Multidisciplinary | Peer-Reviewed
Open Access | Research Journal

Article ID: IRJIST-2026-019
Volume 1 | Issue 2 | 2026

International Research Journal of Innovation in Science and Technology

CFD Analysis of Heat Transfer in a Throttling Needle Valve

Atharva Pacharne[1], Devesh Balwantrao[2]

[1-2]Research Scholar, Vishwakarma Institute of Information Technology, Pune, India
Correspondence: [1]atharva.22310339@viit.ac.in

Abstract

This study presents a Computational Fluid Dynamics (CFD)-based analysis of heat transfer in a throttling needle valve under steady-state conditions. When fluid flows through a partially restricted passage, a sharp pressure drop and velocity increase occur, resulting in significant viscous dissipation and localized heat generation. The objective is to quantify thermal effects, evaluate their spatial distribution, and examine their influence on fluid properties and valve performance. A three-dimensional CFD model was developed using ANSYS Fluent with a pressure-based solver, k–ω SST turbulence model, and energy equation including viscous dissipation. Water was used as the working fluid with an inlet pressure of 10 psi, outlet pressure of 0 psi, and 30% valve opening. Results indicate a peak temperature of approximately 79.5 °C at the needle tip and orifice, along with a fluid temperature rise of 5–10 °C and a mass flow rate of 547.95 kg/s. Turbulent flow behavior with recirculation zones was observed. These findings emphasize the necessity of incorporating thermal considerations in the design of high-pressure throttling systems.

Keywords: CFD, Throttling Valve, Needle Valve, ANSYS Fluent.

1. Introduction

Throttling valves are widely used in hydraulic systems, refrigeration cycles, and chemical process industries for flow and pressure regulation [1][4]. Among these, needle valves provide fine control by varying the effective flow area through axial movement of a tapered needle.

Although throttling is primarily intended for pressure reduction, it inherently involves energy dissipation, which manifests as heat within both the fluid and valve components [2][9]. This phenomenon is often underestimated during design, despite its strong influence on system performance.

Heat generation during throttling is mainly caused by viscous dissipation, where mechanical energy is irreversibly converted into thermal energy due to velocity gradients within the flow [5][7]. Additionally, turbulence downstream of the restriction further enhances energy loss and heat production. These effects are most prominent near the orifice and needle tip, where velocity gradients are highest.

Thermal effects introduce several engineering challenges. Variations in fluid temperature alter viscosity and density, directly impacting flow behavior and pressure drop characteristics [6]. Simultaneously, temperature gradients within the valve body induce thermal stresses, potentially affecting dimensional tolerances and sealing performance. Such issues are critical in precision applications such as fuel injection systems and control valves [3][8].

Due to the complexity of three-dimensional turbulent flow with coupled heat transfer, analytical solutions are impractical. CFD has therefore emerged as a reliable tool for predicting valve performance and internal flow characteristics with high spatial resolution [1][10]. This study focuses on simulating heat generation and transfer in a partially open needle valve using ANSYS Fluent, with emphasis on identifying thermal hotspots and their impact on flow behavior.

2. Methodology

A 1-inch needle valve was modeled, consisting of key components including the valve body, needle, orifice, bonnet, handle, locking nut, and screw thread. The needle moves axially to restrict the annular flow passage near the orifice.

The primary heat generation zones are the needle tip and orifice due to minimal flow area, high velocity, and elevated shear stress. A valve opening of 30% was considered. The scope of the present CFD investigation and its key objective areas are summarized in Table 1.

