**DOI: 10.31319/2519-8106.1(52)2025.323814**
**UDK 538.911**

**Shcherbyna Iryna**, Candidate of Physical and Mathematical Sciences, Associate Professor Department of Higher Mathematics, Physics and General Engineering Disciplines
**Щербина І.В.**, кандидат фізико-математичних наук, доцент кафедри вищої математики, фізики та загальноінженерних дисциплін
ORCID: 0000-0003-3968-4326

**Yurchenko Roman**, undergraduate student, Faculty of Engineering and Technology
**Юрченко Р.А.**, здобувач першого (бакалаврського) рівня вищої освіти, інженерно-технологічний факультет

**Kletskov Oleksandr**, Senior Lecturer Department of Higher Mathematics, Physics and General Engineering Disciplines
**Клєцков О.М.**, старший викладач кафедри вищої математики, фізики та загальноінженерних дисциплін
ORCID: 0000-0003-2587-4647
e-mail: alex.kl87@i.ua

**Sakhno Viacheslav**, Candidate of Physical and Mathematical Sciences,Associate Professor Department of Higher Mathematics, Physics and General Engineering Disciplines
**Сахно В.М.**, кандидат фізико-математичних наук, доцент кафедри вищої математики, фізики та загальноінженерних дисциплін
ORCID: 0000-0002-2314-4547

Dnipro State Agrarian and Economic University, Dnipro
Дніпровський державний аграрно-економічний університет, Дніпро

# SIMULATION OF ENERGY LOSSES DURING THE TRANSPORTATION OF LIQUIDS THROUGH A PIPELINE

# МОДЕЛЮВАННЯ ВТРАТ ЕНЕРГІЇ ПРИ ТРАНСПОРТУВАННІ РІДИН ЧЕРЕЗ ТРУБОПРОВІД

*Pipelines serve as indispensable elements in the transportation of fluids across a wide range of industries. Their operational effectiveness is contingent upon energy losses arising from friction and other contributing factors. This study presents a comparative analysis of analytical methods for quantifying energy losses, with a particular focus on the Darcy-Weisbach equation, and numerical simulations conducted using ANSYS CFX software. The investigation delves into the influence of flow velocity, pipe roughness, and flow regime on energy losses. The results obtained from both analytical and numerical modeling exhibit a high degree of agreement, thereby substantiating the efficacy of numerical methods in optimizing the design of intricate pipeline systems. To illustrate this, a numerical calculation example within a complex pipeline system utilizing ANSYS CFX software is provided.*

***Keywords**: energy losses, Darcy-Weisbach equation, roughness coefficient, Reynolds number, finite element method.*

*Трубопроводи є невід'ємною частиною багатьох промислових процесів, забезпечуючи транспортування рідин і газів. Ефективність їх роботи безпосередньо впливає на економічність та безпеку виробництва. Одним з ключових аспектів аналізу трубопроводів є оцінка втрат енергії, які виникають внаслідок тертя рідини об стінки труби, зміни напрямку потоку, місцевих опорів та інших факторів. Традиційно для оцінки втрат енергії в трубопроводах ви-*

*користовуються аналітичні методи, засновані на фундаментальних законах гідравліки. Одним з найпоширеніших є формула Дарсі-Вейсбаха, яка дозволяє розрахувати втрати тиску на ділянці труби з урахуванням її довжини, діаметра, шорсткості та швидкості потоку. З розвитком обчислювальної техніки все більшого поширення набувають чисельні методи моделювання, які дозволяють з високою точністю досліджувати складні гідродинамічні процеси в трубопроводах. Програмне забезпечення ANSYS CFX є одним з лідерів у цій галузі, надаючи широкий спектр інструментів для моделювання турбулентних потоків, теплообміну та інших явищ. У статті порівняно результати аналітичних розрахунків за формулою Дарсі-Вейсбаха з результатами чисельного моделювання в ANSYS CFX. Було встановлено високу узгодженість між цими двома методами, що підтверджує достовірність чисельного моделювання. Дослідження показало, що на втрати енергії в трубопроводах впливають такі фактори: швидкість потоку — зі збільшенням швидкості втрати енергії зростають, шорсткість трубопроводу — шорсткі стінки труби створюють більший опір потоку, що призводить до збільшення втрат, режим течії — турбулентний режим течії характеризується більшими втратами енергії, ніж ламінарний. Результати дослідження підтверджують можливість використання чисельних методів для оптимізації конструкцій складних трубопровідних систем. Це дозволяє зменшити втрати енергії, підвищити ефективність роботи трубопроводів та знизити експлуатаційні витрати.*

