

DOI: 10.24850/j-tyca-18-03-06

Articles

Empirical analysis and CFD simulation on energy loss coefficient efficiency of contraction flow pipe systems
Modelado empírico y simulación CFD sobre la eficiencia del coeficiente de pérdida de energía en sistemas de tuberías de flujo por contracción

Moh Abduh¹, ORCID: <https://orcid.org/0000-0002-1831-381X>

Fahma D. Arjunanta², ORCID: <https://orcid.org/0009-0000-9685-8789>

M Regan A. Putra³, ORCID: <https://orcid.org/0009-0001-1610-0896>

Azhar A. Darmawan⁴, ORCID: <https://orcid.org/0000-0001-9491-7570>

Lourina E. Orfa⁵, ORCID: <https://orcid.org/0000-0003-4869-215X>

¹University of Muhammadiyah Malang, Indonesia, abduh@umm.ac.id

²University of Muhammadiyah Malang, Indonesia, fahmastudy@gmail.com

³University of Muhammadiyah Malang, Indonesia, reganandika33@gmail.com

⁴University of Muhammadiyah Malang, Indonesia, azharadidfts@umm.ac.id

⁵University of Muhammadiyah Malang, Indonesia, lourinaorfa@umm.ac.id



Author for correspondence: Moh Abduh, abduh@umm.ac.id

Abstract

Pressure loss in pipe systems directly affects the efficiency and sustainability of water distribution networks. Pipe contraction is a major source of local pressure loss, resulting from a sudden or gradual diameter reduction that induces flow separation, increased turbulence, and pressure drop. This study presents an empirical analysis and Computational Fluid Dynamics (CFD) modelling to evaluate the pressure-loss coefficient of contraction pipes under turbulent flow conditions. This study investigated two contraction configurations: sudden diameter reduction and gradual contraction with varying cone lengths (L). The CFD results were compared with empirical calculations and validated against previously published experimental data. The simulations reproduced the general hydraulic trends, with mesh refinement improving numerical accuracy. The results demonstrate that contraction geometry, particularly the contraction ratio and cone length, significantly influences hydraulic efficiency and pressure loss. A gradual contraction with $L > 1D_i$ and a higher contraction ratio reduced the pressure-loss coefficient by up to 25% compared with sudden contraction. Longer contraction lengths also reduced downstream pressure disturbances and the potential for cavitation. These findings indicate that optimising contraction geometry can improve hydraulic performance while reducing energy consumption and overall system costs. The results provide practical guidance for the design of efficient and sustainable water distribution systems and support the application of CFD-assisted optimisation in contraction-pipe design.



Keywords: hydraulic engineering, hydrodynamics, fluid dynamics, mathematical models, computational fluid dynamics, CFD.

Resumen

La pérdida de presión en los sistemas de tuberías afecta directamente a la eficiencia y la sostenibilidad de las redes de distribución de agua. La contracción de las tuberías es una de las principales fuentes de pérdida de presión local y se debe a una reducción repentina o gradual del diámetro que provoca la separación del flujo, un aumento de la turbulencia y una caída de presión. Este estudio presenta un análisis empírico y un modelo de dinámica de fluidos computacional (CFD) para evaluar el coeficiente de pérdida de presión de las tuberías con contracción en condiciones de flujo turbulento. En este estudio se investigaron dos configuraciones de contracción: la reducción brusca del diámetro y la contracción gradual con longitudes de cono (L) variables. Los resultados de la CFD se compararon con cálculos empíricos y se validaron contrastándolos con datos experimentales publicados anteriormente. Las simulaciones reprodujeron las tendencias hidráulicas generales y el refinamiento de la malla mejoró la precisión numérica. Los resultados demuestran que la geometría de la contracción, en particular la relación de contracción y la longitud del cono, influye significativamente en la eficiencia hidráulica y la pérdida de presión. Una contracción gradual con $L > 1D_i$ y un mayor índice de contracción redujo el coeficiente de pérdida de presión hasta en un 25 % en comparación con una contracción brusca. Las longitudes de contracción más largas también redujeron las perturbaciones de presión aguas abajo y el riesgo de cavitación. Estos hallazgos indican que la optimización de la geometría de la contracción

puede mejorar el rendimiento hidráulico al tiempo que reduce el consumo de energía y los costes generales del sistema. Los resultados proporcionan una guía práctica para el diseño de sistemas de distribución de agua eficientes y sostenibles, y respaldan la aplicación de la optimización asistida por CFD en el diseño de tuberías de contracción.

Palabras clave: ingeniería hidráulica, hidrodinámica, dinámica de fluidos, modelo matemático, dinámica de fluidos computacional, CFD.

Received: 07/04/2026

Accepted: 31/07/2026

Published ahead of print: 23/09/2026

1. Introduction

Fluid flow behavior in pipe network systems is a fundamental aspect of fluid mechanics engineering, particularly in clean water distribution networks, industrial piping, and energy transport systems. One of the main factors influencing the performance of piping systems is pressure reduction, which directly affects pumping power, operational costs, and the overall efficiency of the piping system (Bakri et al., 2022; Dadar et al., 2021; Meseret et al., 2024; Ren et al., 2025; Souza et al., 2023). The losses that occur are generally associated with the pressure loss coefficient, which represents the dissipation of mechanical energy caused by friction and other local disturbances that cannot be recovered within a pipe system (Abduh et al., 2024, 2025). Accurate prediction of the pressure loss coefficient is therefore essential for the reliable design and optimization of contemporary fluid distribution infrastructure.



Local pressure loss becomes particularly significant in the presence of geometric discontinuities, such as contractions, expansions, bends, and fittings. Pipe contraction specifically occurs when fluid flows from a pipe with a larger diameter to a smaller cross-section, causing increased “water velocity”, boundary layer separation, and the formation (Mugundhan & Thoroddsen, 2023) of a vena-contracta. These phenomena can cause significant pressure drops and energy losses (head loss) that affect the downstream network (Ho et al., 2023). Although contraction geometry appears simple, the resulting flow behavior is highly complex and sensitive to both geometric parameters and flow parameters (Wang et al., 2021). When planning a fluid network, an engineer must consider many factors. In addition to increasing the flow capacity, it is also necessary to consider the efficiency of pressure loss (head loss) within the network (Altowayti et al., 2021).

Several factors, including contraction ratio, contraction geometry, Reynolds number, and surface roughness (Cao et al., 2026), influence the energy loss coefficient associated with contraction flow (Durmuş, 2024). The implementation of conventional design relies heavily on empirical correlations derived from experiments conducted under limited flow conditions and simple geometries. Although widely applied, modern piping systems exhibit more heterogeneous flow variations, and design efficiency has become a primary consideration.

To address these limitations, numerical modelling and Computational Fluid Dynamics (CFD) serve as essential tools for analyzing internal flow in pipe systems. CFD enables detailed analysis of velocity profiles, pressure distribution, turbulence structures, and energy dissipation mechanisms through the application of the Navier–Stokes

equations. (Ennouri et al., 2019; Lee et al., 2025). This capability helps researchers identify complex flow phenomena, such as flow separation and recirculation, that are difficult to observe experimentally but are crucial for determining local pressure reduction.

Previous CFD-based studies have shown that pressure drops and pressure loss coefficients in contraction flows can be predicted with reasonable accuracy when appropriate turbulence models and mesh strategies are applied. Commonly used turbulence models include the standard $k-\varepsilon$, RNG $k-\varepsilon$, and SST $k-\omega$ models. Their performance varies considerably in capturing flow separation phenomena and flow behavior. However, inconsistencies in reported results indicate that the numerical efficiency and reliability of CFD predictions for pressure loss coefficients still depend strongly on modelling choices (Erhan, 2024; Farooq et al., 2024).

A literature review of 35 recent articles indicates that pipe contraction significantly affects fluid flow characteristics, particularly by increasing turbulence, flow separation, vortices, and energy loss (head loss). The sharper the contraction ratio, the more unstable the pressure distribution becomes, and the greater the energy loss coefficient becomes. Computational Fluid Dynamics (CFD) methods are capable of visualizing pressure distribution, recirculation zones, and turbulence patterns in detail, which are difficult to observe through conventional experiments. Turbulence models, such as the $k-\varepsilon$, $k-\omega$ SST, and Large Eddy Simulation (LES) models, are used to enhance the accuracy of turbulent flow simulations in pipes. However, discussions on pipe geometry models that enhance energy efficiency and implementation effectiveness remain limited.

Comparatively, articles based on fundamental theory, such as those published in the Journal of Fluid Mechanics and Physics of Fluids, place greater emphasis on turbulence mechanisms and mathematical validation, whereas journals such as Water, Sustainability, and Energies are more oriented towards energy efficiency and the sustainability of fluid distribution systems. Recent studies tend to integrate CFD simulation, experimental validation, and energy optimisation to develop piping system designs that are more efficient, stable, and sustainable for applications in water distribution, process industries, and marine and oil and gas pipeline systems. In detail, as described in the following table.

Table 1. Classification of Pipe Contraction Based on Analytical Focus, Evaluation Criteria, and Argument Characteristics

Authors	Article Analysis	Evaluation	Comparative
Abduh et al. (2024)	Examined pressure drop in circular sliced pipes for micro-hydropower turbine housing using geometric and hydraulic approaches.	Strong in geometric model development, although experimental and CFD validation remained limited.	Emphasized the influence of geometry on energy loss, but was simpler than modern LES or RANS approaches.
Alavi, A. et al. (2025)	Investigated flow behaviour and turbulence in marine systems using numerical CFD approaches.	Demonstrated strengths in complex simulations and robust numerical validation.	More comprehensive than simple experimental studies, considering large-scale turbulence interactions.
Altowayti, W. A. H. et al. (2021)	Discussed energy loss and pressure distribution in water flow piping systems.	Numerical modelling was adequate, although mesh sensitivity required further attention.	Consistent with studies reporting increased head loss due to sharp contractions.