Table 1: *Summary of study objectives and analysis scope for the CFD investigation of the throttling needle valve.*

Objective Category	Description
Flow Analysis	To study internal flow characteristics such as velocity distribution, turbulence, and pressure drop across the valve
Thermal Analysis	To evaluate temperature distribution within the valve body and fluid domain
Viscous Heating Study	To quantify heat generation due to pressure loss and velocity gradients
Material Response	To analyze thermal stress and deformation of the needle caused by temperature rise
Fluid Property Variation	To examine how temperature affects viscosity and its impact on flow behavior
Performance Evaluation	To assess how thermal effects influence valve efficiency and pressure regulation
Design Validation	To verify whether the valve operates within safe thermal limits under given conditions

2.1. Governing Equations

The simulation solves the Reynolds-Averaged Navier–Stokes (RANS) equations along with the energy equation. The governing equations include:

- Continuity equation (incompressible flow)
- Momentum equations
- Energy equation with viscous dissipation

The dissipation function accounts for the conversion of mechanical energy into heat due to velocity gradients. The k–ω SST turbulence model was selected due to its ability to accurately predict flow separation and adverse pressure gradient effects in throttling regions.

Viscous Dissipation Function (Φ)

The viscous dissipation function, representing the conversion of mechanical energy into thermal energy due to viscous effects, is expressed as:

$$\Phi = \mu\left[2\left(\left(\frac{\partial u}{\partial x}\right)^2+\left(\frac{\partial v}{\partial y}\right)^2+\left(\frac{\partial w}{\partial z}\right)^2\right)+\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)^2+\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)^2+\left(\frac{\partial w}{\partial x}+\frac{\partial u}{\partial z}\right)^2-\frac{2}{3(\nabla\cdot\vec{V})^2}\right]$$

Turbulence Model Equations (k–ω SST)

- **Turbulent Kinetic Energy (k) Equation**

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho U_j k)}{\partial x_j}=P_k-\beta^*\rho k\omega+\frac{\partial}{\partial x_j}\left[\frac{(\mu+\sigma_k\mu_t)\,\partial k}{\partial x_j}\right]$$

- **Specific Dissipation Rate (ω) Equation**

$$\frac{\partial(\rho\omega)}{\partial t}+\frac{\partial(\rho U_j\omega)}{\partial x_j}=\alpha\frac{\omega}{k}P_k-\beta\rho\omega^2+\frac{\partial}{\partial x_j}\left[\frac{(\mu+\sigma_\omega\mu_t)\,\partial\omega}{\partial x_j}\right]+2(1-F_1)\rho\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial\omega}{\partial x_j}$$

2.2. Meshing and Boundary Conditions

A Poly-Hexcore meshing approach was used, combining polyhedral cells in complex regions with structured hexahedral cells in bulk flow zones.

This method provides:

- Improved accuracy in complex geometries
- Better boundary layer resolution
- Faster convergence
- Reduced computational cost

Boundary conditions:

- Inlet pressure: 10 psi
- Outlet pressure: 0 psi
- No-slip condition at walls
- Conjugate heat transfer enabled
- Steady-state, incompressible flow.

The computational mesh generated using the Poly-Hexcore method is illustrated in Figure 1.

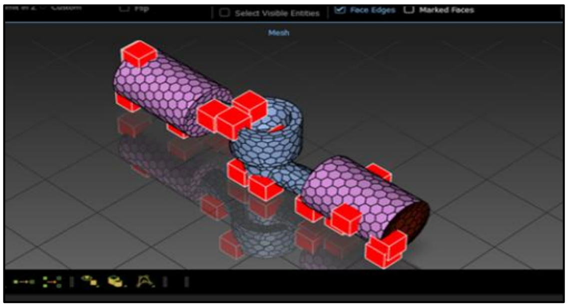

Figure 1: *Poly-Hexcore mesh of the throttling*

needle valve geometry, showing polyhedral elements in complex regions near the orifice and needle tip, and structured hexahedral cells in bulk flow regions for improved accuracy and computational efficiency.

3. Results

The CFD simulation of the throttling needle valve provides detailed insight into the coupled behavior of pressure, velocity, temperature, and turbulence within the flow domain. The results highlight the presence of strong gradients in the vicinity of the orifice and needle tip, which govern both flow characteristics and thermal effects.