***Ключові слова**: втрати енергії, формула Дарсі-Вейсбаха, коефіцієнт шорсткості, число Рейнольдса, метод скінченних елементів.*

## Problem's Formulation

When transporting liquids or gases through pipelines, energy losses are inevitable. These losses are particularly significant when transporting liquids, as they are associated with overcoming the resistance of viscous fluid flow, which is caused by friction against the pipe walls, local resistances (elbows, valves, expansions, etc.), and changes in the kinetic energy of the flow. Accurate calculation of energy losses during liquid transportation is a crucial task for ensuring efficient pipeline operation.

Determining energy losses during transportation through pipelines of various shapes and geometries poses the challenge of selecting the most effective methods for modeling and calculation. The choice of calculation method depends on the specific conditions of the problem and the required accuracy of the results.

Investigating the flow of liquids through a horizontal pipeline is one of the simplest modeling tasks, yet it allows selecting the most accurate parameters for performing more complex calculations. This increases the accuracy and efficiency of constructing a branched pipeline system and allows calculating the necessary pipeline parameters and selecting equipment for pumping the liquid.

## Analysis of recent research and publications

Numerous studies have been dedicated to investigating fluid flow through pipes of various profiles [1, 2, 3]. Recent research and publications demonstrate significant progress in the application of the Darcy-Weisbach equation for analytical calculations [1, 2] and the use of ANSYS Fluent for numerical modeling [5, 6]. The Darcy-Weisbach equation remains the primary tool for determining pressure losses in fluid transportation systems due to its versatility and reliability [3]. Its application is particularly widespread in cases of steady flow in pipes, channels, and porous media [4].

On the other hand, ANSYS Fluent has become an indispensable tool for numerical modeling, allowing researchers to model complex fluid flow scenarios that are difficult or impossible to solve analytically [5—7]. Its application spans a wide range of fields, including fluid dynamics.

The synergy between analytical and numerical approaches is a constant theme in contemporary research. For instance, the Darcy-Weisbach equation is often used for preliminary assessments and validation of results, while ANSYS Fluent provides deeper insights through comprehensive numerical modeling. Such an approach ensures accuracy and efficiency in the analysis and design of fluid transportation systems.

In conclusion, the combination of traditional analytical methods, such as the Darcy-Weisbach equation, and modern computational tools, such as ANSYS Fluent, reflects the dynamic development of research in the field of fluid mechanics. This enables accurate modeling and optimization of fluid transportation systems, contributing to innovations in engineering and applied sciences

**Formulation of the study purpose**

The objective of this study is to compare the results of analytical, numerical calculations, and computer simulations obtained for a horizontal pipeline in order to determine the optimal parameters that allow for the most accurate modeling of fluid flow through pipelines.

**Presenting main material**

As previously determined, pipelines are an integral part of fluid transportation systems in industry, agriculture, and utilities. The efficiency of these systems largely depends on the level of energy losses that occur during fluid flow. One of the key aspects of pipeline system design is the analysis and modeling of these energy losses. They arise due to a whole range of factors related to the geometry of the pipeline, its length, the viscosity and density of the fluid, the speed of laminar or turbulent flow, depending on the operating mode, the roughness of the inner surface of the pipes, and other parameters.

To ensure optimal and efficient operation of pipeline systems, it is necessary to carefully analyze and understand the influence of each of these processes on the state of energy losses.

Conducting such research involves the use of graphical, analytical, and numerical methods, as well as empirical formulas and correlations.

Analytical methods are based on equations that describe the relationship between flow velocity, head loss or energy loss, and pipeline parameters. The most universal method for calculating frictional head loss in straight sections of a pipeline is the Darcy-Weisbach equation. It is considered one of the key equations that allows for the estimation of energy losses due to friction. The Darcy-Weisbach equation is one of the main tools of a hydraulic engineer. It takes into account such pipeline parameters as the length of the pipeline section, pipe diameter, flow velocity, and hydraulic friction coefficient, which depends on the flow regime and pipe roughness and is determined by the Reynolds number.

