

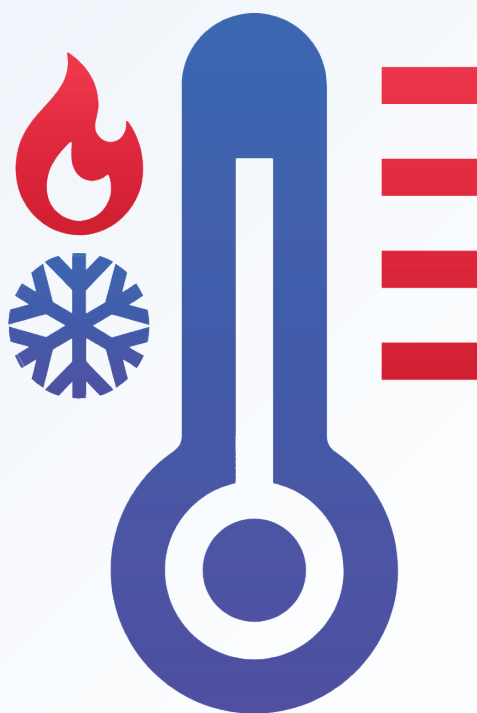
Vol. 47 No. 2 (2026)



ARCHIVES OF THERMODYNAMICS

ISSN: 1231-0956

ISSN: 2083-6023 (Online)



Co-published by



Institute of Fluid Flow Machinery
Polish Academy of Sciences



**Committee on Thermodynamics
and Combustion**
Polish Academy of Sciences



Aims and Scope

The aim of the quarterly journal Archives of Thermodynamics (AoT) is to disseminate knowledge between scientists and engineers worldwide and to provide a forum for original research conducted in the field of thermodynamics, heat transfer, fluid flow, combustion and energy conversion in various aspects of thermal sciences, mechanical and power engineering. Besides original research papers, review articles are also welcome.

The journal scope of interest encompasses in particular, but is not limited to:

- Classical and extended non-equilibrium thermodynamics
 - Thermodynamic analysis including exergy
 - Thermodynamics of heating and cooling
 - Thermodynamics of nuclear power generation
 - Thermodynamics in defence engineering
 - Advances in thermodynamics
 - Experimental, theoretical and numerical heat transfer
 - Thermal and energy system analysis
 - Renewable energy sources including solar energy
 - Secondary fuels and fuel conversion
 - Heat and momentum transfer in multiphase flows
 - Nanofluids
- Energy transition
 - Advanced energy carriers
 - Energy storage and efficiency
 - Energy in buildings
 - Hydrogen energy
 - Combustion and emissions
 - Turbomachinery
 - Thermal and energy system analysis
 - Integrated energy systems
 - Distributed energy generation
 - Thermal incineration of wastes
 - Waste heat recovery.

Supervisory Editors

- K. Badyda, Warsaw University of Technology, Poland
- M. Lackowski, Institute of Fluid Flow Machinery, Gdańsk, Poland

Honorary Editor

- J. Mikielewicz, Institute of Fluid Flow Machinery, Gdańsk, Poland

Editor-in-Chief

- P. Oćłoń, Cracow University of Technology, Cracow, Poland

Section Editors

- P. Lampart, Institute of Fluid Flow Machinery, Gdańsk, Poland
- S. Polesek-Karczewska, Institute of Fluid Flow Machinery, Gdańsk, Poland
- I. Szczygieł, Silesian University of Technology, Gliwice, Poland
- A. Szlęk, Silesian University of Technology, Gliwice, Poland

Technical Editors

- J. Frączak, Institute of Fluid Flow Machinery, Gdańsk, Poland
- S. Łopata, Institute of Fluid Flow Machinery, Gdańsk, Poland

Members of Programme Committee

- D. Kardaś, Inst. Fluid Flow Mach., Gdańsk, Poland (Chairman)
 - J. Badur, Inst. Fluid Flow Mach., Gdańsk, Poland
 - T. Chmielniak, Silesian Univ. Tech., Gliwice, Poland
- P. Furmański, Warsaw Univ. Tech., Poland
 - R. Kobyłeczki, Częstochowa Univ. Tech., Poland
 - S. Pietrowicz, Wrocław Univ. Sci. Tech., Poland
 - J. Wajs, Gdańsk Univ. Tech., Poland

