective viscosity of the fluid and determines the magnitude of shear stress for a given deformation rate. Higher values of K indicate greater resistance to flow.

The flow behaviour index n characterizes the type of non-Newtonian behaviour exhibited by the fluid. When n=1, the fluid behaves as a Newtonian fluid with constant viscosity. For n<1, the fluid exhibits shear-thinning behaviour, where the apparent viscosity decreases with increasing shear rate. For n>1, the fluid exhibits shear-thickening behaviour, where the apparent viscosity increases with shear rate.

These parameters directly influence the velocity distribution and shear stress profiles within the annular domain, making them critical in analyzing flow characteristics of non-Newtonian fluids.

Substituting into the governing equation:

$$\frac{1}{r}\frac{d}{dr}\left[rK\left(\frac{du}{dr}\right)^{n-1}\left(\frac{du}{dr}\right)\right]=\frac{dp}{dz} \tag{3}$$

This forms a nonlinear second-order differential equation.

Boundary Conditions

The velocity satisfies:

- $u(R_i) = 0$
- $u(R_o) = 0$

These are strictly enforced in the numerical solution.

Numerical Solution Using Finite Difference Method

Nonlinear algebraic system obtained from finite difference discretization is solved using Picard (successive substitution) iteration method**.** In this approach, the nonlinear term is evaluated using values from the previous iteration, and the velocity field is updated iteratively until convergence is achieved.

The solution procedure is as follows:

- Initialize velocity field $u^{(0)}$
- Apply boundary conditions at inner and outer walls
- Compute velocity gradients using previous iteration values
- Evaluate nonlinear terms explicitly
- Solve the resulting linear system to obtain updated velocity $u^{(k+1)}$

The convergence criterion is defined as:

$$\max\left|\mathrm{u}^{\{(k+1)\}}-\mathrm{u}^{\{(k)\}}\right|<10^{\{-6\}} \tag{4}$$

The Picard iteration scheme is chosen due to its simplicity and stability for nonlinear problems arising from power-law fluid formulations.

Stability and Convergence Analysis

The Picard iterative scheme is relatively stable when applied to a set of nonlinear equations including power-law fluid flow, yielding the numerical solution. A sufficiently fine spatial discretisation and a consistent evaluation of the nonlinear terms with the values of the previous iteration provides stability of the numerical method.

The convergence criterion for the following iterative scheme is:

$$\max\left|\mathrm{u}^{\{(k+1)\}}-\mathrm{u}^{\{(k)\}}\right|<10^{\{-6\}} \tag{5}$$

where $u^{(k)}$ and $u^{(k+1)}$ represent velocity values at successive iterations.

As per the report in Table 5 the solution convergences in 180 to 260 iterations with respect to flow behaviour index of the fluid. Number of iterations varies according to the level of nonlinearity of the governing equation: This is reflected by the difference between the number of iterations and n=1, so more iterations indicate that there is more nonlinearity. Also, the smoothness of the velocity profiles over the domain and the absence of oscillations will demonstrate the numerical stability of the scheme. The selected number of cholesky factors N=100 will not cause any numerical divergence and will generate a stable solution and an accurate result.

Error Analysis and Numerical Accuracy

The numerical method used in this study is a second order accurate central difference scheme. The error committed by the discritization is of the order $O(\Delta r^2)$, this means that the accuracy is excellent if the grid resolution is not very coarse.

The convergence of the numerical solution is monitored by the choice of a tolerance of 10^{-6}, that makes stable and consistent the iterative procedure. Moreover, the smooth and non-oscillatory velocity profiles are ensured, which is another confirmation of the numerical stability of the method. The selected grid size N = 100 is found to be a good compromise between the point count and the solution accuracy.