Authors	Article Analysis	Evaluation	Comparative
Avci, M., & Papavassiliou, D. V. (2025)	Developed CFD simulations for turbulent flow analysis in varying pipe geometries.	Contributed to design optimisation, although field validation remained limited.	Focused more on numerical optimisation than traditional experimental studies
Bakri, B. et al. (2022)	Investigated the effects of diameter variation and contraction on pressure distribution.	Provided a useful preliminary assessment, although the study scale was relatively limited.	Consistent in showing increased turbulence caused by contractions.
CANLI, E., ATES, A., & BİLİR, Ş. (2021)	Examined energy loss characteristics in pipe joints and contractions.	Empirical methods were effective for practical engineering applications.	Simpler than CFD studies employing three-dimensional simulations.
Cao, L. et al. (2026)	Focused on discharge measurement and flow characteristics using modern instrumentation.	Measurement accuracy was the main strength of the study.	Differed from pure CFD studies by emphasizing experimental validation.
Che Sidik, N. A. et al. (2020)	Applied CFD to evaluate pressure distribution and vortices in contractions.	Simulations were detailed but highly dependent on turbulence model selection.	Supported the argument that contractions intensify flow separation and local turbulence.
Dadar, S. et al. (2021)	Investigated flow efficiency and head loss in water distribution systems	Highly relevant to sustainable distribution systems.	More applicable to clean water systems than fundamental turbulence studies.
Durmus, S. (2024)	Examined hydraulic phenomena in varying pipe diameters.	Provided fundamental experimental contributions.	Findings supported the Darcy–Weisbach theory and classical minor loss coeff.
Ennouri, M. et al. (2019)	Analyzed the effects of Reynolds number on pressure distribution & turbulence.	Mathematical and numerical approaches were comprehensive.	Theoretically stronger than simple experimental studies.

Authors	Article Analysis	Evaluation	Comparative
Erhan, Ö. (2024)	Used numerical simulations to investigate vortices and pressure drop.	Model validation enhanced the study's reliability.	Consistent with CFD studies showing turbulence dominance in sharp contractions.
Farooq, U., Li, S., & Yang, J. (2024)	Investigated the influence of pipe configuration on flow distribution efficiency.	Relevant to the development of sustainable distribution systems.	Supported earlier studies concerning contraction ratio optimisation.
González, J. et al. (2024)	Examined turbulent fluid behaviour in complex geometries using CFD.	Provided detailed modelling of vortices and recirculation zones.	More comprehensive than experimental studies due to enhanced flow visualization.
He, J., Li, C., & Guo, Y. (2023)	Developed CFD-based optimization approaches for piping systems.	Strong integration of simulation and industrial process optimization.	Differed from classical hydraulic studies through its industrial efficiency focus.
Ho, M. L. G. et al. (2023)	Discussed numerical analysis of pressure distribution and turbulent flow.	Offered practical numerical approaches for fluid engineering.	Aligned with CFD studies showing contraction-induced energy losses.
Kadivar, M., Tormey, D., & McGranaghan, G. (2023)	Investigated heat transfer and turbulent flow interactions in pipes.	Strong contribution to coupled thermal–fluid analysis.	More complex due to the inclusion of thermal effects.
Lee, H. et al. (2025)	Discussed flow dynamics in marine and fluid piping systems.	Broad applications in maritime engineering.	Differed from water distribution studies by considering marine environments.
Li, Y. et al. (2026)	Focused on flow efficiency and water quality in distribution systems.	Strong contribution to sustainable water management.	Emphasized water quality more than pure turbulence aspects
Liu, C. et al. (2025)	Investigated fluid flow in industrial oil and gas piping systems.	Relevant for high-pressure and complex fluid applications.	Differed from clean water studies by considering multiphase fluids.

Authors	Article Analysis	Evaluation	Comparative
Meftah Hrairi, Faical Baghdadi, & Waqar Asraar (2024)	Developed CFD simulations for contractions and flow direction changes.	Simulations were detailed but required further experimental validation.	Supported previous findings regarding increased vortices in sudden contractions.
Meseret, M., Wami, D., & Nallamothu, R. B. (2024)	Discussed turbulence phenomena and energy distribution using modern scientific approaches.	Strong in methodology and statistical validation.	Methodologically stronger than several conventional studies.
Moh Abduh, Khairul Iqbal, & Sulianto (2025)	Investigated pressure distribution and turbulent flow in specific geometries.	Contributed to numerical model development.	Consistent with flow separation theory due to sharp geometric changes.
Mugundhan, V., & Thoroddsen, S. T. (2023)	Focused on turbulence mechanisms & vortices in internal flow.	Theoretically and numerically robust.	More fundamental than applied water distribution studies.
Nohmi, M. et al. (2019)	Discussed turbulent flow characteristics in engineering piping systems.	Relevant for mechanical engineering design development.	Supported classical minor loss coefficient theory and turbulence dissipation.
Ren, H. et al. (2025)	Linked energy efficiency with sustainable piping systems.	Major contribution to sustainability engineering.	Broader than pure hydraulic studies due to environmental considerations.
Schatzman, D. M., & Thomas, F. O. (2017)	Investigated funda-mental turbulence phenomena in internal flow.	Extremely strong in fluid theory and mathematical validation.	Served as a key reference for many modern CFD studies.
Shakouri, B., Ismail, I., & Safari, M. J. S. (2023)	Examined the influence of flow systems on environ- mental and energy efficiency aspects.	Highly relevant to environmental concerns.	Differed from pure fluid mechanics studies by including pollution aspects.

Authors	Article Analysis	Evaluation	Comparative
Shrestha, U., & Choi, Y. D. (2020)	Developed process optimisation based on fluid flow systems.	Industrial application and process efficiency were the main strengths.	More applicable to industrial processes than domestic water distribution systems.
Song, X. et al. (2023)	Discussed turbulence behaviour and pressure distribution using fluid physics approaches.	Strong in fundamental turbulence analysis.	More theoretical than practical engineering studies.
Souza, T. R. C. et al. (2023)	Investigated energy efficiency in flow and piping systems.	Relevant to energy conservation and system optimization	Supported the argument that reducing head loss improves system energy efficiency.
Tang, X. et al. (2020)	Discussed energy loss in water distribution systems and geometric variations.	Applied research with contributions to distribution system design.	Consistent with Darcy–Weisbach theory and modern CFD approaches to minor losses.
Wang, H., Krüger, T., & Varnik, F.	Analyzed turbulent flow and vortex formation in contractions.	CFD visualization was a major strength.	Consistent with flow separation theory.
Wu, Y. et al. (2024)	Developed chemical engineering approaches for fluid flow systems.	Strong integration of simulation and industrial applications.	More industry-oriented than fundamental academic studies.
Xu, X. et al. (2021)	Investigated heat transfer and turbulent flow interactions in pipes.	Very strong in coupling heat transfer and fluid dynamics.	More complex due to the inclusion of thermal transfer effects.

A critical review of previous literature indicates that most studies still focus on the magnitude of pressure losses in sudden contraction configurations, with limited attention given to the efficiency of the pressure loss coefficient itself. In many cases, the pressure loss coefficient is still treated as a constant value, with little systematic evaluation of how



geometric modifications or flow regimes influence the efficiency of energy dissipation. Comparative analyses linking CFD-derived loss coefficients with energy-efficient pipe system design remain relatively limited, creating a clear gap between numerical analysis and practical engineering optimisation (Shrestha & Choi, 2020; Wu et al., 2024). The results obtained from the CFD simulations demonstrate a relevant trend, although some differences in values remain. This is because CFD simulations are highly influenced by mesh density within the modelling process; the finer and more numerous the elements, the more accurate and reliable the model becomes. From an economic perspective, project costs are strongly affected by the optimisation of the pipe geometry. The more optimised the contraction pipe design, the greater the efficiency achieved by reducing contraction losses.

The review of pressure reduction due to pipe contraction is not only related to friction and changes in flow direction. Another important aspect that must be considered is the occurrence of negative pressure or cavitation, which may cause damage to the pipe (Li et al., 2026; Liu et al., 2025; Nohmi et al., 2019). The phenomenon of cavitation caused by contraction or fluctuating pressure conditions can result in pipe damage or even structural failure. Therefore, careful planning and comprehensive mitigation measures are required (He et al., 2025; Tang et al., 2020).

From an energy and sustainability perspective, improving the efficiency of pressure loss coefficients in contraction flow systems is highly beneficial for reducing pump energy demand and enhancing the resilience of fluid transport infrastructure (Shakouri et al., 2023; Xu et al., 2021). Even small reductions in local energy losses can result in significant energy savings in large-scale pipe networks. Therefore, a more in-depth

numerical investigation, including the evaluation of its efficiency under various flow and geometric conditions, is highly relevant to contemporary engineering challenges.

Based on these considerations, the novelty of this study lies in the integrated evaluation of the efficiency of energy loss coefficients in pipe contraction flow systems using CFD simulations and practical methods through empirical analysis. This research systematically analyses the relationship between contraction geometry, flow structure, turbulence behaviour, and energy dissipation efficiency. The findings are expected to improve the predictive capability of loss coefficients and provide practical guidance for the design and optimization of energy-efficient pipe systems.

2. Methodology

The developed conceptual framework and hypotheses examine the contraction phenomena occurring in pipe networks, particularly pressure loss caused by turbulence induced by cavitation within pipe geometries featuring reduced dimensions. Geometrical elements considered include changes in flow direction at the inlet and outlet, as well as gradual cross-sectional transitions. Based on this conceptual framework and the proposed hypotheses, empirical data testing was conducted to address the formulated research problems in accordance with the stated concepts and hypotheses. The hypotheses were developed using an empirical model to determine the head loss coefficient (C_{hf}) in contraction flow pipes. Accurate evaluation of C_{hf} requires careful analysis of the effects of shear stress and flow direction changes. The principal parameters

analyzed include flow velocity, Reynolds number, pipe wall roughness, upstream and downstream pipe diameters, and contraction length.