International Advisory Board

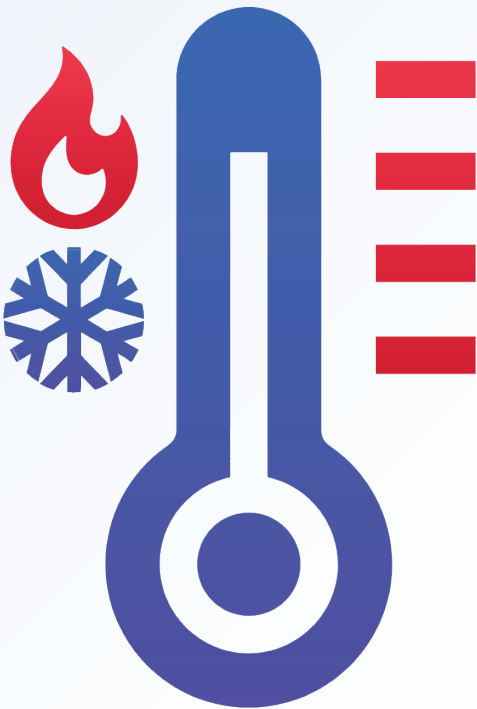
- J. Bataille, Ecole Centr. Lyon, France
 - W. Błasiak, Royal Inst. Tech., Stockholm, Sweden
 - A.C. Benim, Univ. Appl. Sci., Duesseldorf, Germany
 - G.P. Celata, ENEA, Rome, Italy
 - L.M. Cheng, Zhejiang Univ., Hangzhou, China
 - M. Colaco, Federal Univ. Rio de Janeiro, Brazil
 - J.M. Delhay, CEA, Grenoble, France
 - M. Giot, Univ. Catholique Louvain, Belgium
 - K. Hooman, Univ. Queensland, Australia
 - D. Jackson, Univ. Manchester, UK
 - D.F. Li, Kunming Univ. Sci. Tech., China
 - K. Kuwagi, Okayama Univ. Science, Japan
 - J.P. Meyer, Univ. Pretoria, South Africa
- S. Michaelides, Texas Christian Univ., USA
 - W. Muschik, Tech. Univ., Berlin, Germany
 - I. Müller, Tech. Univ., Berlin, Germany
 - H. Nakayama, JAEA, Japan
 - S. Nizetic, Univ. Split, Croatia
 - H. Orlande, Federal Univ. Rio de Janeiro, Brazil
 - M. Podowski, Rensselaer Polyt. Inst., USA
 - R.V. Rao, Sardar Vallabhbhai Nat. Inst. Techn., India
 - A. Rusanov, Inst. Mech. Eng. Probl., Kharkiv, Ukraine
 - A. Vallati, Sapienza Univ. Rome, Italy
 - M.R. von Spakovsky, Virginia Polyt. Inst., USA
 - H.R. Yang, Tsinghua Univ., Beijing, China



ARCHIVES
OF THERMODYNAMICS

ISSN: 1231-0956

ISSN: 2083-6023 (Online)



Co-published by

Institute of Fluid Flow Machinery
Polish Academy of Sciences

Committee on Thermodynamics and Combustion
Polish Academy of Sciences



Editorial Office

IMP PAN Publishers
Institute of Fluid Flow Machinery, Fiszera 14, 80-231 Gdańsk, Poland,
Phone: (+48) 58-341-12-71 int. 230, e-mail: redakcja(at)imp.gda(.)pl
<https://www.imp.gda.pl/archives-of-thermodynamics/>
Journals PAS – Electronic Library Polish Academy of Sciences
<https://journals.pan.pl/ather>

Subscription outside Poland

From 2024, Archives of Thermodynamics appear only in electronic version as an open access journal. However, printed archive volumes or issues are still available. The price of a full volume outside Poland is **120 EUR**. The price of a single issue is **30 EUR**. Archived volumes or issues are available on request. Orders should be sent directly to IMP PAN Publishers:
Institute of Fluid-Flow Machinery,
Fiszera 14, 80-231 Gdansk, Poland,
e-mail: redakcja(at)imp.gda.pl or jfrk(at)imp.gda(.)pl or jrybka(at)imp.gda(.)pl.
Payments should be transferred to the bank account of IMP PAN:
IBAN 28 1130 1121 0006 5498 9520 0011 at Bank Gospodarstwa Krajowego; Code SWIFT: GOSKPLPW.

Prenumerata w Polsce

Od 2024, czasopismo ukazuje się tylko w formie elektronicznej otwartej. Osiągalne są jednak wydania papierowe archiwalne. Cena pojedynczego numeru wynosi **50 PLN**. Cena rocznika **200 PLN**. Zamówienia z określeniem numeru (rocznika), nazwiskiem i adresem odbiorcy należy kierować bezpośrednio do Wydawcy:
Instytut Maszyn Przepływowych PAN
ul. Fiszera 14, 80-231 Gdańsk,
e-mail: redakcja(at)imp.gda(.)pl lub jfrk(at)imp.gda(.)pl lub jrybka(at)imp.gda(.)pl.
Wpłaty prosimy kierować na konto Instytutu Maszyn Przepływowych PAN:
nr 28 1130 1121 0006 5498 9520 0011 w Banku Gospodarstwa Krajowego.