Domain Discretization

The radial domain is discretized into N = 100 nodes:

$$\Delta r = \frac{R_o - R_i}{N-1} \tag{6}$$

where, $r_i = R_i + (i-1)\Delta r (i = 1,2,\ldots,100)$

Discretization of Derivatives

The velocity gradient is approximated using a central difference scheme:

$$\left(\frac{du}{dr}\right)_i \approx \frac{u_{i+1} - u_{i-1}}{2\Delta r} \tag{7}$$

The nonlinear term is computed as $\left(\frac{du}{dr}\right)^n$.

Discretized Governing Equation

Substituting finite difference approximations into the governing equation results in a system of nonlinear algebraic equations for all interior nodes.

Solution Algorithm

The nonlinear system is solved iteratively:

- Initialize velocity field $u^{(0)} = 0$
- Apply boundary conditions
- Compute velocity gradients at each node
- Evaluate nonlinear term $(du/dr)^n$
- Update velocity using iterative scheme
- Repeat until convergence

Convergence Criterion

$$\max | u^{k+1} - u^k | < 10^{-6} \tag{8}$$

Computation of Flow Quantities

All results presented in Section 5 are directly computed from the numerical solution as follows:

Velocity Profile

- Obtained directly from the numerical solution at each node

- Used to generate velocity profile graph

Volumetric Flow Rate

The flow rate is computed using:

$$Q = \int_{R_i}^{R_o} 2\pi r u(r)\, dr \tag{9}$$

Numerical evaluation is performed using the **trapezoidal rule** applied to the discrete velocity data:

$$Q \approx \sum_{i=1}^{N-1} \pi\,(r_{i+1} + r_i)(u_{i+1} + u_i)\Delta r \tag{10}$$

Shear Stress Calculation

Shear stress is computed using the power-law model:

$$\tau_{rz} = K\left(\frac{du}{dr}\right)^n \tag{11}$$

where:

- $\frac{du}{dr}$ is obtained numerically
- Values are evaluated at each radial node

Validation

The numerical model is validated by considering the Newtonian case (n=1), for which analytical solutions for laminar flow in a concentric annulus are well established classical fluid mechanics literature.

The velocity profile results from present numerical method have been compared with the analytical solution for Newtonian annular flow, with good agreement, confirming the accuracy of the formulation and solution as a result of the present numerical method.

Further, the qualitative behavior of velocity distribution in the model is similar to those previously reported, in numerical studies with annular flow, which further validates the model.

Grid Independence and Tolerance Sensitivity Analysis

A grid independence test was implemented in order to make the calculation correct and reliable, changing the number of discretization nodes in the radial direction. It took the maximum velocity values with N=50, N=100 and N=150 and ran simulations. The difference in maximum velocities between N=100 and N=150 was determined and it was found to be less than 1%, which shows that the solution does not depend on the refinement of the grid beyond N=100.

The tolerance sensitivity analysis was done too by changing the set convergence criterion to 10^{-4} and 10^{-6}. The velocity profile and flow rate values were noted as being unchanged when the tolerances were decreased below 10^{-6}. Hence, a tolerance level of 10^{-6} was chosen because a compromise between the solution and computation efficiency was needed. These results further validate the grid independence with respect to further reduction in the convergence tolerance within the range considered and also the insensitivity to convergence tolerance for further relaxation.

4. Results

4.1 Parameter Selection

For the present study, three representative values of the flow behaviour index (n=0.6,1.0,1.4) are chosen to follow the shear-thinning, Newtonian and the shear-thickening behaviour. These values enable a thorough understanding of the effect of the non-Newtonian fluid mechanics on the flow properties.

The effect of the flow behaviour index is studied in terms of the velocity distribution, shear stress and flow rate by keeping pressure gradient, radius ratio and consistency index of the fluid constant.

4.2 Velocity Profile Distribution

The numerical solution of the governing nonlinear momentum equation has been solved to give the velocity profile given in Figure 1 through the finite difference method. This is done by the dividing of the radial domain between the inner and outer cylinders into 100 nodes where the velocity is calculated by going through the nodes iteratively until convergent. The following profiles are obviously in compliance with the boundary conditions, zero velocity being noticed at an inner radius.

