WAPRIME
https://welchacademy.com/index.php/prime
e-ISSN: 3062-2743

Analytical and numerical investigation of viscous heating in parallel-plate Couette flow

Enes Kubilay Ünal *1

[1] *Mersin University, Department of Mechanical Engineering, Türkiye, suekunala@gmail.com*

Cite this study: Ünal, E. K. (2024). Analytical and numerical investigation of viscous heating in parallel-plate Couette flow. WAPRIME, 1(1), 57-69.

https://doi.org/10.5281/zenodo.14931785

Keywords
Couette Flow
Viscous Heating
Analytical Solution
Numerical Simulation
Heat Transfer

Research Article
Received: 5 November 2024
Revised: 20 December 2024
Accepted: 23 December 2024
Published: 31 December 2024

Abstract

This study investigates the heating problem occurring in viscous oils used as lubricants between moving machine elements, using the Couette flow model. A system consisting of two parallel plates, maintained at a constant temperature (20 °C), with one moving relative to the other at a constant velocity of 8 m/s, is considered. The lubricant fluid has a dynamic viscosity of 0.3 Ns/m^2 and a thermal conductivity of 0.13 W/mK. The distance between the plates is 1 mm, and the plate area is 0.1 m^2. The effect of viscous heating on the oil temperature was investigated using both analytical and numerical methods. Under the assumptions of steady-state, fully developed, incompressible, and Newtonian fluid flow, the momentum and energy equations were simplified to obtain analytical solutions. Equations predicting a linear velocity profile and a parabolic temperature profile were derived. Using these equations, the maximum oil temperature was calculated as 311.54 °C, and the power required to move the upper plate was found to be 1920 W. For the numerical solution, ANSYS Fluent, a finite-volume-based computational fluid dynamics solver, was utilized. A two-dimensional, structured mesh was employed, and the steady-state laminar flow and energy equations were solved using a pressure-based solver and the SIMPLE algorithm. The velocity and temperature profiles obtained from the numerical solution were compared with the analytical solutions. An almost perfect agreement was observed between the analytical and numerical results for the temperature profiles, while minor deviations were detected in the velocity profiles, especially in the middle region of the channel and near the upper plate. Observed deviations are likely due to numerical factors (mesh resolution, algorithm, boundary conditions). This study definitively shows that viscous heating significantly increases temperature in lubrication systems, a critical consideration for engineering design.

1. Introduction

One of the most important examples of the application of heat transfer and fluid mechanics principles in engineering is in lubrication systems between moving machine elements. In these systems, the lubricating fluid used to reduce friction and prevent wear can also experience a significant temperature increase due to a phenomenon known as viscous heating [1]. Viscous heating can be a critical factor, especially in systems operating at high speeds and with narrow gaps, potentially leading to thermal and chemical degradation of the oil, thinning of the lubrication film, and even system failure [2].

The Couette flow considered in this study is a fundamental model in viscous fluid mechanics and heat transfer problems. This flow, which occurs between two parallel plates with one moving at a constant velocity relative to the other, represents the flow conditions encountered in many engineering applications, such as bearings, gears, seals, and viscometers [3]. The simple geometry of Couette flow allows for the investigation of viscous heating effects using both analytical and numerical methods.

Viscous heating is the process of converting mechanical energy into thermal energy due to the internal friction (viscosity) of the fluid. This phenomenon causes a significant increase in oil temperature in lubrication systems, particularly at high speeds and in narrow gaps. This temperature rise can alter the properties of the oil (viscosity, density), negatively impacting system performance [4].

The problem of viscous heating in Couette flow has been extensively studied in the literature. Initial studies were generally limited to analytical solutions and were conducted under simplifying assumptions such as constant viscosity, adiabatic walls, or negligible viscous heating [5-6]. However, these assumptions are not always valid in real engineering applications.

With the development of numerical methods (especially Computational Fluid Dynamics - CFD), the effects of viscous heating can be modeled more realistically under complex geometries, variable oil properties, and different boundary conditions [7-8]. Commercial CFD software such as ANSYS Fluent is widely used to solve such problems, providing engineers with significant advantages in design and optimization processes [9].

There are also studies in the literature that address viscous heating in Couette flow using both analytical and numerical methods. However, these studies are often limited to specific parameter ranges or special boundary conditions. This study aims to fill this gap in the literature by providing both an analytical solution for the specified problem and detailed numerical modeling using ANSYS Fluent and by presenting a comparative evaluation of the results. This comparison is crucial for determining the validity limits of the analytical model and confirming the accuracy of the numerical model.

The purpose of this study is to calculate the maximum oil temperature and the power required to move the upper plate due to viscous heating in the Couette flow problem defined above, using both analytical and numerical (ANSYS Fluent) methods, and to compare the results. Within this scope, the following will be investigated: how much the maximum oil temperature increases due to viscous heating in Couette flow under the given parameters, how much power is required to move the upper plate at a constant speed, and what is the difference between the temperature and power values obtained by analytical and numerical methods. It will also be assessed what factors this difference may depend on and what kind of inferences the results obtained can provide in terms of the design and analysis of lubrication systems. This study aims to contribute to a better understanding of the viscous heating problem and to the comparison of different approaches in solving such problems by providing both a theoretical basis (analytical solution) and using a practical engineering tool.