This study employs empirical modelling and Computational Fluid Dynamics (CFD) simulation to investigate the efficiency of the energy loss coefficient in contraction flow pipe systems. The research methodology includes empirical equations, geometric modelling, boundary condition specification, and mesh generation, which are described in detail in the following subsections.

2.1. Geometric models

The geometric model presented illustrates a contraction pipe system, consisting of an upstream pipe with a larger diameter (D_i), a contraction section, and a downstream pipe with a smaller diameter (D_{i+1}). The contraction ratio $\beta = D_{i+1}/D_i$ is considered to evaluate its influence on flow behavior and pressure drop characteristics.

In predicting the pressure drop coefficient in contraction flow pipes, several factors influence the flow behavior, including fluid shear stress along the contraction section, pipe wall roughness, and changes in flow direction resulting from differences in the upstream and downstream pipe cross-sections. The associated parameters include the upstream pipe diameter (D_i), downstream pipe diameter (D_{i+1}), the change-of-direction angle (α) induced by the contraction or contraction gradient, and the length of the contraction section (L) as shown in Figure 1 below.

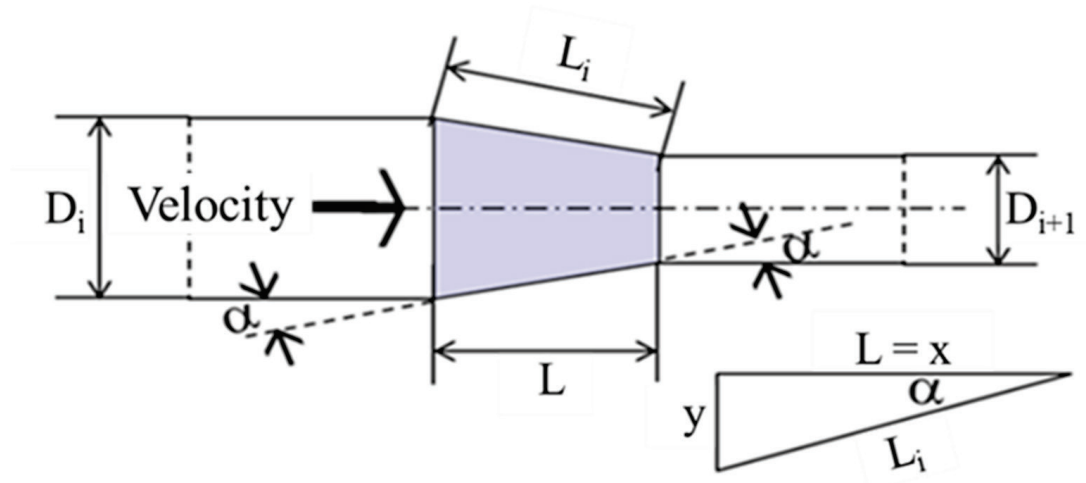


Figure 1. Geometry setup

The total pipe length is designed to ensure fully developed flow at both the inlet and outlet regions. The upstream straight pipe length is set to at least 10–15 times the upstream pipe diameter, while the downstream section is extended to 20 times the upstream pipe diameter, following recommendations from previous studies (Avci & Papavassiliou, 2026; CANLI et al., 2021).

The contraction geometry is modelled as a sudden contraction, which is commonly encountered in water distribution and industrial piping systems, and is known to produce significant local energy losses (Baghdadi et al., 2024).

2.2. Development of equations

The equations were developed based on concepts and hypotheses derived from an empirical approach to the geometric model. Changes in pipe dimensions, including diameter and length, have a significant impact

on the pressure drop coefficient, which ultimately affects the efficiency and effectiveness of water distribution systems. An excessively high pressure drop coefficient can lead to increased pumping energy requirements, reduced system efficiency, and higher operational costs. Therefore, a deeper understanding of the relationship between changes in pipe dimensions and the pressure drop coefficient is essential for designing efficient and sustainable water distribution systems.

The empirical equations developed based on the formulated hypotheses are described as follows:

$$y = \frac{(D_i - D_{i+1})}{2} \quad (1)$$

$$\alpha = \operatorname{tg}^{-1} \left[\frac{y}{x} \right] = \operatorname{tg}^{-1} \left[\frac{(D_i - D_{i+1})}{2L} \right] \quad (2)$$

$$L_i = \frac{L}{\cos \alpha} \quad (3)$$

where:

u = the velocity

D_i = the upstream pipe diameter

D_{i+1} = the downstream pipe diameter

α = the angle of the contracta-gradient

The fluid flow through the contraction pipe system is assumed to be two-dimensional, steady-state, incompressible, and turbulent. Under these assumptions, the flow behavior is governed by the continuity equation. The continuity equation is expressed as:

$$Q_i = Q_{i+1}; A_i \neq A_{i+1}; \text{ and } U_i \neq U_{i+1} \quad (4)$$

where:

Q = the discharge

A = the area of the pipe

U = the velocity of flow

The established energy loss coefficient equation is (C_{hf}):

$$C_{hf} = \delta_f + \delta_p \quad (5)$$

The loss coefficient is caused by friction along the pipe wall:

$$\delta_f = f \frac{L}{\cos \alpha} \left(\frac{rkD_i + rkD_{i+1}}{2} \right) \quad (6)$$

where:

f = the friction coefficient

L = the length of the contraction pipe

rkD = the circumference of the contraction pipe

The loss coefficient caused by a change in flow direction due to contraction:

$$\delta_p = 2[1 - \cos \alpha] \quad (7)$$

Therefore, the equation for pressure loss due to contraction flow (hf) is:

$$\Delta h_f = C_{hf} \frac{u^2}{2g} = \left[f \frac{L}{\cos \alpha} \left(\frac{r k D_i + r k D_{i+1}}{2} \right) + 2(1 - \cos \alpha) \right] \frac{u^2}{2g} \quad (8)$$

where:

hf = the pressure loss

C_{hf} = the head loss coefficient

The fluid flow through the contraction pipe system by CFD is assumed to be three-dimensional, steady-state, incompressible, and turbulent. Under these assumptions, the flow behavior is governed by the continuity equation and the Reynolds-Averaged Navier–Stokes (RANS) equations (Kadivar et al., 2023; Yusof et al., 2020).

The continuity equation is expressed as:

$$\nabla \cdot u = 0 \quad (9)$$

where:

u = the velocity vector

The momentum conservation equation for incompressible flow is written as:

$$\rho (u \cdot \nabla) u = -\nabla p + \mu \nabla^2 u - \rho \nabla \cdot (u') \quad (10)$$

where:

ρ = the fluid density

p = the static pressure

μ = the dynamic viscosity

u' = the Reynolds stress tensor

To close the RANS equations, the standard k - ε turbulence model is employed due to its robustness and proven reliability in internal pipe flow simulations involving contraction and separation phenomena (González et al., 2024; Song et al., 2023). The transport equations for turbulent kinetic energy k and its dissipation rate ε are given by:

$$\rho(u \cdot \nabla)k = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right] + G_k - \rho\varepsilon \quad (11)$$

$$\rho(u \cdot \nabla)\varepsilon = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + C_{1\varepsilon} \frac{\varepsilon}{k} G_k - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (12)$$

Where:

μ_t = the turbulent viscosity

G_k = the production of turbulent kinetic energy

2.3. Boundary conditions

The applied boundary conditions serve to accurately represent the physical behaviour of flow within the contraction pipe system. At the inlet, a uniform velocity profile is prescribed based on the planned flow rate. The turbulence

intensity is specified according to fully developed turbulent flow, typically between 5–10%, in accordance with CFD standards. (ANSYS, 2025).

At the outlet, a pressure outlet boundary condition with zero-gauge pressure is applied, allowing the flow to develop naturally. All pipe walls are treated as no-slip boundaries, with the fluid velocity at the wall set to zero. Wall roughness is neglected, assuming hydraulically smooth pipes, to isolate the effect of geometric contraction on energy loss.

The pressure loss coefficient due to contraction (C_{hf}) is calculated based on the pressure difference between the upstream and downstream sections using:

$$C_{hf} = \frac{\Delta h_f}{U_{i+1}^2 / 2g} \quad (13)$$

where

Δh_f = the local pressure loss

U_{i+1} = the mean velocity in the downstream pipe

g = the acceleration due to gravity

2.4. Mesh generation

Based on CFD, the computational system is discretized using a three-dimensional volumetric mesh, consisting of hexahedral elements in the straight pipe sections and refined polyhedral or tetrahedral elements in the contraction region, which serve to capture velocity gradients and flow separation accurately.

To achieve a refined model, the mesh is concentrated in the contraction zone and along the wall boundaries, where strong velocity and pressure gradients are expected. Inflation layers are applied along the pipe walls to accurately resolve the near-wall flow, ensuring that the dimensionless wall distance values are appropriate for the selected turbulence model.

Grid independence is assessed by comparing simulation results from coarse, medium, and fine meshes. The final mesh is selected when variations in pressure drop and pressure loss coefficient are less than 2%, thereby ensuring numerical accuracy while maintaining computational efficiency.

3. Results

3.1. Flow field characteristics in contraction pipe systems

The empirical analysis employed was based on concepts and hypotheses referring to flow phenomena in pipes with contraction pipe geometry models, as illustrated in Figure 1. This was derived from a simple yet effective representation of pressure drop. As shown in “Figure 2a”, various contraction pipe models with different contraction lengths produce realistic pressure drop values. Increasing the length of the contraction section enhances the effectiveness of pressure drop generation; however, excessively long contraction sections pose practical challenges for real-world implementation.