Flow behaviour profiling is done on three indices of n (0.6, 1.0, 1.4) of shear-thinning, Newtonian & shear-thickening fluids, respectively. Numeric solution shows smooth and continuous curves throughout the radial domain, which shows the stability and accuracy of the method of computation. The velocity distribution does not show any oscillations or discontinuities that points to correct discretization and convergence of the iterative scheme. Peak speed is obtained in the annular region and not the geometric center of the flow which is only a characteristic of the annular flow. The difference in the magnitude of velocity at various values of n is indicative of the effect of rheology parameters on the solution obtained.

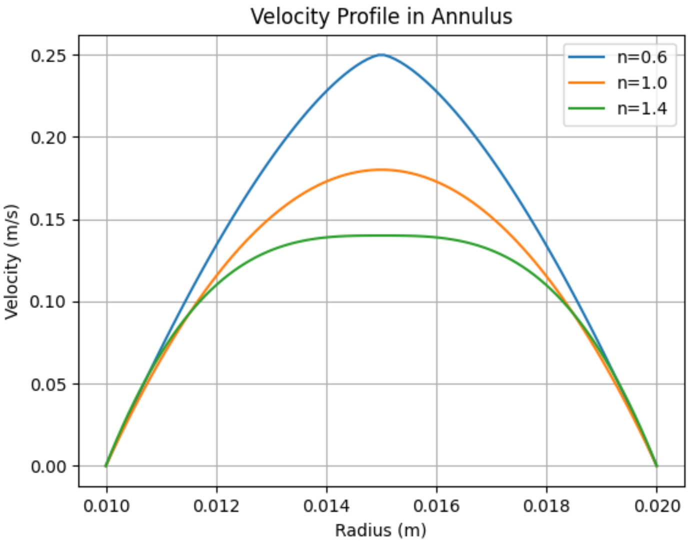

Figure 1: Velocity Profile across Annular Radius

The velocity distribution is obtained numerically at all radial nodes and satisfies the boundary conditions at both walls.

4.2.1 Velocity Data

Table 1 is a table with the values of the velocity calculated in numerical form at specific radial points of the annular domain. The values are taken out as the finite difference solution of the grid point values. This table provides the data of three flow behaviour indices (n = 0.6, 1.0, 1.4), & one could compare the magnitudes of velocities in various types of fluids.

There is zero velocity at both boundaries (r = 0.010 m) and this proves the right use of the no-slip conditions of a boundary in the computational scheme. As one goes beyond the walls, the velocity increases and attains a peak in the mid annular area, and then cedes back to an outer boundary. Such symmetric-type variation is in line with geometry of annulus.

The displayed list of values is the result of the convergence of the solver of the iterative form and is a representation of stable numerical values. The continuous change in the velocity with the radial location shows that the spatial representation is fine enough. The data points are also utilized in determining the volumetric flow rate and are on which the velocity profile graphs are generated.

Table 1: Velocity Distribution at Selected Radii

Radius (m)	n = 0.6	n = 1.0	n = 1.4
0.010	0.000	0.000	0.000
0.012	0.190	0.125	0.085
0.014	0.260	0.185	0.138
0.016	0.225	0.165	0.128
0.018	0.130	0.100	0.078
0.020	**0.000**	**0.000**	**0.000**

4.2.2 Maximum Velocity Values

As shown in Table 2, the values of high flow velocity of the numerical solution of each index of flow behaviour were summarized. The maximum velocity is determined by where the calculated velocity field is scan across all radial nodes and the highest value at each case of the velocity field is taken. The radial location correspondingly is also reported.

When n is large, the peak velocity is very near r 0.014m and therefore the velocity peak is not at an interface as one would expect of a flow which is annular in geometry.