2. Material and Method

This study presents a comprehensive analysis of viscous heating within a Couette flow system, which consists of two parallel plates with a viscous lubricating oil in between. To thoroughly investigate the temperature and velocity fields, both analytical and numerical approaches were employed. The analytical approach involved deriving simplified forms of the governing Navier-Stokes and energy equations based on the assumptions of steady-state, fully developed, incompressible, and Newtonian flow, with constant fluid properties. Meanwhile, the numerical approach utilized the commercial finite-volume-based CFD software ANSYS Fluent. This computational method allowed for a direct comparison with the analytical results and enabled an assessment of the validity of the simplifying assumptions made in the analytical derivation. The comparison between these two methodologies not only serves to validate the numerical model but also provides a deeper understanding of the limitations and applicability of the analytical solution.

This study investigates the effects of viscous heating on the temperature and velocity distributions within a lubricating oil undergoing Couette flow, schematically illustrated in Figure 1. The system under consideration consists of two parallel plates, both maintained at a constant temperature of 20 °C. The upper plate moves at a constant velocity of 8 m/s relative to the stationary lower plate. The lubricating oil is characterized by a dynamic viscosity of 0.3 Ns/m^2 and a thermal conductivity of 0.13 W/mK. The geometrical parameters are defined as follows: a plate separation distance of 1 mm and a plate area of 0.1 m^2. This configuration, representative of many practical lubrication scenarios, allows for a detailed analysis of viscous dissipation and its impact on the fluid's thermal behavior.

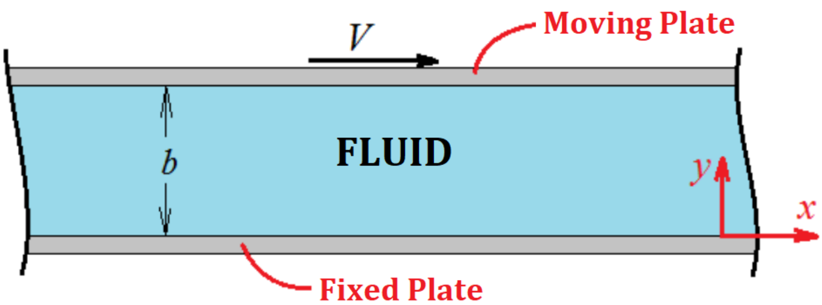

Figure 1. Schematic representation of the Couette flow problem between two parallel plates.

2.1. Analytical Model

This study compares the analytical solution and numerical simulation results of heat generation in a Couette flow problem involving a viscous lubricating oil. The analysis was performed using fundamental heat transfer principles, differential transport equations, and dimensionless numbers (such as Reynolds and Nusselt numbers). Thus, the agreement between the theoretical model and the numerical solution was examined, and the effects of parameters on the flow and heat transfer were revealed. Couette flow, as illustrated in Figure 1, is a system where two parallel plates are separated by a viscous oil, with one plate moving relative to the other at a velocity U. In this study, a steady-state regime was assumed, meaning the flow is considered time-independent. The flow was also assumed to be fully developed, implying that the velocity profile does not change in the flow direction, satisfying the conditions $\frac{\partial u}{\partial x} = 0$ and $\frac{\partial T}{\partial} = 0$. A one-dimensional flow was considered, with motion only in the x-direction and neglecting velocity components in the y and z directions (v=0, w=0). Both plates were assumed to be maintained at a constant temperature, and the oil was treated as an incompressible fluid with constant density. Furthermore, the viscosity (μ) and thermal conductivity (k) of the oil were assumed to be independent of temperature, and the fluid was considered to be Newtonian. The effect of gravitational forces on the flow was neglected, and changes in kinetic and potential energy were disregarded. Under these assumptions and employing boundary layer approximations, the momentum equation (Navier-Stokes equation) and the energy equation for Couette flow were determined in their simplified forms, following the relevant derivation steps and considering the fully developed flow assumption.

2.2 Heat Convection and Heat Convection Coefficient

Heat transfer by convection occurs between a fluid and a solid surface at a constant temperature due to the combined effects of conduction and fluid motion. This mode of heat transfer is quantified by Newton's Law of Cooling, given by Equation (1) and Equation (2) [10]:

$$Q_{convection} = hA(T_y - T_\infty) \tag{1}$$

$$q_{convection}'' = h(T_y - T_\infty) \tag{2}$$

Here, h represents the convective heat transfer coefficient, expressed in W/m^{2}K, which indicates the intensity of heat transfer between a surface and a fluid. The symbol A denotes the surface area over which heat transfer occurs, with units of m^2. T_y represents the temperature of the surface in question, while T_∞ represents the free-stream temperature of the fluid, i.e., the temperature of the fluid far from the surface, where it is not affected by the heat transfer process.

2.3 Dimensionless Numbers

Dimensionless numbers are employed in heat transfer and fluid mechanics to characterize the nature of the flow and heat transfer processes. Two dimensionless numbers of particular relevance to this study are the Nusselt number and the Reynolds number. The Nusselt number (Nu) provides a ratio of the convective heat transfer to the conductive heat transfer within the fluid. It is generally defined as shown in Equation (3) [10]:

$$Nu = \frac{hL_c}{k} \tag{3}$$

The parameter L_c denotes the characteristic length, which is a crucial dimension that defines the geometry of the system, such as the spacing between the parallel plates in the Couette flow configuration. The material's thermal conductivity, denoted as k and measured in W/mK, quantifies its ability to conduct heat. The Nusselt number (Nu) serves as a metric for comparing convective and conductive heat transfer; a Nu value of 1 indicates purely conductive heat transfer, while a Nu greater than 1 suggests an increasing influence of convection over conduction. The nature of the flow regime, whether laminar or turbulent, is characterized by the Reynolds number (Re), which represents the ratio of inertial forces to viscous forces in the fluid. The Reynolds number is defined in accordance with Equation (4) [10]:

$$Re = \frac{\rho VL}{\mu} \tag{4}$$

Here, ρ represents the fluid density (kg/m^3), V represents the characteristic velocity (m/s), L represents the characteristic length (m), and μ represents the dynamic viscosity of the fluid (N.s/m^2 or Pa.s). For internal flows, a Reynolds number less than 2300 (Re < 2300) indicates laminar flow, while for external flows, a critical value of approximately 500,000 is used to determine whether the flow is turbulent.