3.1. Convergence Graph

The convergence history of the simulation is presented in Figure 2.

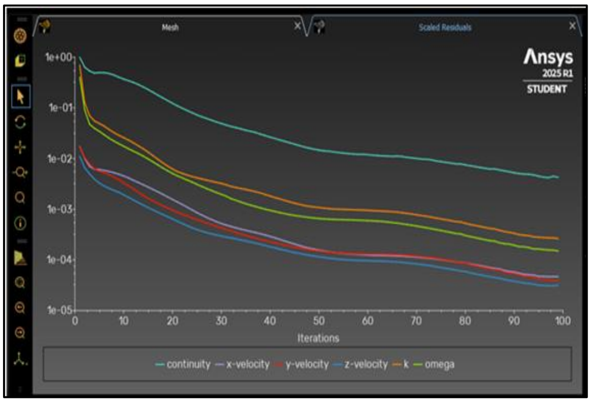

***Figure 2:** Convergence graph showing residual decay of continuity, momentum, energy, and turbulence equations over iterations.*

The residuals of continuity, momentum, energy, and turbulence equations exhibit a steady and monotonic decrease with increasing iterations. All residuals fall below the prescribed threshold of 10^{-4}, indicating numerical stability and proper convergence of the solution.

The smooth decay trend without oscillations confirms that the selected solver settings, turbulence model, and meshing strategy are appropriate for capturing the flow physics accurately.

This ensures that the results obtained are reliable and not affected by numerical divergence or instability.

3.2. Pressure Contours

The pressure distribution within the valve is illustrated in Figure 3.

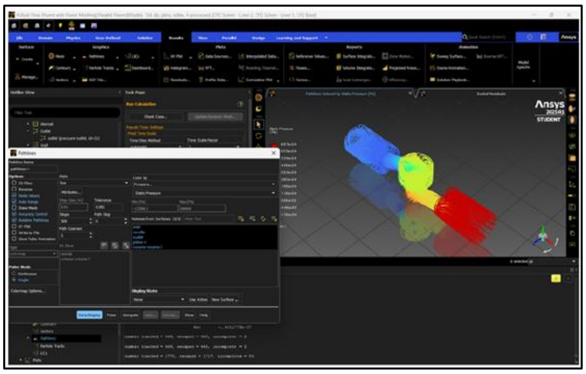

***Figure 3:** Pressure contour illustrating the sharp pressure drop across the throttling region near the orifice.*

A relatively uniform high-pressure region is observed upstream of the valve, followed by a sharp pressure drop at the orifice. This behavior is characteristic of throttling flow, where pressure energy is rapidly dissipated as the fluid passes through a restricted area.

Downstream of the needle tip, the pressure stabilizes near the outlet condition. The steep pressure gradient in the throttling region confirms that the orifice acts as the primary zone of energy dissipation. This localized pressure drop is directly responsible for viscous heating observed in the system.

3.3. Turbulence Contour

Figure 4 presents the turbulence distribution within the valve.

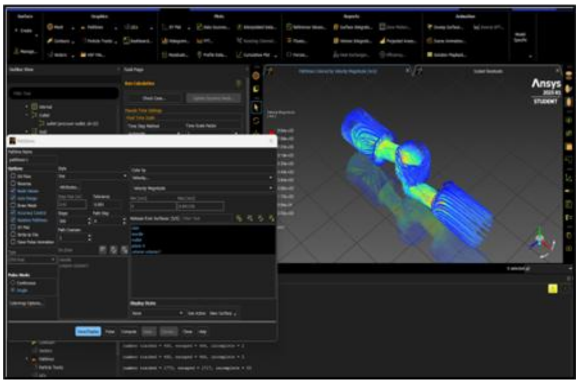

***Figure 4:** Turbulence contour showing regions of high turbulence intensity and recirculation downstream of the needle tip.*

High turbulence intensity is observed in the vicinity of the orifice due to strong velocity gradients. As the fluid expands downstream, recirculation zones are formed, indicating flow separation and mixing.