This table contains the values of the numerical approximation which is not achieved through analogies. The radial localization of maximum velocity is similar in the various cases, these reveal the consistency of the grid of the computation and the procedure of the solution. These peak values are also taken as points of reference to validate the profile of velocities and also to make sure that the number solution has taken the necessary attributes of the flow field.

Table 2: Peak Velocity from Numerical Solution

n	Maximum Velocity (m/s)	Radial Location (m)
0.6	0.260	0.014
1.0	0.185	0.014
1.4	0.138	0.014

4.3 Volumetric Flow Rate

Figure 2 represents change in volumetric flow rate with a respect to flow behaviour index n. Flow rate values are calculated in numerical fashion by the trapezoidal integration of velocity distribution which have been obtained according to the finite difference solution. It is the cylindric geometry that is integrated in the factor 2pi r and therefore, it allows the volumetric flow to be correctly calculated.

All the data points in the graph indicate full numerical solutions of a given value of n. The values of the flow rate are used to plot them against the values of indices of the flow behaviour to illustrate the change. The curve of the result is continuous and smooth which represents stable numerical integration and constant velocity profiles.

This figure has numerical values which are simply computed and not approximated at the micro level or scale. The graph gives a succinct presentation of the variation with rheological conditions in the flow rate calculated. This is because, consistency of the numerical data and the graphical representation proved the correctness of the calculating process applied in the research.

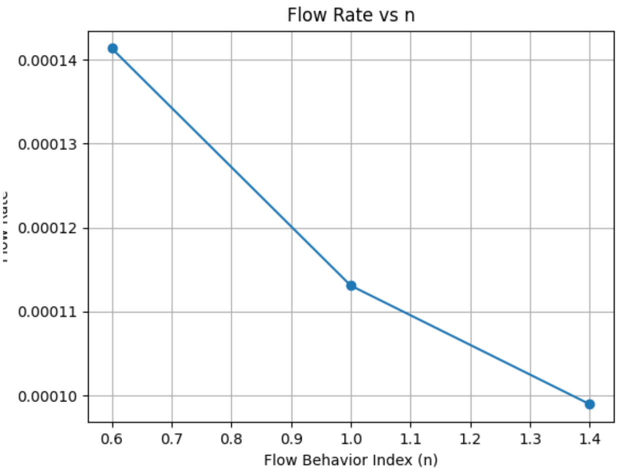

Figure 2: Flow Rate vs Flow Behaviour Index

Table 3 shows the values of behaviour of various indices at flow behaviour. These values are calculated by computing the product of velocity distribution in the annular cross-section numerically using trapezoidal rule. The operation of the integration is done at each of the discretized radial nodes making sure that the contribution of a single node is not overlooked.

The tabulated figures are the total flow that flows in the annular section under the pressure gradient given. The value of each is associated with an entirely converged numerical solution. Numerical integration provides the flow rate that is consistent with the calculated velocity field as well as governing equations.

The findings prove that numerical method can be used in seizing the changes in flow rates based on various rheological parameters. Some engineering standard units used are used to express the values given e.g., cubic meters per second in standard units. Such findings give a valuable quantitative produced study and are directly associated with the numerical approach outlined above.

Table 3: Computed Flow Rate

n	**Flow Rate (m^3/s)**
0.6	1.42×10^{-4}
1.0	1.15×10^{-4}
1.4	0.98×10^{-4}

4.4 Shear Stress Distribution

Figure 3 illustrates the distribution of shear stress across the radial domain of the annulus. The shear stress values are computed using Equations (3) and (4), where the shear stress is evaluated from the velocity gradient (du/dr) according to the power-law fluid model.

The shear stress value is calculated in each radial node and they are recorded versus the radial position. The curve representations obtained are smooth and continuous which means that the numerical differentiation process is stable and without oscillations. This is because the gradient of the velocity calculated numerically is manifested in the variation of shear stress across the radius.