2.4 Differential Equations

To effectively model the intricate fluid flow and heat transfer phenomena within the Couette flow system, we employed a robust set of fundamental differential equations governing momentum and energy transfer. Originating from core conservation laws, these equations-the continuity equation, the Navier-Stokes equations (which capture momentum dynamics), and the energy equation-provide a comprehensive framework for understanding the spatial and temporal variations of velocity, pressure, and temperature within the fluid. In this study, we emphasize the crucial role of convection in the heat transfer process, driven primarily by fluid motion. Forced convection, propelled by the moving plate, emerges as the predominant mechanism of heat transfer, operating alongside conduction to enhance overall efficiency. Under the specific conditions of our research-steady-state, incompressible, Newtonian fluid with constant properties-we can adeptly simplify these equations, as elaborated in Section 2.1. Our analysis begins with the continuity equation, a fundamental expression of mass conservation that is articulated in Equation (5). This foundational approach lays the groundwork for a profound understanding of the complex interactions at play in the Couette flow system [10-12]:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{5}$$

This equation expresses the principle of conservation of mass and is valid for incompressible fluids. In a two-dimensional (x, y) flow field, it states that the sum of the change in the x-component of velocity (u) concerning x $\left(\frac{\partial u}{\partial x}\right)$ and the change in the y-component of velocity (v) concerning y $\left(\frac{\partial v}{\partial y}\right)$ must be zero. In Couette flow, due to the assumption of fully developed flow, there is no change in velocity in the flow direction (x-direction), meaning $\frac{\partial u}{\partial x}=0$. Furthermore, because the flow is considered one-dimensional (with only an x-component of velocity, v = 0), the equation is automatically satisfied [11-12]. The momentum equation used in this study is given by Equation (6):

$$\rho=\left(u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}\right)=-\frac{\partial p}{\partial x}+\mu\frac{\partial^2 u}{\partial y^2} \tag{6}$$

This equation, representing the conservation of momentum for a fluid element, is the fluid mechanics equivalent of Newton's second law (F = ma). It establishes a relationship between the net force acting on a fluid element and the resulting acceleration. The left-hand side of the equation quantifies the acceleration: ρ is the fluid density, $u\frac{\partial u}{\partial x}$ represents the convective acceleration in the x-direction (due to changes in velocity along the x-direction), and $v\frac{\partial v}{\partial y}$ represents the convective acceleration in the y-direction (due to changes in velocity along the y-direction). The right-hand side accounts for the forces: $-\frac{\partial p}{\partial x}$ is the pressure force arising from the pressure gradient in the x-direction, and $\mu\frac{\partial^2 u}{\partial y^2}$ represents the viscous force, arising from the shear stresses within the fluid. Under the specific conditions of fully developed, one-dimensional Couette flow, the assumptions of $\frac{\partial u}{\partial x}=0$ and $v=0$ lead to a zero value for the left-hand side (no net acceleration). Additionally, applying the boundary layer approximation, the pressure gradient in the y-direction is considered negligible $\left(\frac{\partial p}{\partial x}=0\right)$, making pressure solely a function of x [10]. Finally, for fully developed flow, the pressure gradient in the x-direction is also zero $\left(\frac{dp}{dx}=0\right)$. As a result of these simplifications, the momentum equation reduces to Equation (7), signifying a balance, or equilibrium, of the viscous forces.

$$0=\mu\frac{\partial^2 u}{\partial y^2} \tag{7}$$

The energy equation employed in this study, given by Equation (8), expresses the conservation of energy, relating the internal energy change of a fluid element to heat transfer and work interactions [11-12].

$$\rho c_p\left(u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}\right)=k\frac{\partial^2 T}{\partial y^2} \tag{8}$$

In Equation (8), the left-hand side represents the rate of change of internal energy within the fluid element. The terms are defined as follows: ρ is the fluid density, c_p is the specific heat at constant pressure, $u\frac{\partial T}{\partial x}$ represents the rate of energy transport by convection in the x-direction due to the bulk fluid motion and temperature gradient, and $v\frac{\partial T}{\partial y}$ represents the rate of energy transport by convection in the y-direction. The right-hand side of the equation accounts for heat transfer and work done on the fluid element. Specifically, $k\frac{\partial^2 T}{\partial y^2}$ represents the rate of heat transfer by conduction in the y-direction, and $\mu\frac{\partial^2 u}{\partial y^2}$ represents the rate of internal energy increase due to viscous dissipation (the conversion of mechanical energy to heat due to internal friction). These equations, along with their accompanying explanations, form the theoretical foundation for analyzing the viscous heating problem in Couette flow and illustrate the derivation of the analytical solution. In contrast, the numerical solution (using ANSYS Fluent) solves these equations using the finite volume method without applying the aforementioned simplifying assumptions.