Such an analytical method is convenient for analyzing simple systems, but it is quite limited by assumptions that do not take into account all aspects of real conditions.

The main problems of applying the analytical method based on the Darcy-Weisbach equation are, first of all, the idealization of conditions. Most analytical models assume smooth pipe walls, which significantly differs from reality. The roughness of the pipeline surface significantly affects the value of the hydraulic friction coefficient, which itself is quite difficult to find by solving the transcendental Colebrook-White equation and requires numerical iterative procedures.

In addition, most models assume fluid properties with constant density and viscosity. This makes them unsuitable for calculating the flow of fluids with variable properties. As a rule, the models assume steady-state flow regimes. Although in most real pipeline systems, sharp changes and pulsations of pressure and fluid velocity are often observed.

There are also problems with the geometry of pipelines. The equation works well for circular straight sections, but becomes significantly more complicated when the pipeline uses pipes with non-circular cross-sections, and there is the presence of branches, bends, and valves, which is typical for real pipeline systems.

The application of numerical methods allows overcoming these difficulties and taking into account both the complex geometry of pipelines and non-stationary processes. It is possible to numerically investigate the nonlinear relationship between various parameters that affect energy losses and resistance.

One of the effective modern numerical methods is the finite element method (FEM). It allows for the investigation of complex geometries and various flow conditions. This approach divides the pipeline into a finite number of elements for which the equations of fluid motion are solved. Numerical models are a powerful tool for optimizing designs and predicting the behavior of systems under real-world conditions.

The results of modeling energy losses are used for pipeline design, the development of standards, and the improvement of the overall energy efficiency of the system. The selection of the optimal diameter, material, and flow parameters helps to reduce operating costs and increase system reliability.

To determine the validity of using the numerical method (FEM) for this class of problems, a comparison was made between the analytical calculation for the classical problem of energy losses in a horizontal pipeline and its solution using the numerical method based on computer modeling in one of the most common programs based on the finite element method, namely AnsysCFX.

In this study, the problem of determining energy losses in a horizontal pipeline 500 meters long and 0.2 meters in diameter was solved. The flow velocity is within the range of 1—10 m/s. The roughness coefficient is within the range of 0.02—0.06.

Based on theory [2], for a steady flow of an ideal incompressible fluid in a horizontal pipeline, Bernoulli's equation takes the following form:

$$\frac{\rho_1}{\rho g} + \frac{v_1^2}{2g} = \frac{\rho_2}{\rho g} + \frac{v_2^2}{2g} + h_f,$$

where $\frac{\rho_1}{\rho g}$ and $\frac{\rho_2}{\rho g}$ — are the specific pressures at the beginning and end of the pipeline, respectively, $\frac{v_1^2}{2g}$and$\frac{v_2^2}{2g}$—are the specific kinetic energies at the beginning and end of the pipeline, respectively, and $h_f$ — represents the head loss due to friction.

Head losses due to friction will be calculated using the Darcy-Weisbach equation. The Darcy-Weisbach equation is used to calculate head losses in pipelines caused by friction between the fluid flow and the pipe walls. It is a fundamental tool in the analysis of hydraulic systems and is expressed as follows:

$$h_f = \frac{\lambda \cdot L \cdot V^2}{2 \cdot g \cdot D},$$

where: $h_f$ — head loss due to friction, m; $\lambda$ — Darcy friction factor; $L$ — length of the pipeline, m; $V$ — average flow velocity, m/s; $g$ — acceleration due to gravity, m/s²; $D$ — pipe diameter, m.