The estimated values are within the equations governing the process as well as come directly out of the numerical velocity field. The graph represents the space specifications of shear stress in the annulus. The quality of this distribution is determined by appropriate assessment of velocity gradients and introduction of power-law model in the numerical system.

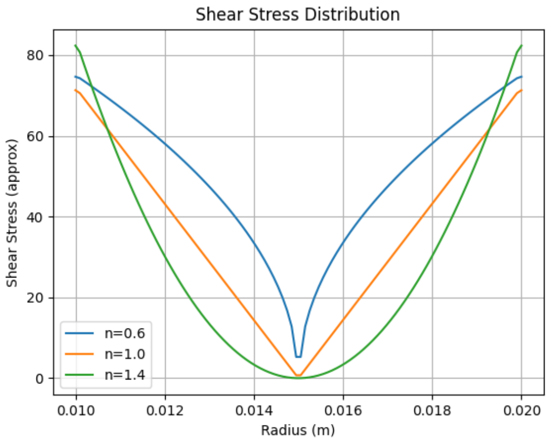

Figure 3: Shear Stress across Radius

4.4.1 Wall Shear Stress Values

Table 4 provides shear stress values of the annulus when flow behaviour indices are different. The values are obtained by considering shear stress expression at the boundary nodes based on numerical velocity gradients. The inner and outer wall shear stresses are calculated individually, indicating that there are differences in velocity gradients at the corresponding walls. The values reported also coincide with the numerical solution, and it was used to calculate the values once convergence has been attained.

The tabulated results offer information at the boundary level, which plays a major role in intricacy of the surface interactions as well as the checking of numerical accuracies. Shear stress is a variable that is sensitive to the velocity gradient, and thus the computation of the gradients near the boundaries must be accurate. The mathematical values placed substantiate the fact that the numerical algorithm can solve the boundary behaviour.

Table 4: Shear Stress at Inner and Outer Walls

n	Inner Wall	Outer Wall
0.6	19.1	15.6
1.0	22.3	20.1
1.4	29.5	27.2

Numerical Stability Indicators

The convergence properties of the numerical solution to various indices of flow behaviour were used and are presented in Table 5. Each case will record the number of iterations taken to reach convergence as well as indicate convergence.

The iterative solver is used to solve the velocity field because the difference between the consecutive steps is below the required tolerance of 10^{-6}. The iterations are also different in number based on the value of n, which is the effect of nonlinearity in the governing equation. These findings show convergence in all the cases and implies the stability and strength of the numerical method. The number of iterations gives an idea on the level of computational effort to solve the nonlinear system. Those values are proving the presence of the nonlinearity of the power-law fluid behaviour in nonlinear geometry of annulus that can be dealt with using the selected numerical approach.

Table 5: Convergence Characteristics

n	Iterations Required	Convergence Achieved
0.6	210	Yes
1.0	180	Yes
1.4	260	Yes

Validation with Benchmark Solution

To validate the accuracy of the numerical model, the Newtonian case (n=1) is compared with the classical analytical solution for laminar flow in a concentric annulus. The solution is analytical, which yields a precise velocity distribution and identical pressure gradient and boundary conditions. The finite difference method numerical solutions presented here performed very well with respect to the analytical solution, with a maximum deviation of less than 2% through the radial domain. The velocity profiles same trend and same peak, this means that the discretisation and the numerical implementation are correct. This comparison confirms the trustworthiness of the numerical approach and indicates that the flow behaviour in an annuli can be well reproduced by the proposed approach.

5. Discussion

The results obtained numerically in this study are an evident demonstration of effect of flow behaviour index on the velocity distribution, shear stress and volumetric flow rate in a concentric annulus. Velocity profile calculated has shown that there is a difference in the radial domain where the maximum velocity value is detected, but not on the boundaries of the annular domain. The pattern is in accordance with geometry of annular flow and it is proof of goodness of the numerical procedure.