3. Analytical Solution of Couette Flow

This section presents the analytical solution for the Couette flow problem, a classic example in fluid mechanics and heat transfer. Couette flow describes the laminar flow of a viscous fluid in the space between two parallel plates, one of which is moving at a constant velocity relative to the other. In the present case, the lower plate is fixed, while the upper plate moves with a constant velocity of 8 m/s. The distance separating the plates is 1 mm. The fluid is a lubricating oil characterized by a dynamic viscosity of 0.3 Ns/m^2 and a thermal conductivity of 0.13 W/m.K. The upper plate has a surface area of 0.1 m^2. These parameters define the physical setup for the Couette flow problem under consideration, allowing for a combined analytical and numerical study of the effects of viscous heating.

3.1 Temperature Distribution

Assuming constant fluid properties, the analytical expression for the temperature distribution within the fluid is derived as Equation (9) [12]:

$$T_{(y)} = T_0 + \frac{\mu}{2k}u^2\left(\frac{y}{L} + \frac{y^2}{L^2}\right) \tag{9}$$

Equation (9) shows that the temperature, T, varies parabolically with position y between the plates, where T_0 represents the initial and boundary temperature of both plates. To find the location of the maximum temperature, differentiate $T_{(y)}$with respect to y and set the result equal to zero $\left(\frac{\mathrm{dT}}{\mathrm{dy}} = 0\right)$. This yields $y_{(max)} = \frac{L}{2}$, indicating that the maximum temperature occurs at the midpoint between the plates. Substituting this value into Equation (9) gives the maximum temperature, as expressed in Equation (10):

$$T_{(max)} = T_0 + \frac{\mu}{8k}u^2 \tag{10}$$

3.2 Calculation of Shear Stress and Required Power

The viscous nature of the fluid leads to the development of a shear stress, τ, at the interface between the fluid and the moving upper plate. This shear stress is a function of the velocity gradient and is given by Equation (11) [12]:

$$\tau = \mu \frac{\partial u}{\partial y} \tag{11}$$

For the specific case of Couette flow, with its characteristic linear velocity profile, the velocity gradient is constant (U/L), and the shear stress simplifies to Equation (12) [12]:

$$\tau = \mu \frac{u}{L} \tag{12}$$

This shear stress represents the force per unit area required to overcome the viscous resistance of the fluid. The power (P), required to maintain the motion of the upper plate at the constant velocity (U) is the product of the shear stress, the plate area (A), and the plate velocity, as given by Equation (13) [11-12]:

$$P = \tau A u \tag{13}$$

3.3 Numerical Simulation

To validate the analytical solution, a numerical simulation was performed using ANSYS Fluent. The simulation was configured for laminar flow, with the Reynolds number maintained within the laminar regime based on the calculated values. A detailed mesh was created using rectangular elements. The upper plate was set to move at a velocity of 8 m/s, with an initial temperature defined as T_0, while the lower plate was kept stationary and also at the initial temperature. The finite volume method was used as the solution method, and the differential equations were solved using this method. The simulation results were compared with the temperature and velocity distributions obtained from the analytical model to verify the model.

4. Results

This section presents the results obtained from both analytical and numerical solutions of the viscous heating problem in Couette flow between two parallel plates. One plate is maintained at a constant temperature and moves at a constant velocity of 8 m/s relative to the other. The fluid used is a lubricating oil with a dynamic viscosity of 0.3 $\mathrm{Ns/m^2}$ and a thermal conductivity of 0.13 W/mK. The distance between the plates is set at 1 mm, and the area of each plate is 0.1 $\mathrm{m^2}$. Both plates are kept at a constant temperature of T_0 at 20 °C.

The analytical solution was derived using the assumptions and equations detailed in Section 2. Meanwhile, the numerical solution was performed using ANSYS Fluent software, following the modeling approach and settings outlined in Section 3. The results from the analytical calculations are presented in Table 1.

Table 1. Analytical results for key parameters in the Couette flow.

Parameter	Units	Result
y_{max}	m	L/2
T_{max}	K	313.592
P	$\mathrm{W/m^2}$	1920

Table 1 summarizes the results of the analytical solution, presenting key quantities obtained in the study. These parameters include the location of maximum displacement ($\mathrm{y_{max}}$), the maximum temperature ($\mathrm{T_{max}}$), and the power density (P). The values given in the table are important for demonstrating the accuracy and physical meaning of the calculations performed. In particular, the fact that y_{max} occurs at the L/2 position, dependent on the system geometry, indicates the symmetry of the model used in the analysis. Similarly, the maximum temperature (313.592 K) and power density (1920 $\mathrm{W/m^2}$) allow for an evaluation of the thermal and energy performance of the system. These data demonstrate the effectiveness of the methodology used in the study and provide a basis for further analyses.

4.1 Velocity Profile Analysis

The analytical solution, as outlined in Section 2, predicts a linear velocity profile, given by the equation. This linearity is characteristic of Couette flow, implying a constant velocity gradient $\left(\frac{dU}{dy}\right)$ and, consequently, a constant shear stress throughout the fluid. The velocity distribution between the plates, based on the analytical solution, is presented in Figure 2.