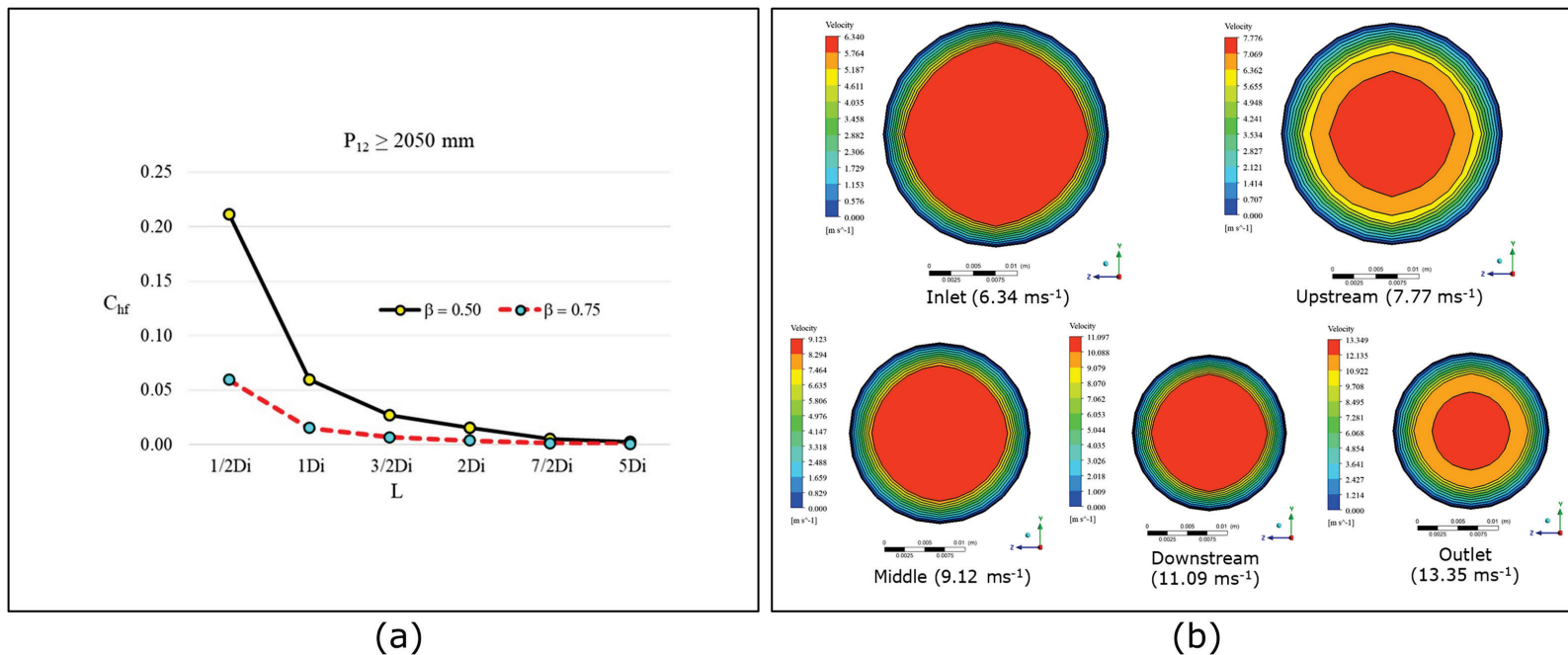


Figure 2. Characteristic and contraction flow (P_{12}); (a) various contractions, (b) velocity contours with $\beta=0.75$, $L=0.5D_i$ and $U_{\max}$.

CFD simulations provide a detailed depiction of the hydrodynamic behaviour of turbulent flow through contraction pipes. "Figure 2b" shows the velocity contours and streamlines for a representative contraction ratio. As the flow enters the contraction section, a significant acceleration occurs due to the reduction in cross-sectional area, in accordance with the continuity requirement $Q_i = Q_{i+1}$.

The analysis results indicate that variations in the pressure loss coefficient are influenced by the contraction ratio and the length of the contraction pipe. An increase in the contraction ratio and contraction length leads to a reduction in the pressure loss coefficient. Conversely, changes in the water pressure head have no significant effect on the pressure loss coefficient.

The velocity magnitude increases sharply within the contraction zone, reaching its maximum at the downstream inlet. The strongest velocity gradients near the pipe wall accompany this acceleration (Schatzman & Thomas, 2017), particularly at higher contraction angles (α). Flow separation occurs immediately downstream of the contraction edge, forming a recirculation zone whose size increases as the contraction ratio β decreases. This phenomenon directly contributes to additional energy dissipation and local pressure drop, as reported in previous studies (Song et al., 2023).

The flow reattachment length downstream of the contraction is highly dependent on β and the Reynolds number. For mild contractions, the flow reattaches quickly and exhibits a relatively uniform velocity profile; in contrast, sudden contractions produce extended separation regions, resulting in persistent non-uniformity in the downstream flow field.

3.2. Pressure distribution and energy loss behavior

“Figure 3” illustrates the distribution of static pressure along the axial direction of the pipe system. A gradual pressure drop is observed along the upstream straight pipe due to wall friction, followed by a sharp pressure drop at the contraction section. This sudden pressure reduction highlights that the contraction-induced loss is more dominant than the frictional loss along the flow path.

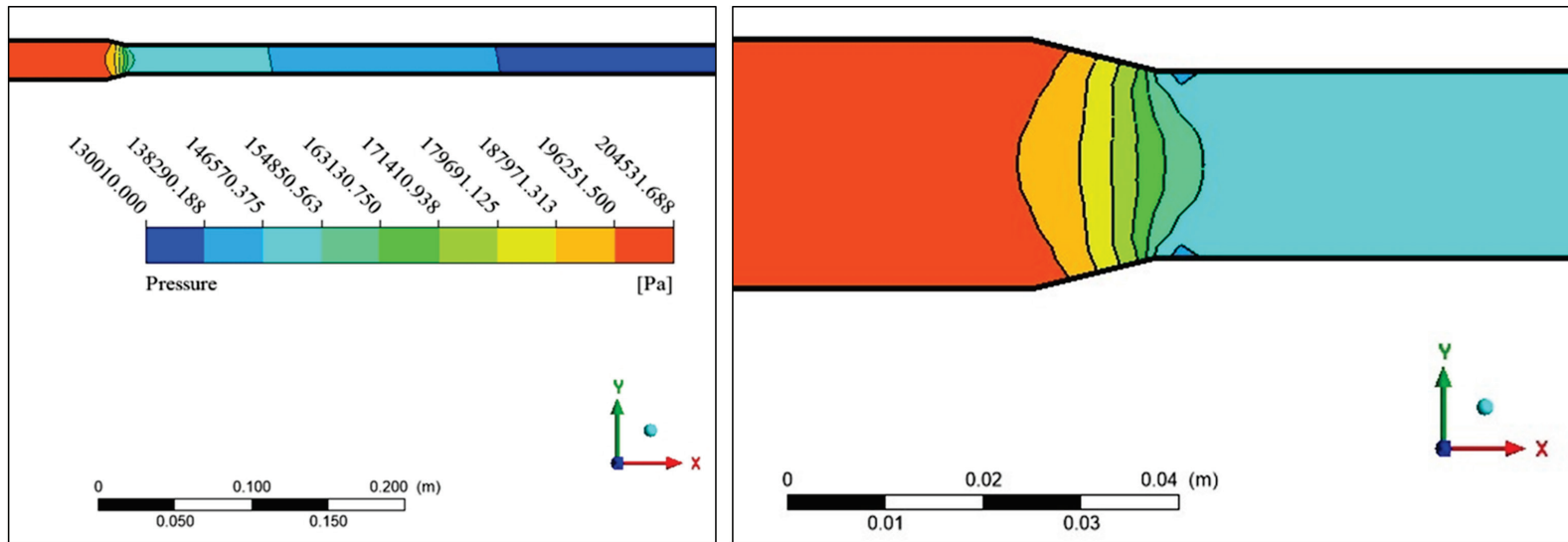


Figure 3. Pressure distribution (P_{12}) with $\beta=0.75$, $L=0.5D_i$ (Pressure: 130 – 204.5 kPa).

The magnitude of the pressure drop at the contraction section increases as the contraction ratio β decreases. This behavior is caused by enhanced flow separation and greater turbulence production resulting from the abrupt change in flow direction. These observations are consistent with classical pipe flow theory and experimental findings reported in previous studies.

The computed local pressure drop (Δh_f) is used to evaluate the contraction pressure loss coefficient (C_{hf}). The results indicate that C_{hf} increases nonlinearly with increasing contraction angle (α), with geometric effects playing a dominant role in energy dissipation within contraction flow systems.

3.3. Contribution of shear stress loss and directional change loss

The total pressure loss coefficient (C_{hf}) is decomposed into shear stress-related loss (δ_f) and directional change loss (δ_p), as defined in the governing equations. "Figure 1" compares the relative contributions of these two components for different contraction geometries.

The results demonstrate that δ_p , which is associated with changes in flow direction, accounts for the majority of energy loss in sudden contraction configurations. This contribution becomes increasingly significant as the contraction angle α increases, confirming the theoretical relationship $\delta_p = 2(1 - \cos \alpha)$.

In contrast, the frictional component δ_f contributes a smaller but non-negligible portion of the total loss, particularly for longer contraction lengths. This suggests that while sudden geometric changes significantly influence energy dissipation, the contraction length and wall interaction still impact the overall system efficiency. These findings explain the limitations of simple empirical models that neglect friction effects.