The friction factor depends on the flow regime (laminar or turbulent) and the characteristics of the pipe surface. The flow regime is determined using the Reynolds number, Re.

$$\mathrm{Re} = \frac{V \cdot D}{v}.$$

The calculation of the Reynolds number for our calculation conditions showed (Tabl. 1) that the fluids have a turbulent flow. For an accurate determination of the friction factor, the Colebrook-White equation is used:

*Table 1*. Comparison of energy losses (analytical and numerical calculations) for different roughness coefficients

| Velocity $V$ m/s | Re, $10^4$ | *E, 10³* J/kg at ε=0,02 | | Relative error % | *E*, 10³ J/kg at ε=0,04 | | Relative error % | *E*, 10³ J/kg at ε=0,06 | | Relative error % |
|---|---|---|---|---|---|---|---|---|---|---|
| | | analytical | numerical | | analytical | numerical | | analytical | numerical | |
| 1 | 2 | 0,1271 | 0,1273 | 0,157 | 0,19462 | 0,19461 | 0,005 | 0,2625 | 0,262, | 0,008 |
| 5 | 100 | 3,177 | 3,174 | 0,094 | 4,865 | 4,862 | 0,062 | 6,565 | 6,563 | 0,030 |
| 10 | 200 | 12,71 | 12,709 | 0,008 | 19,46 | 19,458 | 0,010 | 26,25 | 26,248 | 0,008 |

$$\frac{1}{\sqrt{\lambda}} = -2\log\left(\frac{\varepsilon}{3{,}7D} + \frac{2{,}51}{\mathrm{Re}\sqrt{\lambda}}\right),$$

where ε is the absolute roughness of the pipe wall, m.

Since the head loss due to friction is expressed in units of head $h_f$, we can obtain the equivalent energy loss per 1 kg of fluid.

$$E = h_f \cdot g.$$

Head losses ($h_f$) in a pipeline depend on the length ($L$), diameter ($D$), flow velocity ($v$), and friction factor ($\lambda$). Regression analysis shows that as the flow velocity increases, head losses increase proportionally to the square of the velocity (Fig. 1). This is due to the fact that in the Darcy-Weisbach equation, the term $v^2$ is dominant.

Calculations show that the flow regime is turbulent (Re > 2300), which is typical for high velocities and small pipe diameters. In turbulent flow, head losses and energy losses are significantly higher than in laminar flow, as friction in the flow increases due to the formation of eddies. Changing the roughness coefficient of the pipe wall significantly affects energy losses, so increasing the roughness coefficient from 0.02 to 0.06 doubles the energy losses.

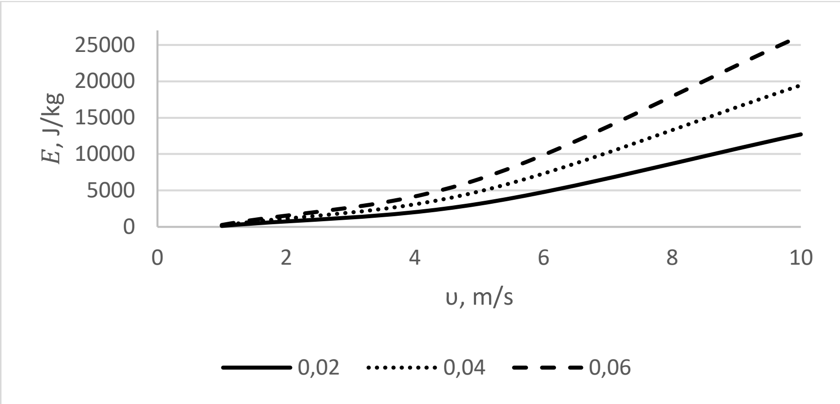


*Fig. 1.* Dependence of energy losses on flow velocity at different values of pipe roughness coefficient

As mentioned above, the Colebrook-White equation for calculating the friction factor is transcendental and requires iterative methods to solve.

The analytical solution is quite complex. To automate the calculation process, Python was used in this work. It is a powerful tool for modeling hydraulic problems, including determining energy losses in pipelines.

In this example, we will calculate the energy losses in a horizontal pipeline using the Darcy-Weisbach equation.