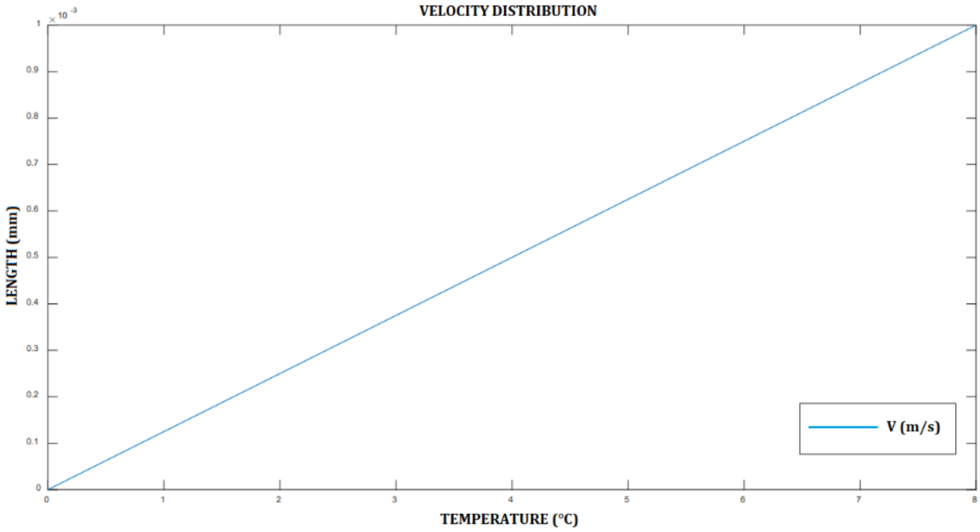

Figure 2. Velocity distribution between the plates according to the analytical solution.

Figure 3 visualizes the velocity distribution obtained from ANSYS Fluent, using velocity vectors (arrows) and a color scale. This figure confirms the linear nature of the velocity profile and the fully developed nature of the flow (i.e., no change in the velocity profile in the x-direction). Arrows represent velocity vectors, and colors indicate velocity magnitude. The length and color of the arrows represent the magnitude of the velocity, ranging from 0 m/s (blue) at the lower plate to a maximum 8 m/s (red) at the upper plate.

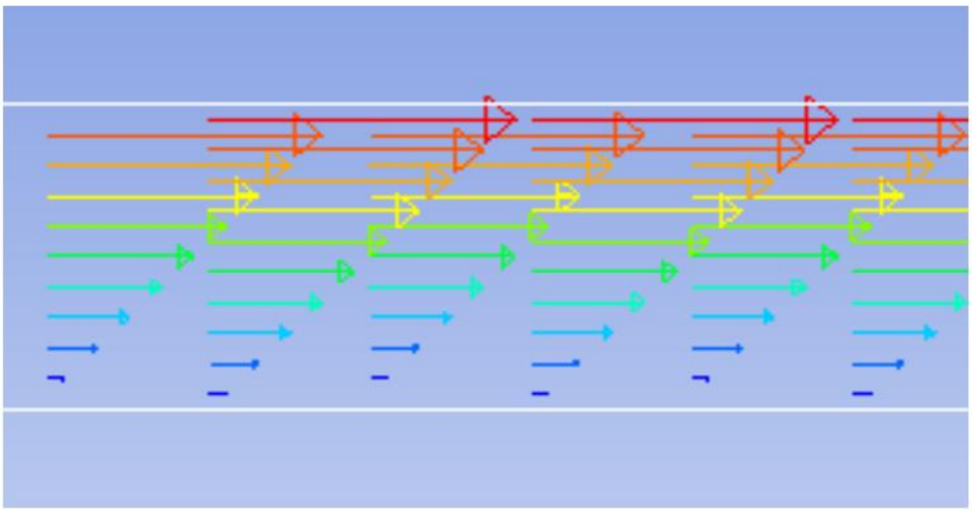

Figure 3. Velocity distribution between the plates according to the simulation results.

Figure 4 compares the analytical and numerical solutions for the velocity profile in Couette flow. The numerical solution exhibits a generally linear velocity profile and shows good agreement with the analytical solution (represented by red squares). However, noticeable deviations between the two solutions are observed, particularly in the central region of the channel and near the upper plate. Figure 4 displays the x-component of velocity (u) as a function of the distance between the plates (y-coordinate).

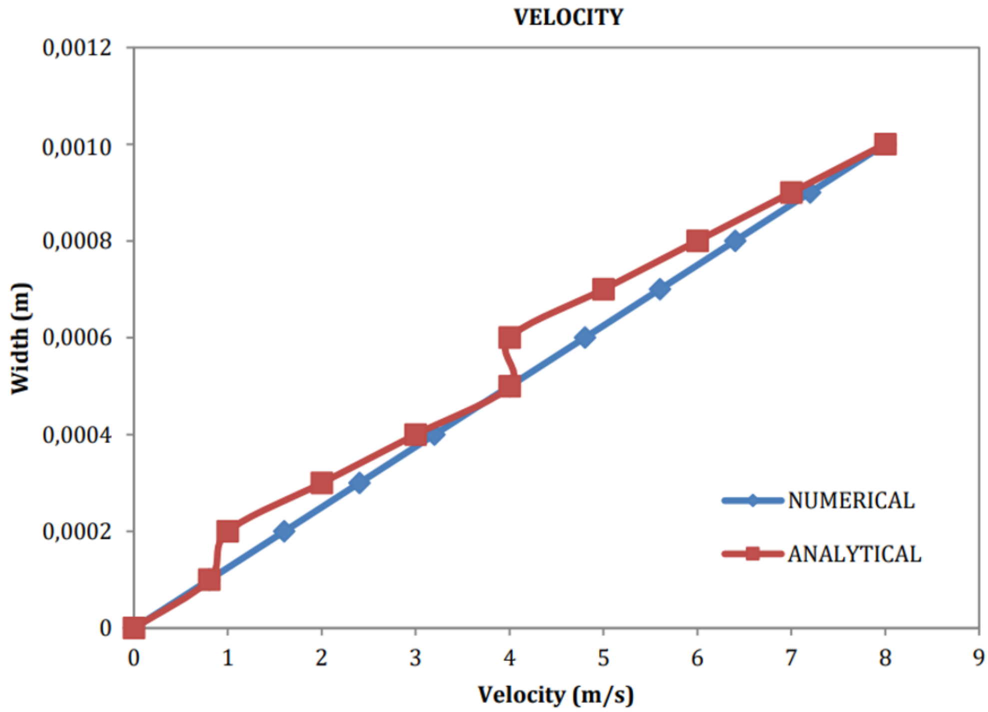

Figure 4. Comparison of analytical and numerical results for velocity distribution.