These recirculation regions enhance energy dissipation and contribute significantly to heat generation within the fluid. The results validate the selection of the k–ω SST turbulence model, which is capable of accurately capturing near-wall behavior and separated flow structures.

3.4. Thermal Stress Distribution

The thermal stress distribution within the valve structure is shown in Figure 5.

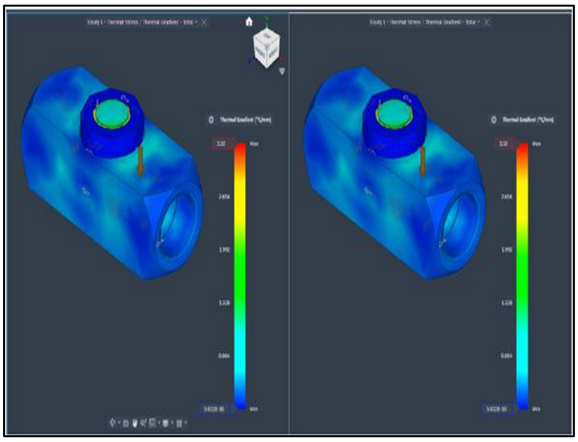

Figure 5: *Thermal stress distribution within the valve body, showing localized high-gradient regions near the orifice and needle tip due to viscous heating effects.*

The highest thermal gradients are concentrated near the orifice and needle tip, where the temperature rise is most significant.

A maximum thermal gradient of approximately 3.32 °C/mm is observed in these regions. This localized heating can induce differential thermal expansion between the needle and valve body, potentially affecting sealing performance and structural integrity.

The rest of the valve body experiences relatively low thermal gradients, indicating that thermal effects are highly localized rather than uniformly distributed.

3.5. Quantitative Results

The key numerical results obtained from the simulation are summarized in Table 2.

Table 2: *Summary of key CFD simulation results including pressure drop, temperature rise, flow behavior, and thermal effects.*

Parameter	**Value / Observation**	**Remarks**
Pressure Drop	~10 psi	Major pressure drop occurs across the orifice (throttling effect)
Maximum Valve Temp.	79.5 °C (≈ 80 °C)	Peak temperature observed at needle tip and orifice region
Temp. Rise (Fluid)	5–10 °C	Caused by viscous dissipation and frictional effects
Mass Flow Rate	547.95 kg/s	High acceleration due to reduced flow area
Flow Behavior	Turbulent	Recirculation zones present near needle region
Thermal Effect	Significant	Influences viscosity variation and pressure drop

The results presented in Table 2 confirm that throttling induces significant energy dissipation within the valve, primarily concentrated at the orifice region. This energy conversion manifests as both pressure loss and localized heat generation, which in turn alters fluid properties and flow behavior. The combined influence of turbulence, viscous effects, and thermal gradients governs the overall performance of the valve. These interactions highlight the importance of considering coupled thermo-fluid effects rather than relying solely on pressure-based analysis for accurate prediction of valve behavior.

4. Discussion

The results align with viscous dissipation theory, where heat generation scales with the square of velocity gradients. The orifice region, having steep gradients, acts as the primary heat generation zone. The temperature rise from 25 °C to 79.5 °C corresponds to a reduction in fluid viscosity by approximately 50–60%, significantly affecting flow resistance and pressure drop.

Temperature-dependent fluid properties introduce feedback mechanisms: as viscosity decreases, flow resistance reduces, altering pressure distribution. This highlights the limitations of isothermal CFD models in accurately predicting valve performance.

Thermal gradients lead to differential expansion in valve components, potentially causing sealing degradation and leakage. Therefore, materials with low thermal expansion coefficients, such as hardened stainless steel or Inconel, are recommended.