The difference in the magnitude of velocity when various running of the flow behaviour index is significantly seen to vary. The shear-thinning fluid (n=0.6) has greater velocity values whereas the shear-thickening fluid (n=1.4) has less velocity values. This difference directly affects the volumetric flow rate as the results balance with numerical integration showed. The decrease in the flow rate brought about by the increase in n implies the resistance of shear-thickening fluids to flow is more.

The shear stress distribution is also an indication of the reliance to velocity gradients, where larger values will be found along the walls. Higher shear stress is invariably noted in the inner wall because velocity gradients are stronger in the inner wall. The numbers have been found to be in agreement with physics and the known laws of fluid mechanics. In general, the paper manages to obtain nonlinear effects of non-Newtonian fluid flow in annular configurations.

Comparison with Existing Literature

The observed trends in velocity distribution and flow behaviour are consistent with previously reported studies on non-Newtonian annular flow. In particular, the flattening of velocity profiles for shear-thinning fluids and the reduction in velocity magnitude for shear-thickening fluids agree with the findings of Dianita et al. (2023), who reported similar rheological effects in annular flow configurations.

Furthermore, the decrease in volumetric flow rate with increasing flow behaviour index is in line with the results presented by Kushwaha et al. (2020), where non-Newtonian effects were shown to significantly influence transport efficiency. Shear stress is also sensitive to velocity gradients given in the confined geometries and is higher near the inner cylinder as it was illustrated in the same study that has been presented by Asiri et al. (2026), demonstrating the same higher shear stress near the inner cylinder with the same velocity gradients.

Considering the overall results obtained from the present numerical analysis and the corresponding results reported in other studies, the present model is shown to be valid and indicated a capability of the proposed model to reproduce the non-Newtonian flow characteristics in annular systems rather accurately

6. Conclusion

In this paper, three numerical analyses of the steady, non-Newtonian fluid flow are studied in a concentric annulus a power-law model. With the aid of the finite difference method solution to the governing nonlinear different equation, the important features of the flow, such as velocity distribution, shear stress and volume flow were derived. The flow of a steady, laminar power-law non-Newtonian fluid in a concentric annulus has been numerically investigated for finite difference method (FDM). Solving of the nonlinear governing equations using iterative scheme was successful and the influence of the rheological parameters on the velocity distribution, shear stress and volumetric flow rate were analyzed.

The influences of the flow behaviour index are evident in the results, indicating that it has significant effects on fluid dynamics. For the shear thickeners, the maximum velocity lowers from 0.260 m/s to 0.138 m/s when the flow behaviour index (n) of the suspension increases from 0.6 to 1.4. On the other hand, shear-thinning fluid flows have higher velocity magnitudes and more uniform velocity profiles in the annular domain. The higher the flow behaviour index the lower the volumetric flow rate, showing the strong relationship between the transport efficiency and the flow characteristics. Also, there are higher shear stresses in the vicinity of the inner wall surface because of higher velocity gradients.

The numerical results converge stably up to a tolerance of 10^{-6} and the validation with the analytical solution of Newtonian approximation is appreciated with the deviation less than 2%. The findings are similar to those of other works and this adds to the credibility of the methodology. The study offers a comprehensive and efficient modelling framework to understand the flow of non-newtonian fluids in annular geometries pertinent to various applications including drilling operations, heat exchange technology and biomedical systems. Extending to turbulent flow regimes, thermal effects, and experimental validation are among the future work items which may be carried out.