Table 2. The head loss coefficient with the Moody friction factor (f)

Head (mm)	Pi [kPa]	Moody (f)	Shear stress (δf) 10-4				Directional change (δp) 10-2			
			1/2Di	1Di	2Di	5Di	1/2Di	1Di	2Di	5Di
Contraction Pipe Dimension with Di = 1 inch [Di+1 = 3/4Di], $\beta = 0.75$										
675	64.94	0.0169	0.155	0.303	0.602	1.503	5.971	1.544	0.389	0.062
1050	101.02	0.0164	0.150	0.293	0.583	1.456	5.971	1.544	0.389	0.062
1425	137.10	0.0160	0.146	0.285	0.568	1.417	5.971	1.544	0.389	0.062
1800	173.18	0.0156	0.142	0.279	0.554	1.382	5.971	1.544	0.389	0.062
2050	197.23	0.0153	0.140	0.274	0.545	1.361	5.971	1.544	0.389	0.062
Contraction Pipe Dimension with Di = 1 inch [Di+1 = 1/2Di], $\beta = 0.50$										
675	64.94	0.01586	0.135	0.249	0.486	1.208	21.115	5.971	1.533	0.250
1050	101.02	0.01506	0.128	0.236	0.462	1.147	21.115	5.971	1.533	0.250
1425	137.10	0.01440	0.122	0.226	0.441	1.096	21.115	5.971	1.533	0.250
1800	173.18	0.01382	0.117	0.217	0.424	1.052	21.115	5.971	1.533	0.250
2050	197.23	0.01346	0.114	0.211	0.413	1.025	21.115	5.971	1.533	0.250

“Table 2” compares the relative contributions of pressure loss due to friction (δ_f) and pressure loss due to changes in flow direction (δ_p) to the total pressure loss coefficient (C_{hf}). The results show that pressure loss from changes in flow direction dominates the total energy dissipation for all considered contraction ratios.

As the contraction angle α increases, δ_p rises sharply, consistent with the theoretical relationship $\delta_p = 2(1 - \cos \alpha)$. This confirms that abrupt changes in flow direction are the primary mechanism causing contraction pressure losses.

Although smaller in magnitude, the friction component increases with contraction length and contributes non-negligibly to the total loss, particularly for longer contraction sections. This finding highlights the importance of including friction effects when modelling contraction flows, especially in cases where the contraction length is not negligible.

3.4. Turbulence characteristics and energy dissipation

The turbulence kinetic energy (k) contours shown in “Figure 4” reveal intense turbulence generation within the contraction region and the downstream area. High values of k are concentrated near the contraction edge and along the shear layers formed by flow separation.

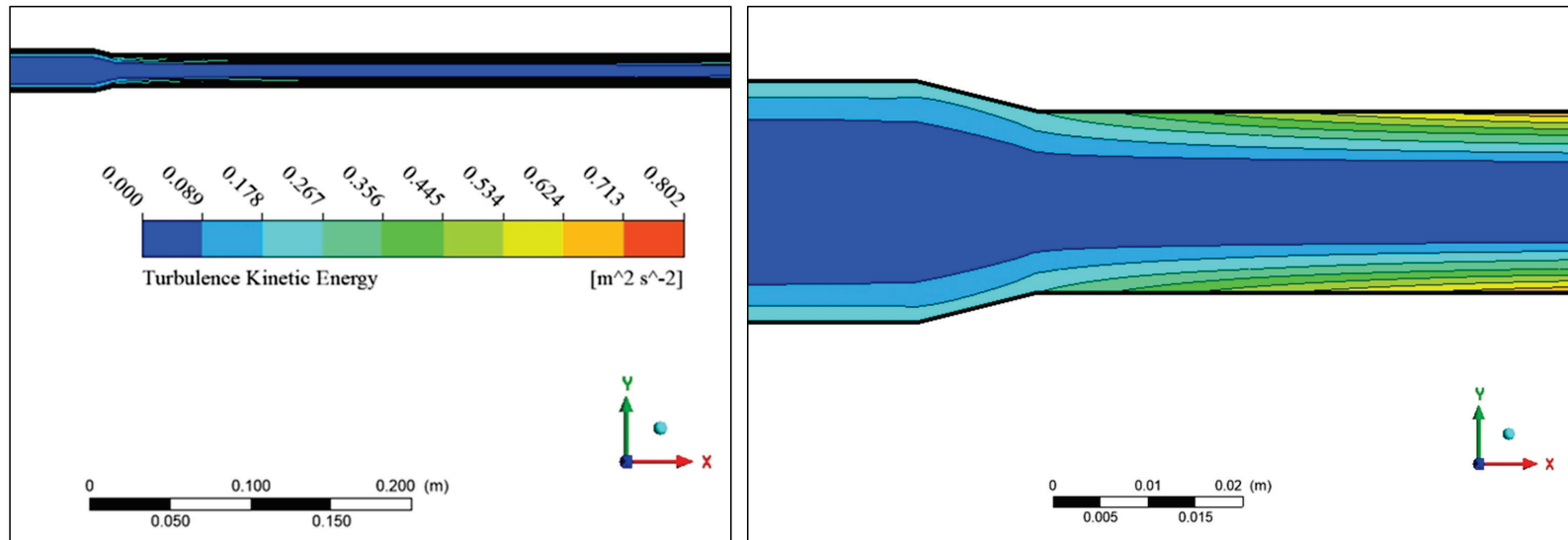


Figure 4. Turbulence characteristics (P_{12}) with $\beta=0.75$, $L=0.5D_i$, (turbulence kinetic energy 0 – 0.8 $\text{m}^2 \text{s}^{-2}$).

As the flow progresses downstream, turbulence gradually decays, and the velocity profile redevelops towards a fully developed turbulent pipe flow. The standard $k-\epsilon$ turbulence model successfully captures these trends, demonstrating its suitability for modelling contraction-induced turbulence in internal pipe flows.

The enhanced turbulence production directly correlates with increased energy dissipation, reinforcing the strong relationship between contraction geometry, turbulence intensity, and pressure loss coefficient.

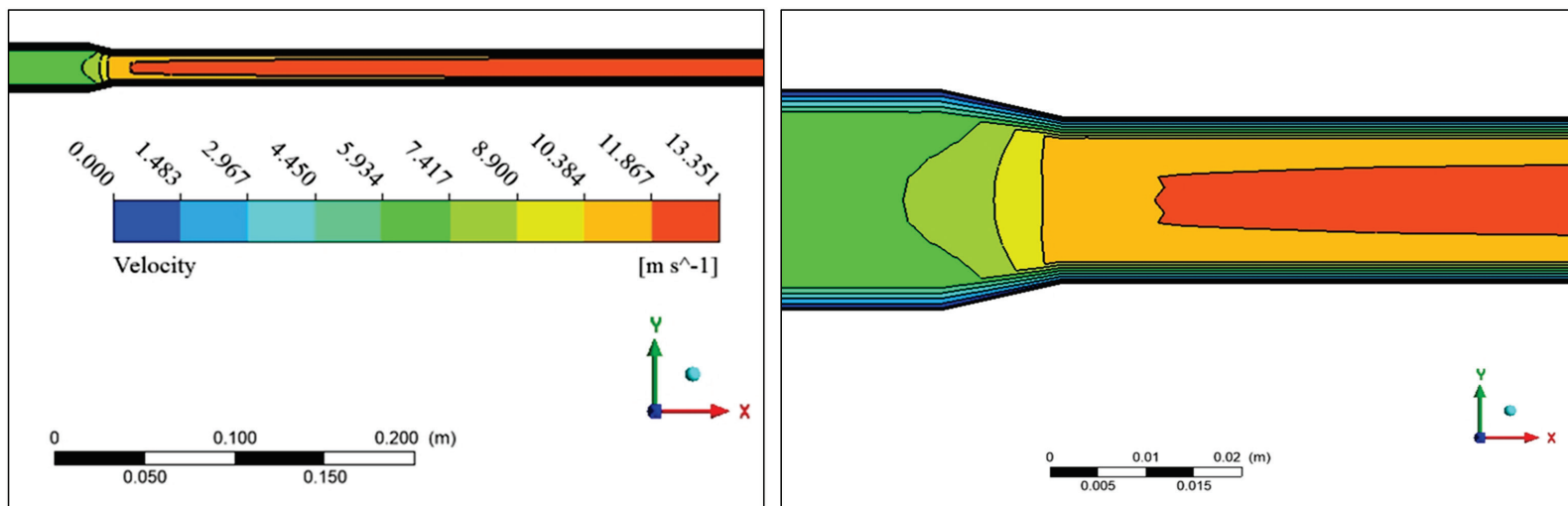


Figure 5. Velocity contour (P_{12}) with $\beta=0.75$, $L=0.5D_i$, inlet (5.93 ms^{-1}) and outlet (13.35 ms^{-1}).

“Figure 5” shows the velocity contours and streamlines for a representative contraction ratio. As the flow enters the contraction section, a pronounced water velocity occurs due to the reduction in cross-sectional area, in accordance with the continuity condition $Q_i = Q_{i+1}$. The velocity magnitude increases sharply within the contraction zone and

reaches its maximum downstream of the contraction as it enters the downstream pipe.

Flow separation is immediately observed downstream of the contraction edge, forming a recirculation region characterized by low-velocity vortices. The size and intensity of this recirculation zone increase as the contraction ratio β decreases, indicating stronger adverse pressure gradients for sharper contractions. This behavior is consistent with classical observations of sudden contraction flows and confirms that geometric discontinuities significantly disturb the flow structure (González et al., 2024).

The streamlines show gradual flow reattachment downstream, indicating that the selected upstream and downstream pipe lengths are sufficient to achieve fully developed flow beyond the contraction zone. This flow pattern directly contributes to increased energy dissipation and local pressure loss.

The axial distribution of static pressure along the pipe centerline is shown in "Figure 3". A gradual pressure drop is observed in the upstream straight pipe due to wall friction, followed by a sudden and substantial decrease in pressure across the contraction section.

This abrupt pressure drop confirms that local losses associated with the contraction are more dominant than pressure losses due to friction. The magnitude of the pressure drop increases as the contraction ratio decreases, reflecting enhanced turbulence production and flow separation.

Downstream of the contraction, the pressure partially recovers as the flow redevelops, although the original pressure level is not fully restored. This irreversible pressure drop reflects energy dissipation

caused by turbulence and viscous effects, thereby justifying the use of the local pressure loss coefficient (C_{hf}) to characterize contraction losses.