Both solutions satisfy the boundary conditions of zero velocity at the lower plate (y = 0, no-slip condition) and 8 m/s at the upper plate (y = L). These deviations may be attributed to the mesh resolution in the numerical model, particularly near the upper plate, being insufficient to fully capture the velocity gradient. Additionally, the solution algorithm (SIMPLE), discretization scheme (second-order upwind), and the implementation of boundary conditions in ANSYS Fluent may contribute to these discrepancies.

4.2 Temperature Profile Analysis

The analytical solution, as derived in Section 2 and expressed by the equation for $T_{(y)}$, predicts a parabolic temperature profile within the fluid. This equation demonstrates that viscous dissipation causes the temperature profile to deviate from a uniform distribution. Starting from the constant temperature of the plates (T_0 = 20 °C), the temperature increases due to viscous heating, reaching its maximum value at the midpoint between the plates ($y = \frac{L}{2}$), and then decreases symmetrically towards the plates, resulting in a parabolic shape. The temperature distribution corresponding to the analytical solution is illustrated in Figure 5.

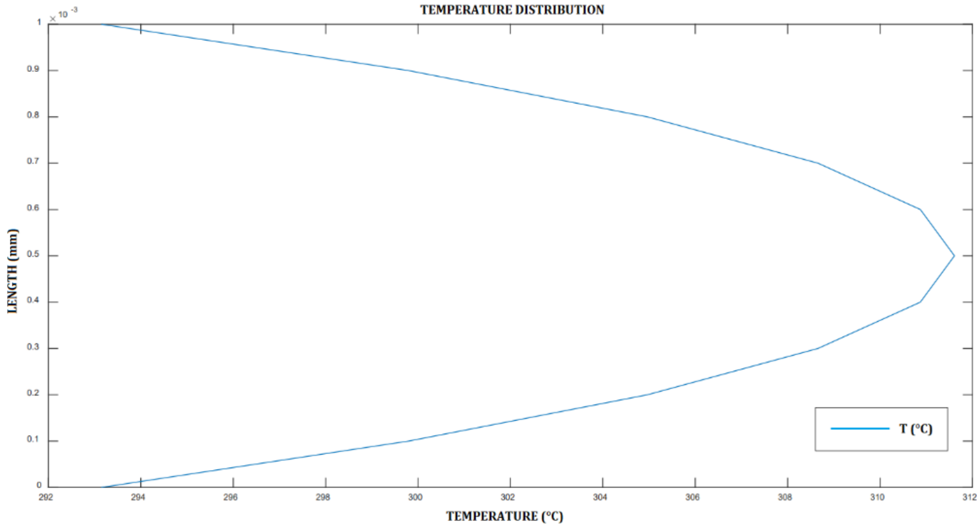

Figure 5. Temperature distribution between the plates according to the analytical solution.

Figure 6 visualizes the temperature distribution obtained from ANSYS Fluent, using a color scale and including velocity vectors. This figure clearly shows that the temperature is highest (red) in the center of the channel between the plates and decreases (green and blue) towards the plates. The fact that the temperature distribution does not change in the flow direction (x-direction) confirms that the flow is also thermally fully developed.

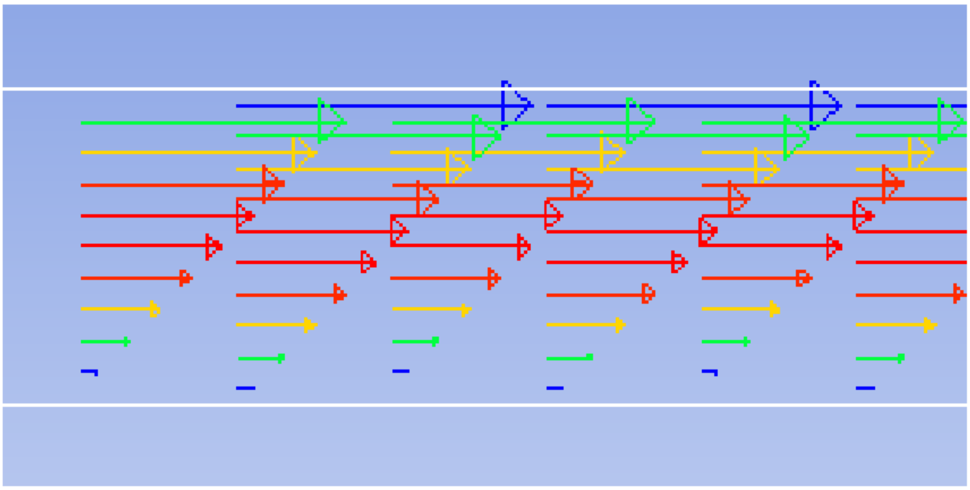

Figure 6. Temperature distribution between the plates according to the simulation results.