The calculation code is presented below:

```
importmath
# Constants
rho = 1000 # density of water, kg/m³
mu = 0.001 # dynamic viscosity of water, Pa·s
g = 9.81 # acceleration due to gravity, m/s²
# Pipeline parameters
length = 500 # length of the pipeline, m
diameter = 0.2 # diameter of the pipeline, m
roughness_range = [0.02, 0.06] # range of roughness coefficients, m
velocity_range = range(1, 11) # range of flow velocities, m/s
# Function to calculate Reynolds number
defreynolds_number(velocity, diameter, rho, mu):
return (rho*velocity*diameter)/mu
# Function to determine the friction factor using the Colebrook-White equation
deffriction_factor(Re, roughness, diameter):
if Re < 2000: # laminar flow
return 64 / Re
else: # turbulent flow
f = 0.02 # initial approximation for friction factor
for _ in range(20): # iterations for convergence
f = 1/(−2.0 * math.log10(roughness/(3.7*diameter) + 2.51/(Re*math.sqrt(f))))**2
return f
# Main calculation
loop print("Velocity (m/s) | Roughness (m) | Energy loss (J/kg)")
print("−" * 50)
```

for velocity in velocity_range: Re = reynolds_number(velocity, diameter, rho, mu) for roughness in [roughness_range[0], (roughness_range[0] + roughness_range[1])/ 2, roughness_range[1]]:
f = friction_factor(Re, roughness, diameter)
# Darcy-Weisbach formula
head_loss = f*(length / diameter)*(velocity**2)/(2*g) # head loss, m
energy_loss_per_kg = head_loss*g # energy loss, J/kg
print(f"{velocity:10.2f} | {roughness:14.4f} | {energy_loss_per_kg:20.2f}")

The results obtained using this program are in complete agreement with the results of the analytical calculation.

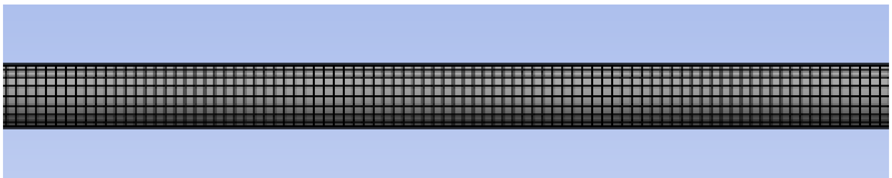

*Fig. 2.* Example of a computational mesh

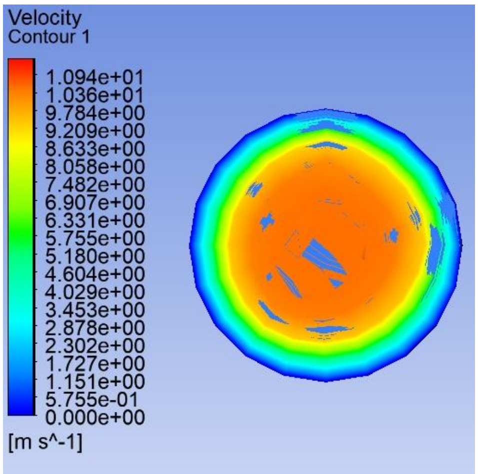


*Fig. 3.* Velocity distribution at the pipe outlet

Numerical analysis was conducted using ANSYS CFX, a software based on the finite element method. ANSYS CFX is employed for modeling laminar and turbulent flow of Newtonian and non-Newtonian fluids, both single-phase and multiphase, in subsonic and supersonic regimes. The numerical analysis methodology involved first creating a geometry model of the required configuration, followed by the assignment of boundary conditions such as "inlet", "outlet", and "wall". Subsequently, a computational mesh (Fig. 2) was generated for numerical calculations using the finite element method. The mesh was refined near the pipe walls to ensure more accurate calculations at the boundary between the two boundary conditions. Calculation parameters included: pipe length — 500 meters, diameter — 0.2 meters, flow velocity within the range of 1—10 m/s, roughness coefficient within the range of 0.02—0.06, and wall thickness — 4.9 mm, Elementsize — 0,2 m, nodes — 12892770, elements— 11618680, Numberofiteration — 100.

One of the key stages of simulation modeling is the interpretation of the obtained calculation results. The processing of simulation results in ANSYS CFX is carried out in a separate Results module.

Numerical analysis using ANSYS CFX allows not only to calculate the final values of various parameters but also to obtain values at any point in the pipeline (Fig. 3).