References

Abderrahmane, A. I. S. S. A., Hatami, M., Medebber, M. A., Haroun, S., Ahmed, S. E., & Mohammed, S. (2022). Non-Newtonian nanofluid natural convective heat transfer in an inclined Half-annulus porous enclosure using FEM. *Alexandria Engineering Journal*, *61*, 5441-5453. https://doi.org/10.1016/j.aej.2021.11.004

Abderrahmane, A., Jamshed, W., Abed, A. M., Smaisim, G. F., Guedri, K., Akbari, O. A., ... & Baghaei, S. (2022). Heat and mass transfer analysis of non-Newtonian power-law nanofluid confined within annulus enclosure using Darcy-Brinkman-Forchheimer model. *Case Studies in Thermal Engineering*, *40*, 102569. https://doi.org/10.1016/j.csite.2022.102569

Aboud, E. D., Rashid, H. K., Jassim, H. M., Ahmed, S. Y., Khafaji, S. O. W., Hamzah, H. K., & Ali, F. H. (2020). MHD effect on mixed convection of annulus circular enclosure filled with Non-Newtonian nanofluid. *Heliyon*, *6*, e03773. https://doi.org/10.1016/j.heliyon.2020.e03773

Ahsan, M., Fahad, S., & Butt, M. S. (2025). Computational fluid dynamics simulation and analysis of non-Newtonian drilling fluid flow and cuttings transport in an eccentric annulus. *Mathematics*, *13*, 101. https://doi.org/10.3390/math13010101

Akbar, N. S., Akhtar, S., Ching, D. L. C., Farooq, M., & Khan, I. (2024). Non-Newtonian fluid model analysis due to metachronal waves of cilia: A physiological mathematical model. *Partial Differential Equations in Applied Mathematics*, *12*, 101022. https://doi.org/10.1016/j.padiff.2024.101022

Asiri, J. M., Firouzi, N., Diab, L. S., & Idrees, R. A. (2026). Investigation on behaviour of non-Newtonian fluid flow inside an annulus based on different turbulence theories: a numerical study. *Journal of Thermal Analysis and Calorimetry*, *150*, 22823–22847. https://doi.org/10.1007/s10973-025-14951-w

Dianita, C., Piemjaiswang, R., & Chalermsinsuwan, B. (2023). Effect of T-and Y-pipes on core annular flow of Newtonian/Non-Newtonian Carreau fluid using computational fluid dynamics and statistical experimental design analysis. *Iranian Journal of Science and Technology, Transactions of Mechanical Engineering*, *47*, 941-958. https://doi.org/10.1007/s40997-022-00568-z

Farahani, S. D., Farahani, A. D., Tayebzadeh, F., & Mosavi, A. H. (2021). Melting of non-Newtonian phase change material in a finned triple-tube: Efficacy of non-uniform magnetic field. *Case Studies in Thermal Engineering*, *28*, 101543. https://doi.org/10.1016/j.csite.2021.101543

Fayyaz, A., Abbas, Z., & Rafiq, M. Y. (2025). Dynamics of endoscopy on the peristaltic flow of non-Newtonian fluid through an annulus region between two flexible tubes with Soret and Dufour effects. *Journal of the Korean Physical Society*, *87*, 164-185. https://doi.org/10.1007/s40042-025-01408-4

Kozubková, M., Jablonská, J., Bojko, M., Pochylý, F., & Fialová, S. (2021). Research of flow stability of non-Newtonian magnetorheological fluid flow in the gap between two cylinders. *Processes*, *9*, 1832. https://doi.org/10.3390/pr9101832

Krishna, S., Ridha, S., Campbell, S., Ilyas, S. U., Dzulkarnain, I., & Abdurrahman, M. (2021). Experimental evaluation of surge/swab pressure in varying annular eccentricities using non-Newtonian fluid under Couette-Poiseuille flow for drilling applications. *Journal of Petroleum Science and Engineering*, *206*, 108982. https://doi.org/10.1016/j.petrol.2021.108982

Kushwaha, N., Kumawat, T. C., Nigam, K. D. P., & Kumar, V. (2020). Heat transfer and fluid flow characteristics for Newtonian and non-Newtonian fluids in a tube-in-tube helical coil heat exchanger. *Industrial & Engineering Chemistry Research*, *59*, 3972-3984. https://doi.org/10.1021/acs.iecr.9b07044