3.5. Geometric parameters and flow conditions

“Table 3” summarizes the geometric configurations and flow conditions used in the empirical simulations, including upstream and downstream diameters, contraction length, contraction ratio β , and Reynolds number. All cases fall within the fully turbulent flow regime, ensuring that the results are representative of practical water distribution systems.

Table 3. Geometric parameters and flow conditions

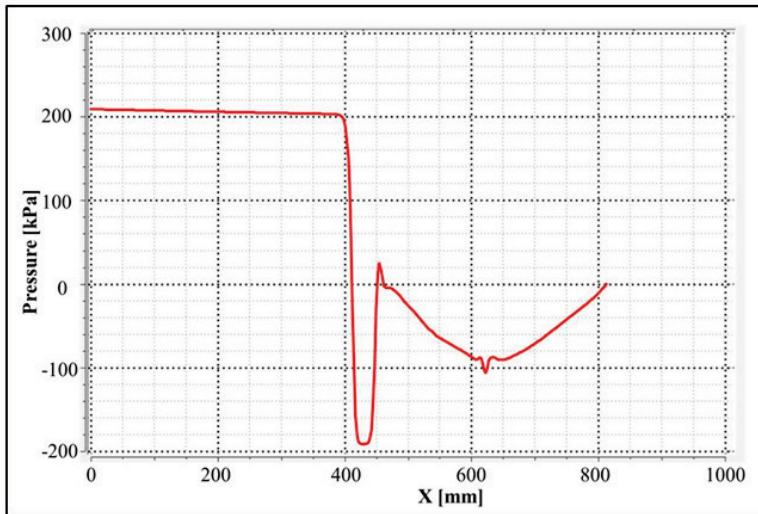
Head (mm)	Pi [kPa]	β	U [ms ⁻¹]		Re [x 10 ⁴]			Moody (f)	C _{hf} [x 10 ⁻²]			
			U _i	U _{i+1}	D _i	D _{i+1}	Flows		1/2D _i	1D _i	2D _i	5D _i
Contraction Pipe Dimension with D _{i+1} = 3/4D _i , β = 0.75												
675	64.94	0.75	3.64	6.47	9.24	12.33	TF	0.01694	5.973	1.547	0.396	0.078
1050	101.02	0.75	4.54	8.07	11.53	15.37	TF	0.01641	5.973	1.547	0.395	0.077
1425	137.10	0.75	5.29	9.40	13.44	17.91	TF	0.01597	5.973	1.547	0.395	0.077
1800	173.18	0.75	5.94	10.56	15.10	20.13	TF	0.01558	5.973	1.547	0.395	0.076
2050	197.23	0.75	6.34	11.27	16.11	21.48	TF	0.01534	5.973	1.547	0.395	0.076
Contraction Pipe Dimension with D _{i+1} = 1/2D _i , β = 0.50												
675	64.94	0.50	3.64	14.56	9.24	18.49	TF	0.01586	21.116	5.974	1.549	0.262
1050	101.02	0.50	4.54	18.16	11.53	23.06	TF	0.01506	21.116	5.974	1.549	0.261
1425	137.10	0.50	5.29	21.15	13.44	26.86	TF	0.01440	21.116	5.974	1.549	0.260
1800	173.18	0.50	5.94	23.77	15.10	30.19	TF	0.01382	21.116	5.974	1.549	0.260
2050	197.23	0.50	6.34	25.37	16.11	32.22	TF	0.01346	21.116	5.974	1.549	0.260

Notes: TF = Turbulent flow.

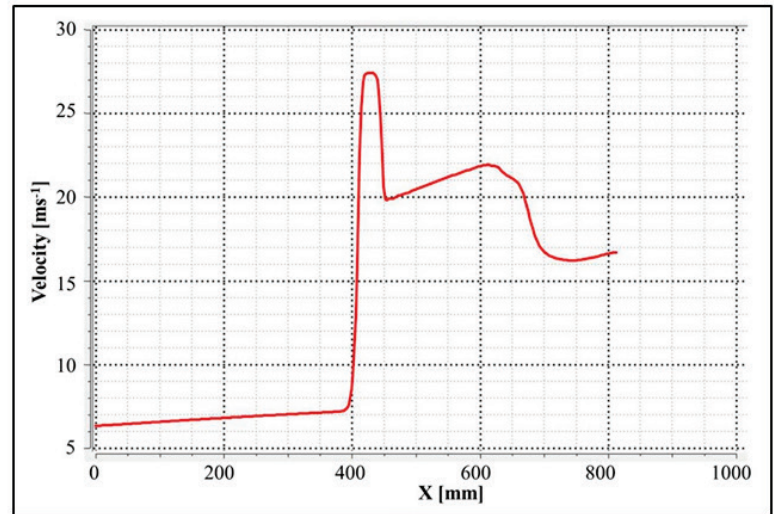
The systematic variation of β allows for a direct assessment of the geometric effects on flow behavior and pressure loss, while maintaining consistent conditions. This controlled parametric approach strengthens the reliability of the comparative analysis presented in the subsequent figures. “Figure 4” presents the contours of turbulent kinetic energy (k) in the contraction pipe system. The highest turbulence intensity is localized near the contraction edge and within the shear layers formed as a result of flow separation.

The peak values of k occur immediately downstream of the contraction, where strong velocity gradients and recirculation promote turbulence production. As the flow progresses downstream, turbulence gradually decays, and the velocity profile redevelops towards a fully developed turbulent pipe flow.

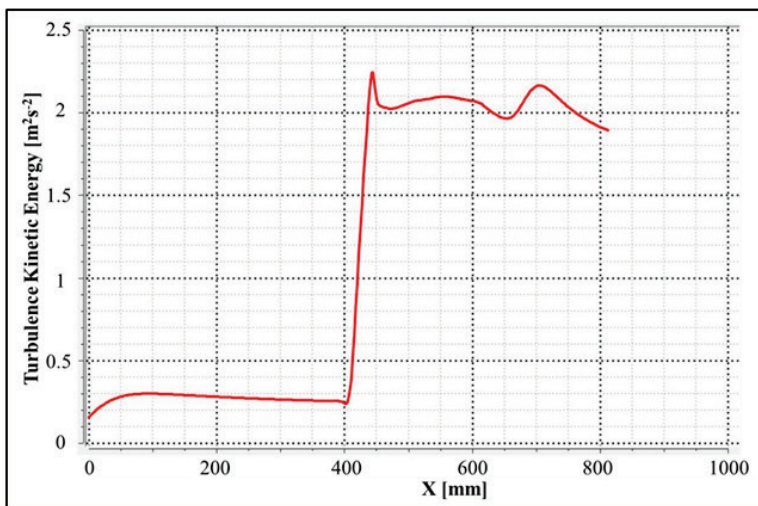
The ability of the standard k - ϵ turbulence model to capture these trends demonstrates its suitability for simulating contraction-induced turbulence in internal flows. The observed turbulence behaviour explains the irreversible pressure drops identified in “Figure 6” and provides a physical basis for the computed pressure loss coefficients.



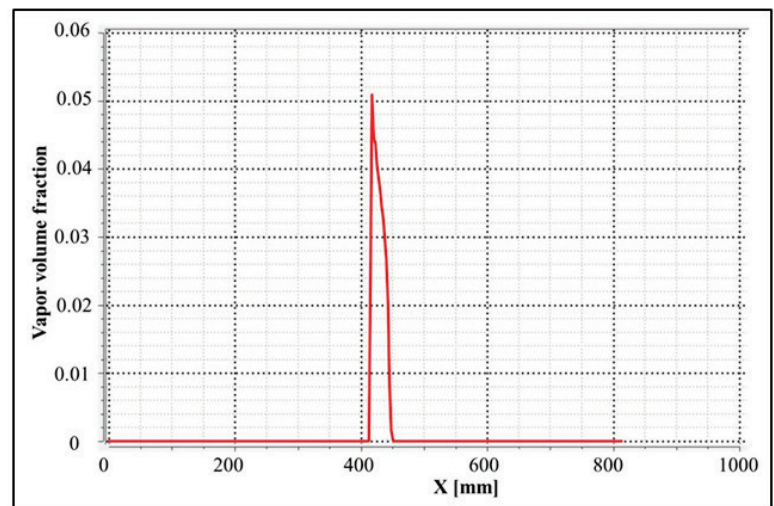
(a)



(b)



(c)



(d)

Figure 6. Variation of (a) pressure, (b) velocity, (c) turbulence, and (d) cavitation with head 2050 mm, $\beta=0.50$ and $L=0.5D_i$.

"Figure 6" and "Table 3" illustrate the variation in pressure loss as a function of the contraction ratio β . A clear inverse relationship is observed: as β decreases, C_{hf} increases significantly.

This trend reflects the combined effects of increased velocity, stronger flow separation, and higher turbulence production associated with sharper contractions. For small β values, the rapid rise in C_{hf} indicates a decline in hydraulic efficiency, highlighting the sensitivity of pressure losses to geometric design.

Importantly, the results demonstrate that contraction geometry optimization can substantially reduce pressure losses in pipe systems, providing practical guidance for designing more efficient and sustainable water distribution networks.

Table 4. Relative Pressure Drop Ratio (CRPD)

Head (mm)	Empirical Analysis (CRPD)					CFD Results (CRPD)				
	L=1/2Di	L=2Di	L=5Di	Flows		L=1/2Di	L=2Di	L=5Di	Flows	
Contraction Pipe Dimension with Di+1 = 3/4Di, β = 0.75										
675	0.341	0.323	0.322	TF	NC	0.221	0.091	0.230	TF	NC
1050	0.338	0.323	0.322	TF	NC	0.222	0.228	0.227	TF	NC
1425	0.341	0.323	0.322	TF	NC	0.223	0.185	0.229	TF	NC
1800	0.341	0.323	0.322	TF	NC	0.223	0.217	0.029	TF	NC
2050	0.341	0.323	0.322	TF	NC	0.223	0.185	0.031	TF	NC
Contraction Pipe Dimension with Di+1 = 1/2Di, β = 0.50										
675	1.972	1.653	1.632	TF	Cav.	1.316	1.284	1.510	TF	Cav.
1050	1.972	1.673	1.632	TF	Cav.	1.276	1.373	1.509	TF	Cav.
1425	1.972	1.653	1.632	TF	Cav.	1.326	1.389	1.532	TF	Cav.
1800	1.972	1.653	1.632	TF	Cav.	1.269	1.404	1.534	TF	Cav.
2050	1.972	1.653	1.632	TF	Cav.	1.269	1.401	1.524	TF	Cav.