Articles in Archives of Thermodynamics are abstracted and indexed within:

Arianta • Baidu Scholar • BazTech • Cabell’s Directory • Chemical Abstracts Service (CAS) – CAplus • CNKI Scholar (China National Knowledge Infrastructure) • CNPIEC • EBSCO • Elsevier – SCOPUS • Genamics JournalSeek • Google Scholar • Inspec • Index Copernicus • J-Gate • Journal Citation Reports (ESCI) • Journal TOCs • Naviga (Softweco) • POL-index • Polymer Library • Primo Central (ExLibris) • ProQuest • ReadCube • SCImago (SJR) • TDNet • TEMA Technik und Management • Ulrich’s Periodicals Directory/Ulrichsweb • Web of Science (Core Collection) • WorldCat (OCLC)

ISSN 1231-0956

ISSN 2083-6023 (Online)

Copyright © 2026 by the Authors under license CC BY-NC-ND 4.0.

Publication funding of this journal is provided by resources of the Polish Academy of Sciences and the Institute of Fluid Flow Machinery

Contents

1) Dilbag Singh Mondloe, Harish Kumar Ghritlahre, Gajendra Kumar Agrawal, Kamini Rathore, Hemant Naik, Onika Parmar Effects of roughness geometry on heat transfer, fluid flow and thermo-hydraulic performance of solar air heaters: A comprehensive review.....	5–57
2) Lamia Medjahed, Hamza Bousbaa, Mohammed Benramdane, Bouchra Nor El Hou da Hagani Three-dimensional modelling of combustion and emissions characteristics in direct-injection diesel engines using detailed chemistry based on the SAGE combustion model.....	59–68
3) S.K. Abhilasha, Manoranjan Parhi, Harjot Singh Gill, Nagaraj Patil, M. Um eshaiah Delineating the influence of solvent fraction and temperature-dependent heat source/sink on magneto-convective flow: A response surface methodology and an artificial neural network approach.....	69–85
4) Marek Majdak Influence of spacing of waterwall tubes on their operating parameters.....	87–97
5) Vijay Singh Bisht, Desh Bandhu Singh, Pushpendra Kumar, Prabhakar Bhandari, Rupesh Kumar Tipu, Sandeep Singh, Ankur Singh Bist Investigation of Heat Transfer and Flow Characteristics in Solar Air Heaters with a Spherical Turbulator Using Machine Learning Models	99–118
6) Kamil Bokisz, Adam Ruziewicz, Maciej Cholewiński Preliminary zero-dimensional thermodynamic analysis of a gas turbine using EBSILON Professional software	119–130
7) Md Sayeed Iftekhar Yousuf, Jahangir Alam, M.G. Murtaza, Md Touhidul Islam, Md Jashim Uddin, Tasnim UI Islam, Md Fazley Elahi Non-Similar Analysis of MHD Ferro-Nanofluid Flow over an Exponential Stretching/Shrinking Sheet with Particle Effects.....	131–140
8) Ahmed Jan, Farida Aslam, Rubina Adnan Convective Transfer Analysis of Magnetised Ternary Hybrid Nanofluid over a Linear Stretching Sheet: Non-Similar Solutions	141–150
9) Jaya Bohra, Vijay Singh Bisht, Prabhakar Bhandari, Kshitij Pandey, Sandeep Singh, Ankur Singh Bist, Lalit Ranakoti, Shivasheesh Kaushik, Kapil Kumar Sharma Exergy-Based Performance Assessment of a Solar Air Heater Equipped with Zig-Zag Structured Baffles	151–157
10) Shamkant Desale, Premendra Bansod, Arakerimath Rachyya, Srinidhi Campli Experimental Analysis of Energy Output in Evacuated Tube Heat Pipe Collector with Nanofluids	159–165
11) Patrick Dumaz Efficiency of Energy Conversion Cycles in Finite Time Thermodynamics: The Case of Nuclear-Driven Carnot Cycle	167–178
12) Sumanraju Veerabathini, Narsimhulu Sanke, Ramamohan Rao Tinnavelli Performance evaluation of cogeneration plant at various turbine inlet temperatures.....	179–186

13) Suneetha Sangapatnam, Gunisetty Ramasekhar, Anusuiya Chakravarty, Venugopal Pallapuz
Theoretical analysis on bio-magnetic Casson hybrid nanofluid blood flow
through a stenotic artery with a heat source/sink..... 187–199

14) Rajkumar Thamizhvel, Natarajan Vaithianathan, Narayanan Sethuraman,
Subramanian Vinodraj, Lenin Ambetharasu, Palani Surendar
Production and characterisation of pyrolysis oil from discarded medical plastic (syringe) waste
with common rail direct injection engine assessment 201–210



Co-published by
Institute of Fluid-Flow Machinery
Polish Academy of Sciences
Committee on Thermodynamics and Combustion
Polish Academy of Sciences