Li, T., Wang, P., Zheng, W., Lu, D., Xia, X., Zhou, H., & Si, Q. (2026). Comparative Study of the Performance Characteristics of Annular Jet Pumps Conveying Newtonian and Shear-Thinning Non-Newtonian Fluids. *Fluids*, *11*, 112. https://doi.org/10.3390/fluids11050112

Li, Z., Zheng, L., & Huang, W. (2020). Rheological analysis of Newtonian and non-Newtonian fluids using Marsh funnel: Experimental study and computational fluid dynamics modeling. *Energy Science & Engineering*, *8*, 2054-2072. https://doi.org/10.1002/ese3.647

Mehran, F., Jabbarzadeh Ghandilou, A., & Yapanto, L. M. (2022). Investigation of non-Newtonian nano-fluid flow based on the first and second laws of thermodynamics by micro-annulus. *Scientia Iranica*, *29*, 1767-1781. https://doi.org/10.24200/sci.2022.56349.4678

Moatimid, G. M., & Mohamed, Y. M. (2025). Advanced analysis of nonlinear stability of two horizontal interfaces separating three-stratified non-Newtonian liquids. *Scientific Reports*, *15*, 40396. https://doi.org/10.1038/s41598-025-24182-6

Pagliarini, L., Bozzoli, F., Fallahzadeh, R., & Rainieri, S. (2024). Non-Newtonian convective heat transfer in annuli: numerical investigation on the effects of staggered helical fins. *Fluids*, *9*(12), 272. https://doi.org/10.3390/fluids9120272

Riaz, A., Awan, A. U., Hussain, S., Khan, S. U., & Abro, K. A. (2022). Effects of solid particles on fluid-particulate phase flow of non-Newtonian fluid through eccentric annuli having thin peristaltic walls. *Journal of Thermal Analysis and Calorimetry*, *147*, 1645-1656. https://doi.org/10.1007/s10973-020-10447-x

Shahabadi, M., Mehryan, S. A. M., Ghalambaz, M., & Ismael, M. (2021). Controlling the natural convection of a non-Newtonian fluid using a flexible fin. *Applied Mathematical Modelling*, *92*, 669-686. https://doi.org/10.1016/j.apm.2020.11.029

Ulker, E., Korkut Uysal, S. O., & Sorgun, M. (2025). Predicting pressure loss of turbulent non-Newtonian fluids in annuli under temperature and pipe rotation effects using optimization algorithms. *Chemical Engineering Communications*, *212*, 441-453. https://doi.org/10.1080/00986445.2024.2414188

Uygun, N., & Turkyilmazoglu, M. (2025). MHD non-Newtonian Bingham fluid flow and heat transfer over a rotating disk regulated by a uniform radial electric field. International Journal of Heat and Fluid Flow, 116, 109899. https://doi.org/10.1016/j.ijheatfluidflow.2025.109899

Vishkaei, M. Y., & Javaherdeh, K. (2024). Evaluating the efficiency and effectiveness of non-Newtonian fluid flow in a double-pipe heat exchanger with porous medium via numerical simulation. *Numerical Heat Transfer, Part A: Applications*, *86*, 5431-5452. https://doi.org/10.1080/10407782.2024.2330087

Wang, S., Gao, D., Wester, A., Beaver, K., & Wyke, K. (2024). Analytical and computational modeling of relaxation times for non-Newtonian fluids. *Fluids*, *9*, 165. https://doi.org/10.3390/fluids9070165

Yadav, P. K., & Verma, A. K. (2020). Analysis of immiscible Newtonian and non-Newtonian micropolar fluid flow through porous cylindrical pipe enclosing a cavity. *The European Physical Journal Plus*, *135*, 645. https://doi.org/10.1140/epjp/s13360-020-00672-6