The following program calculates the energy loss in the ResultsExpressions module:
*E*=(massFlowAve(Pressure)@inlet/massFlowAve(Density)@inlet-massFlowAve(Pressure)@outlet /massFlowAve(Density)@outlet)+(0.5*massFlowAve(Velocity)@inlet*massFlowAve(Velocity) @inlet−0.5*massFlowAve(Velocity)@outlet *massFlowAve(Velocity)@outlet )

Increasing the complexity of the pipeline geometry leads to a more complex calculation program, but does not affect the key output parameters such as inlet and outlet velocities, pressure, and density.

Comparison of the results obtained by analytical methods and FEMdemonstrates good agreement. Thus, the proposed numerical modeling methodology can be applied to more complex and branched pipeline systems.

## Conclusions

The conducted analysis has shown that energy losses in pipelines are significantly influenced by flow parameters such as velocity, pipe diameter, and wall roughness. Analytical methods, particularly the use of the Darcy-Weisbach equation, provide accurate results for simple systems. However, numerical modeling performed in ANSYS CFX has demonstrated high accuracy even for complex conditions and allows for calculations considering real physical processes.

The obtained results confirm the effectiveness of using numerical methods for analyzing and optimizing complex branched pipeline systems. This opens up possibilities for designing branched networks considering various operating scenarios, which contributes to reducing energy consumption and increasing the reliability of transportation systems.

## References

[1] Vozniak, L.V., Gimer, P.R., Merduh, M.I., Panevnik, O.V. (2012). Hidravlika [Hydraulics]. Ivano-Frankivsk: IFNTUNG.
[2] Vambol S.O., Mishchenko I.V., Kondratenko O.M. (2016). Tekhnichna mekhanika ridyny ta hazu [Technical mechanics of liquids and gases]. Kharkiv: NUTZU.
[3] Didur V. A., Zhuravel D. P., Palishkin M. A. (2015). Hidravlika [Hydraulics]. Kherson: Oldi-plus.
[4] Didur V.A., Zhuravel D.P. (2019). Tekhnichna mekhanika ridyny ta hazu [Technical mechanics of liquids and gases]. Melitopol: Color Print.
[5] ANSYS FLUENT 14.5 Theory Guide. (2012). ANSYS Inc. ANSYS Help, [Electronic resource]. Available: https://ansyshelp.ansys.com.
[6] ANSYS. Fundamental FEA Concepts and Applications. A Guidebook for use and Applicability of Workbench Simulation Tools from ANSYS, Inc. [Electronic resource]. Available: https://www.cae.tntech.edu/~chriswilson/FEA/ANSYS/ANSYSguide_fea-concepts.pdf
[7] Finite Element Analysis: Theory and Application with ANSYS. [Electronic resource]. Available: https://www.pinterest.com/pin/ebook-pdf-finite-element-analysis-theory-and-application-with-ansys-4th-edition-dollartree4books-in-2022--1105141196033257914

## Список використаної літератури

1. Возняк Л.В., Гімер П.Р., Мердух М.І., Паневник О.В. Гідравліка: навчальний посібник. Івано-Франківськ: ІФНТУНГ, 2012. 327 с.
2. Вамболь С.О., Міщенко І.В., Кондратенко О.М. Технічна механіка рідини і газу: підручник. Харків: НУЦЗУ, 2016. 300 с.
3. Дідур В. А., Журавель Д. П., Палішкін М. А. Гідравліка. Підручник. Херсон: Олді-плюс, 2015. 624 с.
4. Дідур В.А., Журавель Д.П. Технічна механіка рідини і газу: підручник. Мелітополь: Колор Принт, 2019. 476 с.
5. ANSYS FLUENT 14.5 Theory Guide, ANSYS Inc. ANSYS Help, 2012. [Electronic resource]. Available: https://ansyshelp.ansys.com .
6. ANSYS, Fundamental FEA Concepts and Applications, A Guidebook for use and Applicability of Workbench Simulation Tools from ANSYS, Inc. [Electronic resource]. Available: https://www.cae.tntech.edu/~chriswilson/FEA/ANSYS/ANSYSguide_fea-concepts.pdf
7. Finite Element Analysis: Theory and Application with ANSYS. [Electronic resource]. Available: https://www.pinterest.com/pin/ebook-pdf-finite-element-analysis-theory-and-application-with-ansys-4th-edition-dollartree4books-in-2022--1105141196033257914/

*Надійшла до редколегії 24.01.2025*