Notes:

US = Upstream

DS = Downstream

TF = Turbulent flow

NC = no Cavitation

Cav.=Cavitation occurs

“Table 4” presents a comparison of the relative pressure drop ratios derived from the empirical analytical model and the CFD simulations. The discrepancy between the two approaches remains within $\pm 30\%$ for both cavitating and non-cavitating conditions, with the CFD results consistently lower than those obtained from the empirical analysis. This suggests that the empirical method yields a more conservative safety margin compared with the CFD simulation model.

Table 5. Pressure and cavitation status

Head (mm)	U [ms-1]		Pi [kPa]	L=1/2Di			L=2Di			L=5Di		
	Ui	Ui+1		Pi+1	σ		Pi+1	σ		Pi+1	σ	
Contraction Pipe Dimension with Di+1 = 3/4Di, β = 0.75												
675	3.64	6.47	64.94	42.81	3.00	NC	43.97	3.00	NC	44.04	3.00	NC
1050	4.54	8.07	101.02	66.59	3.04	NC	68.40	3.04	NC	68.51	3.04	NC
1425	5.29	9.40	137.10	90.37	3.06	NC	92.83	3.06	NC	92.97	3.06	NC
1800	5.94	10.56	173.18	114.15	3.07	NC	117.26	3.07	NC	117.44	3.07	NC
2050	6.34	11.27	197.23	130.01	3.07	NC	133.55	3.07	NC	133.75	3.07	NC
Contraction Pipe Dimension with Di+1 = 1/2Di, β = 0.50												
675	3.64	14.56	64.94	-63.12	0.59	Cav.	-42.43	0.59	Cav.	-41.07	0.59	Cav.
1050	4.54	18.16	101.02	-98.19	0.60	Cav.	-66.01	0.60	Cav.	-63.89	0.60	Cav.
1425	5.29	21.15	137.10	-133.26	0.60	Cav.	-89.58	0.60	Cav.	-86.70	0.60	Cav.
1800	5.94	23.77	173.18	-168.32	0.61	Cav.	-113.15	0.61	Cav.	-109.52	0.61	Cav.
2050	6.34	25.37	197.23	-191.70	0.61	Cav.	-128.87	0.61	Cav.	-124.73	0.61	Cav.

Notes:

Water conditions at a temperature of 20 °C

$\rho = 998 \text{ kgm}^{-3}$

$P_v = 2.339 \text{ kPa}$

σ = Cavitation number

NC = No Cavitation

Cav. = Cavitation occurs P_{i+1} [kPa].

The flow behavior described in “Table 5” indicates that variations in head and contraction arm length are not the primary causes of bubble flow or cavitation (Alavi et al., 2025). Instead, the contraction ratio (β) plays a dominant role, particularly for steep contraction ratios, which induce a significant head reduction and high flow velocities, leading to increased flow turbulence.

4. Discussion

The CFD-predicted values of C_{hf} show good agreement with the theoretical formulation proposed in this study. Minor deviations are attributed to three-dimensional flow effects and turbulence, which cannot be fully predicted.

This phenomenon is caused by extreme contraction, which induces cavitation in the downstream section. The occurrence of cavitation becomes the primary cause of the extreme pressure change from positive pressure to negative (suction) pressure. The analysis results indicate that such cavitation events can be avoided by selecting a smoother contraction geometry, resulting in a relative pressure drop ratio of less than one ($CRPD < 1$), as presented in “Table 3”. Furthermore, the results obtained from the empirical analysis and CFD simulations demonstrate a consistent trend.

Larger deviations occur at the smallest contraction ratios, where complex three-dimensional flow structures and strong turbulence effects become more dominant. These results indicate that, while the analytical model provides a reliable first-order prediction, CFD analysis remains necessary to capture the detailed flow physics in extreme contraction



scenarios accurately. Although the accuracy of the results is highly influenced by mesh modelling, the smaller the mesh size and the greater the number of constituent mesh elements, the closer the constructed geometry approaches the ideal shape, thereby producing more accurate results.

The analysis results confirm that contraction geometry plays a dominant role in determining pressure drop in pipe systems. Sudden contractions with small β values lead to strong flow separation, high turbulence levels, and large pressure drop coefficients, thereby increasing pumping energy requirements. The results obtained from the empirical and CFD methods show noticeable differences. The pressure loss ratio predicted by the empirical method tends to be higher than that obtained from the CFD method, as presented in "Table 4". This discrepancy may be attributed to several factors, including the more rigorous numerical iteration process and the detailed mesh modelling employed in CFD simulations, which produce more precise results.

From a design perspective, smoother transitions or larger contraction ratios should be adopted where possible to minimize pressure losses due to directional changes. The combined empirical-CFD analytical approach developed in this study provides a practical and reliable framework for predicting contraction-induced pressure drops and optimizing pipe system design.

Referring to the results obtained from both the empirical and CFD methods, the moderate and optimal contraction pipe geometry model is characterized by a value of $\beta \geq 0.75$ and a contraction pipe length (L) $\geq$ the upstream pipe diameter (D_i). With this geometry, the pressure loss

coefficient tends to be low and does not generate significant cavitation or dominant turbulence.

The results of this study indicate that pressure drops caused by contractions represent a significant source of inefficiency in pipe systems. Sudden contractions generate excessive turbulence and irreversible pressure losses, thereby increasing pumping energy requirements and operational costs.

From a sustainability perspective, optimizing contraction geometry offers a practical strategy for reducing energy consumption and improving the long-term performance of water infrastructure. The methodology developed in this study provides a reliable framework for evaluating and minimizing energy losses, supporting the design of more efficient and sustainable water distribution networks.

5. Conclusions

This study focuses on the efficiency of the pressure drop coefficient in contraction flow pipe systems using a combined empirical and CFD-based approach. The main conclusions can be outlined as follows:

1. The conceptual framework and hypotheses were developed based on an examination of contraction phenomena in pipe networks, particularly pressure losses caused by cavitation and turbulence in contraction pipes. The main parameters include flow velocity, Reynolds number, pipe wall roughness, upstream–downstream pipe diameters, and contraction length.

2. Sudden pipe contractions significantly alter flow behavior, causing strong acceleration, flow separation, and increased turbulence, which lead to substantial local pressure losses.

3. Directional change losses dominate over friction losses for all contraction geometries, particularly for sharp contractions with small contraction ratios.

4. The CFD simulation results demonstrate relevant trends, although differences in values still occur due to the influence of mesh element density in the modelling process. The denser and more numerous the mesh elements used, the more accurate the simulation results become. Optimization of the contraction pipe geometry also affects energy efficiency and the overall cost efficiency of the project.

5. The pressure loss coefficient increases rapidly as the contraction ratio decreases, indicating reduced hydraulic efficiency and increased pumping energy demand.

6. CFD predictions show good agreement with the proposed analytical formulation, validating its applicability for practical hydraulic design.

7. Optimizing contraction geometry represents an effective strategy for improving energy efficiency and supporting sustainable water infrastructure development.

8. Under conditions of increasingly lower contraction ratios and steep gradients, cavitation occurs due to an extreme drop in pressure.

9. Referring to the results obtained from both the empirical and CFD methods, the moderate and optimal contraction pipe geometry model is characterized by a value of $\beta \geq 0.75$ and a contraction pipe length (L)

$\geq$ the upstream pipe diameter (D_i). With this geometry, the resulting pressure loss coefficient tends to be low and does not generate significant cavitation or dominant turbulence.

These findings provide valuable guidance for engineers and decision-makers to reduce energy consumption in water distribution systems. Future research could extend this analysis to include gradual contractions, surface roughness effects, and transient flow conditions, thereby further enhancing the sustainability assessment of pipe networks.

6. Acknowledgments

The authors express their sincere appreciation for the support of all resources, including academic and computational facilities, provided by the University of Muhammadiyah Malang.

7. Funding

Funding support for this research was provided by the University of Muhammadiyah Malang.

8. Statement of conflict of interest

The authors declare that there is no conflict of interest regarding the publication of this article.

9. Statement on the use of IA

All scientific content, data analysis, interpretation of results, and conclusions were developed and verified by the authors, who take full responsibility for the content of this article.

10. References

Abduh, M., Iqbal, K., & Sulianto. (2025). Prediction of Flow Resistance Coefficient Correction in Modified Pipe Bends. *CFD Letters*, 17(9), 178–193.

<https://doi.org/https://doi.org/10.37934/cfdl.17.9.178193>

Abduh, M., Orfa, L. E., Sulianto, S., & Iqbal, K. (2024). Pressure Drop Analysis of Turbine Housing Model with Circular Sliced Pipe for Micro Hydropower Generation. *Aceh International Journal of Science and Technology*, 13(2), 92–102.

<https://doi.org/10.13170/aijst.13.2.37998>

Alavi, A., Ghasemnezhad, M., Sangtarash, A., & Roohi, E. (2025). Passive cavitation mitigation on hydrofoils via porous media: A comparative study of LES and RANS models. *Ocean Engineering*, 341, 122756.

<https://doi.org/https://doi.org/10.1016/j.oceaneng.2025.122756>

Altowayti, W. A. H., Othman, N., Tajarudin, H. A., Al-Dhaqm, A., Asharuddin, S. M., Al-Gheethi, A., Alshalif, A. F., Salem, A. A., Din, M. F. M., Fitriani, N., & AL-Towayti, F. A. H. (2021). Evaluating the Pressure and Loss Behavior in Water Pipes Using Smart Mathematical Modelling. *Water*, 13(24), 3500.