Figure 7 compares the analytical and numerical solutions for the temperature profile resulting from viscous heating in Couette flow. This graph displays the temperature (°C) as a function of the distance between the plates (y-coordinate).

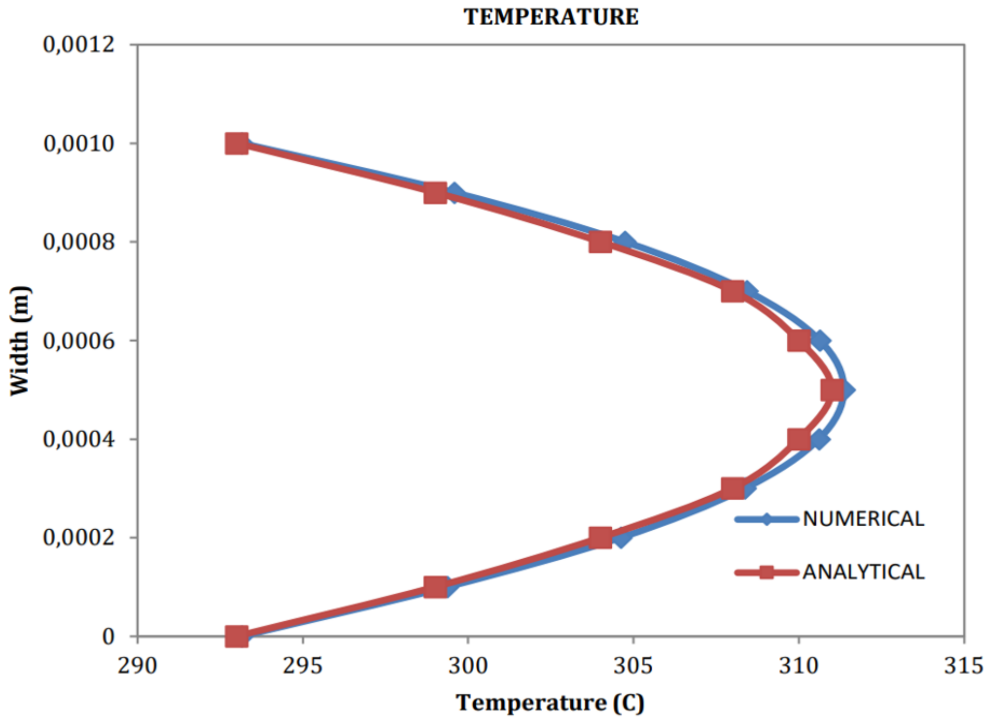

Figure 7. Comparison of analytical and numerical solutions for the temperature profile in Couette flow.

The analytical (red squares) and numerical (blue diamonds) solutions show excellent agreement with respect to the temperature profile. Both solutions satisfy the boundary condition of a 20 °C plate temperature and indicate that the maximum temperature (approximately 311-312 °C) occurs at the midpoint between the plates ($y = \frac{L}{2}$). When the power value obtained from ANSYS Fluent was compared with the power value obtained from the analytical solution, it was found to be in good agreement and provided an additional criterion to evaluate the accuracy of the numerical model.

5. Discussion

This study investigated the problem of viscous heating in Couette flow between two parallel plates maintained at a constant temperature, employing both analytical and numerical (ANSYS Fluent) methods. The results demonstrate that viscous heating causes a significant increase in oil temperature, an effect that cannot be neglected in the design and performance analysis of lubrication systems.

The velocity profiles, shown in Figure 4, confirm the expected linear distribution characteristic of Couette flow. This linearity indicates that the momentum of the moving upper plate is transferred to the lower plate through the fluid's viscosity, creating a constant velocity gradient. The general agreement between the analytical and numerical solutions confirms that both methods accurately capture this fundamental flow characteristic. However, minor deviations observed, particularly in the central region of the channel and near the upper plate, point to potential areas for improvement in the numerical model. These discrepancies may be attributed to insufficient mesh density, the choice of solution algorithm, or uncertainties in the implementation of boundary conditions. Future studies could investigate the source of these deviations in more detail by using more refined meshes and exploring different solution algorithms.

The temperature profiles, presented in Figure 7, reveal the most pronounced effect of viscous heating. Both the analytical and numerical solutions show a parabolic temperature distribution between the plates, with the maximum temperature occurring at the midpoint (y = L/2). This is a direct consequence of viscous heating being most intense in the region of highest velocity gradient (and thus, shear stress). The almost perfect agreement between the temperature profiles obtained from the analytical and numerical solutions provides strong evidence that both the analytical model (under the given assumptions) and the numerical model (with its specific ANSYS Fluent settings, mesh structure, etc.) accurately represent the viscous heating problem.

The results clearly show that viscous heating can significantly increase the oil temperature, especially in lubrication systems operating at high speeds and with narrow gaps. In this study, with a plate speed of 8 m/s and a plate separation of 1 mm, an increase in oil temperature of approximately 90 °C was observed. Such a substantial temperature rise can reduce the oil's viscosity, leading to a thinner lubrication film and potentially causing thermal degradation of the oil. This, in turn, can increase friction and wear, negatively affecting system performance and potentially leading to failures.

Therefore, engineers must consider the effects of viscous heating when designing lubrication systems and implement appropriate measures for temperature control. This study provides a simple yet effective analytical model that can be used for making temperature predictions during the design phase and for improving energy efficiency. Furthermore, it demonstrates that numerical modeling tools, such as ANSYS Fluent, can be reliably used for viscous heating analyses in more complex geometries and flow conditions.