The k–ω SST model demonstrates superior accuracy for predicting separation and near-wall behavior. Residual convergence below 10^{-4} confirms numerical stability.

5. Conclusions

This study demonstrates that throttling in a needle valve is governed by coupled thermo-fluid effects rather than pressure reduction alone. CFD results reveal that viscous dissipation at the orifice and needle tip generates localized thermal hotspots, with peak temperatures reaching ~79.5 °C and fluid temperature rise of 5–10 °C. These thermal effects significantly reduce fluid viscosity, altering flow resistance and pressure characteristics, thereby highlighting the limitations of isothermal assumptions in valve analysis.

The presence of steep thermal gradients (~3.32 °C/mm) indicates potential risks of localized thermal stress and material deformation, which may affect sealing performance and long-term reliability. Turbulent flow structures and recirculation zones further intensify energy dissipation and heat generation.

Overall, the findings establish that accurate prediction of needle valve performance requires integrated thermal analysis, making it a critical consideration in the design of high-pressure and precision flow control systems.

References

[1] Davis, J. A., and Stewart, M., “Predicting Globe Control Valve Performance – Part I: CFD Modeling,” Journal of Fluids Engineering, Transactions of the ASME, vol. 124, no. 3, pp. 772–777, 2002. https://doi.org/10.1115/1.1490108

[2] “Determination of the Valve Sizing Coefficient of Globe Valve Using CFD,” (n.d.).

[3] Filo, G., Lisowski, E., and Rajda, J., “Design and Flow Analysis of an Adjustable Check Valve by Means of CFD Method,” Energies, vol. 14, no. 8, pp. 1–14, 2021. https://doi.org/10.3390/en14082237

[4] Gjerver, A., Frostad, R., Barisic, V., Hodkiewicz, M., Woods, C., Fekete, M., Braathen Torjusen, A., and Wilhelm Kluwer, J., “Machine-Interpretable Engineering Design Standards for Valve Specification,” IEEE Access, vol. 14, pp. 3471–3490, 2026. https://doi.org/10.1109/ACCESS.2025.3643528.

[5] Grace, A., and Frawley, P., “Experimental Parametric Equation for the Prediction of Valve Coefficient (Cv) for Choke Valve Trims,” International Journal of Pressure Vessels and Piping, vol. 88, no. 2–3, pp. 109–118, 2011. https://doi.org/10.1016/j.ijpvp.2010.11.002

[6] Herakovič, N., Duhovnik, J., and Šimic, M., “CFD Simulation of Flow Force Reduction in Hydraulic Valve,” Tehnički Vjesnik, vol. 22, no. 2, pp. 453–463, 2015. https://doi.org/10.17559/TV-20141128090939.

[7] Khowal, R., Jadhav, D. N., and Chaudhari, T., “Flow Coefficient Valve Calculation Using CFD Analysis for Needle Valve,” *International Journal on Theoretical and Applied Research in Mechanical Engineering (IJTARME)*, Special Issue, Vol. 5, No. 2, 2016.

[8] Koyunbaba, E., “Computational Fluid Dynamics Application for Determining Flow Characteristics of Valves,” 2008.

[9] Rahmeyer, W., and Driskell, L., “Control Valve Flow Coefficients,” (n.d.).

[10] Rezaey, S., “Numerical Investigation of a Globe Control Valve and Estimation of Its Loss Coefficient at Different Opening States,” European Journal of Computational Mechanics, vol. 29, no. 4–6, pp. 549–576, 2020. https://doi.org/10.13052/ejcm1779-7179.294610.

[11] Scuro, N. L., Angelo, E., Angelo, G., and Andrade, D. A., “CFD Analysis of the Flow Dynamics of a Directly-Operated Safety Relief Valve,” Nuclear Engineering and Design, vol. 328, pp. 321–332, 2018. https://doi.org/10.1016/j.nucengdes.2018.01.024

Publisher’s Note & Copyright