<https://doi.org/10.3390/w13243500>

- ANSYS. (2025). *ANSYS Fluent Theory Guide 2025* (R2 ed., Vol. 1). 2025 AANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.
https://ansyshelp.ansys.com/public/account/secured?returnurl=/Views/Secured/corp/v242/en/flu_th/flu_th.html
- Avci, M., & Papavassiliou, D. V. (2026). Numerical Investigation of Inlet Velocity Profile Effects on Developing Laminar Flow in a Circular Pipe. *Applied Sciences*, 16(3), 1215.
<https://doi.org/10.3390/app16031215>
- Bakri, B., Pallu, S., Sulhairi, Pongmanda, S., Arai, Y., & Ihsan, M. (2022). Effects of Fluctuation in Discharge and Head on Flow Parameters and Head Losses in Pipeline Networks. *IOP Conference Series: Earth and Environmental Science*, 1117(1), 012069.
<https://doi.org/10.1088/1755-1315/1117/1/012069>
- CANLI, E., ATES, A., & BİLİR, Ş. (2021). Developing Turbulent Flow in Pipes and Analysis of Entrance Region. *Academic Platform Journal of Engineering and Science*, 9(2), 332–353.
<https://doi.org/10.21541/apjes.818717>
- Cao, L., Zhou, S., Wang, H., Liu, C., Wei, R., Wang, W., Hu, X., & Zhang, N. (2026). The dominant role of relative roughness in microscale flow transition: Redefining the critical Reynolds number for microflow measurement. *Flow Measurement and Instrumentation*, 108, 103151.
<https://doi.org/https://doi.org/10.1016/j.flowmeasinst.2025.103151>

- Dadar, S., Đurin, B., Alamatian, E., & Plantak, L. (2021). Impact of the Pumping Regime on Electricity Cost Savings in Urban Water Supply System. *Water*, 13(9), 1141. <https://doi.org/10.3390/w13091141>
- Durmuş, S. (2024). Optimization of contraction cone length in an open-circuit wind tunnel. *Balıkesir Üniversitesi Fen Bilimleri Enstitüsü Dergisi*, 26(2), 431–440. <https://doi.org/10.25092/baunfbed.1339334>
- Ennouri, M., Kanfoudi, H., Bel Hadj Taher, A., & Zgolli, R. (2019). Numerical Flow Simulation and Cavitation Prediction in a Centrifugal Pump using an SST-SAS Turbulence Model. *Journal of Applied Fluid Mechanics*, 12(1), 25–39. <https://doi.org/10.29252/jafm.75.253.28771>
- Erhan, Ö. (2024). Increasing energy efficiency and flow coefficient by determining fluid behaviours with computer aided design and CFD analysis in isolation valves. *Advances in Mechanical Engineering*, 16(10). <https://doi.org/10.1177/16878132241260587>
- Baghdadi, F., Hrairi, M., & Asraar, W. (2024). A Numerical Comparison of 2D and 3D CFD Modelling for Contraction and Expansion Geometries with an Emphasis on Solid Particles Erosion. *CFD Letters*, 16(6), 157–168. <https://doi.org/10.37934/cfdl.16.6.157168>
- Farooq, U., Li, S., & Yang, J. (2024). Numerical Analysis of Flow Characteristics and Energy Dissipation on Flat and Pooled Stepped Spillways. *Water*, 16(18), 2600. <https://doi.org/10.3390/w16182600>

- González, J., Meana-Fernández, A., Pérez, I. V., & Oro, J. M. F. (2024). Minor Loss Coefficient for Abrupt Section Changes in a Cylindrical Pipe Using a Numerical Approach. *Fluids*, 9(7), 152. <https://doi.org/10.3390/fluids9070152>
- He, J., Li, C., & Guo, Y. (2025). Modeling Transient Vaporous Cavitating Flow in Pipelines by a Two-Phase Homogeneous Flow Model. *Processes*, 13(11), 3510. <https://doi.org/10.3390/pr13113510>
- Ho, M. L. G., Tan, L.-L., Hung, Y. M., & Oon, C. S. (2023). Thermohydraulic and Irreversibility Analyses of Swirl Flows Utilizing Distorted Radial Fins: Entropy Generation and Entransy Dissipation Evaluation. *International Journal of Energy Research*, 2023, 1–25. <https://doi.org/10.1155/2023/5557828>
- Kadivar, M., Tormey, D., & McGranaghan, G. (2023). A comparison of RANS models used for CFD prediction of turbulent flow and heat transfer in rough and smooth channels. *International Journal of Thermofluids*, 20, 100399. <https://doi.org/10.1016/j.ijft.2023.100399>
- Lee, H., Lee, C., Kim, M.-S., & Seok, W. (2025). Numerical Investigation of Cavitation Models Combined with RANS and PANS Turbulence Models for Cavitating Flow Around a Hemispherical Head-Form Body. *Journal of Marine Science and Engineering*, 13(4), 821. <https://doi.org/10.3390/jmse13040821>
- Li, Y., Gao, J., Chen, J., Li, K., Qi, S., & Lin, S. (2026). Characterization of leak flow from pipes under the effect of cavitation in water distribution systems. *Water Research*, 291, 125129. <https://doi.org/10.1016/j.watres.2025.125129>

- Liu, C., Ao, D., Li, X., & Li, Y. (2025). Numerical Simulation of High-pressure Cavitation Characteristics of Thick Orifice Plate. *Journal of Pipeline Science and Engineering*, 100365. <https://doi.org/10.1016/j.jpse.2025.100365>
- Meseret, M., Wami, D., & Nallamotheu, R. B. (2024). Experimental study on the parameters affecting the performance of spiral tube pump. *Scientific Reports*, 14(1), 22425. <https://doi.org/10.1038/s41598-024-73234-w>
- Mugundhan, V., & Thoroddsen, S. T. (2023). Circulation in turbulent flow through a contraction. *Journal of Turbulence*, 24(11–12), 577–612. <https://doi.org/10.1080/14685248.2023.2284187>
- Nohmi, M., Kagawa, S., Tsuneda, T., Tsuru, W., & Yokota, K. (2019). Numerical Analysis of Contraction Geometry Effects on Cavitation Choking in a Piping System. In *Volume 3B: Fluid Applications and Systems*. American Society of Mechanical Engineers. <https://doi.org/10.1115/AJKFluids2019-5359>
- Ren, H., Zhang, T., Tian, Q., Yang, H., Tian, Y., Guo, L., & Ren, K. (2025). A Fault Diagnosis Method for Pump Station Units Based on CWT-MHA-CNN Model for Sustainable Operation of Inter-Basin Water Transfer Projects. *Sustainability*, 17(24), 11383. <https://doi.org/10.3390/su172411383>
- Schatzman, D. M., & Thomas, F. O. (2017). An experimental investigation of an unsteady adverse pressure gradient turbulent boundary layer: embedded shear layer scaling. *Journal of Fluid Mechanics*, 815, 592–642. <https://doi.org/10.1017/jfm.2017.65>

- Shakouri, B., Ismail, I., & Safari, M. J. S. (2023). Energy loss and contraction coefficients-based vertical sluice gate's discharge coefficient under submerged flow using symbolic regression. *Environmental Science and Pollution Research*, 30(31), 76853–76866. <https://doi.org/10.1007/s11356-023-27388-1>
- Shrestha, U., & Choi, Y.-D. (2020). A CFD-Based Shape Design Optimization Process of Fixed Flow Passages in a Francis Hydro Turbine. *Processes*, 8(11), 1392. <https://doi.org/10.3390/pr8111392>
- Yusof, S. N. A., Asako, Y., Sidik, N. A. C., Mohamed, S. B., & Japar, W. Mohd. A. A. (2020). A Short Review on RANS Turbulence Models. *CFD Letters*, 12(11), 83–96. <https://doi.org/10.37934/cfdl.12.11.8396>
- Song, X., Ma, J., Li, Y., Sun, X., & Zhao, Y. (2023). Analysis of the flow phenomenon of fluid undergoing a sudden contraction to an annular gap. *Physics of Fluids*, 35(9). <https://doi.org/10.1063/5.0169034>
- Souza, T. R. C. de, Andrade, J. T. M. de, Serrano, R. O. P., Vidigal, T. H. D. A., Viana, E. M. de F., Bastos, A. S., & Martinez, C. B. (2023). Energy Efficiency Analysis of Pumping Systems Impacted by the Golden Mussel: A Case Study in the Brazilian Amazon. *Energies*, 16(4), 1858. <https://doi.org/10.3390/en16041858>
- Tang, X., Duan, X., Gao, H., Li, X., & Shi, X. (2020). CFD Investigations of Transient Cavitation Flows in Pipeline Based on Weakly-Compressible Model. *Water*, 12(2), 448. <https://doi.org/10.3390/w12020448>

- Wang, H., Krüger, T., & Varnik, F. (2021). Geometry and Flow Properties Affect the Phase Shift between Pressure and Shear Stress Waves in Blood Vessels. *Fluids*, 6(11), 378. <https://doi.org/10.3390/fluids6110378>
- Wu, Y., Li, G., Guo, P., Zhang, H., Ye, X., Su, Y., & An, X. (2024). CFD Investigation on Resistance Balance of Complex Pipe Networks and Optimization Based on Artificial Neural Network. *Industrial & Engineering Chemistry Research*, 63(50), 22134–22149. <https://doi.org/10.1021/acs.iecr.4c03248>
- Xu, X., Fang, L., Li, A., Wang, Z., & Li, S. (2021). Numerical Analysis of the energy loss mechanism in cavitation flow of a control valve. *International Journal of Heat and Mass Transfer*, 174, 121331. <https://doi.org/10.1016/j.ijheatmasstransfer.2021.121331>