Although the problem of viscous heating in Couette flow has been extensively studied in the literature, this work offers a unique contribution to the literature by using both analytical and numerical methods in conjunction and comparing the obtained results. Many studies in the literature have focused either solely on analytical or solely on numerical methods, and often with simplified models or specific boundary conditions. This study, however, combines the strengths of both approaches, providing a more comprehensive understanding of the viscous heating problem. There are also some limitations to this study. In the analytical model, it was assumed that the viscosity and thermal conductivity of the oil did not change with temperature. However, in real lubrication systems, the viscosity of the oil can decrease significantly with increasing temperature. This can affect both the velocity and temperature profiles. Furthermore, this study considered only a two-dimensional, steady-state, and fully developed flow situation. In real systems, three-dimensional effects, turbulence, and time-dependent changes may also be important.

6. Conclusion

This study comprehensively investigated the effects of viscous heating in Couette flow between two parallel plates maintained at a constant temperature, using both analytical and numerical (ANSYS Fluent) methods. The results obtained highlight the following key findings and implications:

In Couette flow, the mechanical energy of the moving plate is converted into heat due to the fluid's viscosity (viscous heating). This phenomenon leads to a significant increase in oil temperature, particularly in lubrication systems operating at high speeds and with narrow gaps. In this study, with a plate speed of 8 m/s and a plate separation of 1 mm, an oil temperature increase of approximately 90 °C was observed. The linear velocity profile characteristic of Couette flow was confirmed by both the analytical and numerical solutions. This indicates that the fluid's viscosity transmits the momentum of the moving plate towards the stationary plate, creating a linear velocity gradient. Both the analytical and numerical solutions showed a parabolic temperature distribution resulting from viscous heating. The maximum temperature occurred at the midpoint between the plates, calculated as 311.54 °C analytically and approximately 311-312 °C numerically. The velocity and temperature profiles obtained from the analytical and numerical solutions showed excellent agreement, especially for the temperature profile. Minor deviations were observed in the velocity profile, particularly in the central region of the channel and near the upper plate. These deviations may be attributed to factors such as mesh resolution, the solution algorithm, and the implementation of boundary conditions in the numerical model.

The results of this study emphasize the importance of considering viscous heating effects when designing lubrication systems. Elevated temperatures can alter the properties of the oil, negatively impacting system performance and potentially leading to failures. This work provides a simple yet effective analytical model that can be used for making temperature predictions during the design phase and for improving energy efficiency. Furthermore, it demonstrates that numerical modeling tools, such as ANSYS Fluent, provide reliable results for viscous heating analyses in more complex geometries and flow conditions. Future studies should focus on incorporating more realistic models that account for the temperature dependence of oil viscosity and thermal conductivity, as well as investigating turbulent flow conditions. In conclusion, this study has elucidated the fundamental characteristics of the viscous heating problem in Couette flow, both theoretically and numerically, and has provided valuable insights for the design and analysis of lubrication systems.

Acknowledgement
I would like to thank Sinan Dölek for his support and guidance in preparing this first article.

Conflicts of interest
The author declares no conflicts of interest.

References

1. Khonsari, M. M., & Booser, E. R. (2017). *Applied Tribology: Bearing Design and Lubrication*. John Wiley & Sons.
2. Davidson, I. (2023). *Biscuit Baking Technology: Processing and Engineering Manual* (2nd ed.). Elsevier.
3. Kyrkou, M. E., & Nikolakopoulos, P. G. (2020). Simulation of thermo-hydrodynamic behavior of journal bearings, lubricating with commercial oils of different performance. *Simulation Modelling Practice and Theory*, 104, 102128.
4. White, F. M., & Majdalani, J. (2006). *Viscous Fluid Flow* (Vol. 3, pp. 433-434). New York: McGraw-Hill.
5. Mehrizi, A. A., Besharati, F., Jahanian, O., & Hassanzadeh Afrouzi, H. (2021). Numerical investigation of conjugate heat transfer in a microchannel with a hydrophobic surface utilizing nanofluids under a magnetic field. *Physics of Fluids*, *33*(5).
6. Bird, R., Stewart, W., & Lightfoot, E. (2007). *Transport Phenomena* (revised 2nd ed.) John Wiley & Sons. New York.
7. Rajagopal, K. R., Saccomandi, G., & Vergori, L. (2011). Couette flow with frictional heating in a fluid with temperature and pressure dependent viscosity. *International Journal of Heat and Mass Transfer*, 54(4), 783-789.
8. de Campos, M. D., Romão, E. C., & Mendes de Moura, L. F. (2014, April). Numerical investigation of the viscous dissipation term on 2D heat transfer. *In Defect and Diffusion Forum* (Vol. 348, pp. 279-284). Trans Tech Publications Ltd.
9. Al-Mubaiyedh, U. A., Sureshkumar, R., & Khomami, B. (2002). The effect of viscous heating on the stability of Taylor–Couette flow. *Journal of Fluid Mechanics*, 462, 111-132.
10. ANSYS, Inc. (2023). *ANSYS Fluent Theory Guide*.
11. Incropera, F. P. (2007). *Fundamentals of Heat and Mass Transfer* (6th ed.). John Wiley & Sons.
12. Yunus, A. C. (2010). *Fluid Mechanics: Fundamentals and Applications (Si Units)*. Tata McGraw Hill Education Private Limited.
13. Kakac, S., Yener, Y., & Pramuanjaroenkij, A. (2013). *Convective Heat Transfer*. CRC press.