Copyright© 2026 by the Authors under licence CC BY-NC-ND 4.0

<http://www.imp.gda.pl/archives-of-thermodynamics/>



Effects of roughness geometry on heat transfer, fluid flow and thermo-hydraulic performance of solar air heaters: A comprehensive review

Dilbag Singh Mondloe^a, Harish Kumar Ghritlahre^{b*}, Gajendra Kumar Agrawal^c,
Kamini Rathore^d, Hemant Naik^e, Onika Parmar^f

^aDepartment of Mechanical Engineering, Chhattisgarh Institute of Technology (CGIT), Jagdalpur, Chhattisgarh 494001, India
^bDepartment of Energy and Environmental Engineering, Chhattisgarh Swami Vivekanand Technical University, Bhilai, Chhattisgarh 491107, India
^cDepartment of Mechanical Engineering, Government Engineering College, Bilaspur, Chhattisgarh 495009, India
^dDepartment of Civil Engineering, Rungta College of Engineering and Technology Bhilai, Chhattisgarh 490024, India
^eDepartment of Mechanical Engineering, O. P. Jindal University Raigarh, Chhattisgarh 496109, India
^fDepartment of ET&T, Rungta College of Engineering and Technology Bhilai, Chhattisgarh 490024, India
*Corresponding author email: harish.ghritlahre@gmail.com

Received: 05.07.2024; revised: 04.11.2025; accepted: 04.12.2025

Abstract

In the present review article, an effort has been made to review various experimental and numerical research, which include heat transfer rate and fluid flow characteristics of a modified solar air heater (SAH) using various artificial roughness. As it is clear that artificial roughness enhances the heat transfer and performance, in this review study, various types of roughened SAH performances have been examined. These various roughness patterns are V-shape, W-rib, arc shape, spherical ball and S-type, and their placement arrangements, like transverse, inclined and longitudinal, are discussed in this article. The comparative studies of thermo-hydraulic performance with heat transfer and fluid flow investigations of various roughened SAHs have been described to identify optimal performance parameters to select the optimum configuration that can be further used for various applications. In addition to this, various correlations of the Nusselt number and friction factor developed by various investigators have also been reported. The main aim of this study is to evaluate the SAH performance using various roughness patterns. It also provides several essential recommendations for effectively designing and implementing SAHs. This review encompasses a comprehensive analysis of 178 research articles published between 1988 and 2024, focusing on advancements in the field. The findings showcase the peak values for the Nusselt number as 7.61, the friction factor reaching 6.48 times that of a smooth duct for discrete multiple-arc ribs, and the maximum thermal-hydraulic performance reported at 4.73 for dimple or protrusion ribs.

Keywords: Solar air heaters; Roughness geometry; Heat transfer; Fluid flow; Thermo-hydraulic performance

Vol. 47(2026), No. 2, 5–57; doi: 10.24425/ather.2026.158685

Cite this manuscript as: Mondloe, D.S., Ghritlahre, H.K., Agrawal, G.K., Rathore, K., Naik, H., & Parmar, O. (2026). Effects of roughness geometry on heat transfer, fluid flow and thermo-hydraulic performance of solar air heaters: A comprehensive review. *Archives of Thermodynamics*, 47(2), 5–57.

1. Introduction

Various energy resources are utilised in industry, transport, the residential sector, etc. The energy sources widely used are natural gas, coal, fossil fuel, oil, renewable and other sources. It is

a fact that sustainable energy sources are a global problem. Across the globe, treaties have been employed and implemented to check and restrain global warming, climate change and energy depletion quickly. These resources are fixed in quantity, but at the same time, their exploration and utilisation are hugely

Nomenclature

A_c	– surface area of collector, m ²
A_o	– area of orifice meter, m ²
B/S	– relative roughness length
C_d	– coefficient of discharge
C_{pf}	– air specific heat, J/(kg K)
c/H	– relative baffle height
d_b	– ball diameter, m
d_p	– perforated diameter, m
d/D	– relative print diameter
d/w	– relative gap position of additional gap in symmetrical rib
d/x	– relative gap position
D_h	– hydraulic diameter, m
D_j	– jet diameter, m
e	– height of roughness (rib height), mm
e/d_d	– dimple depth to diameter ratio
$e/D, e/D_h$	– relative roughness height
e/w	– rib aspect ratio
e^+	– roughness Reynolds number
e'/e	– rib chamfered aspect ratio
f	– friction factor
g	– width of gap, gap of discrete rib, m
$g/e, g_w/H_r$	– relative gap width
g/P	– relative groove position
g'/e	– relative gap size of additional gap in symmetrical gap elements
$G(e^+)$	– heat transfer roughness function
G_d/L_v	– relative gap distance
h	– heat transfer coefficient, W/(m ² K)
h/H	– relative height ratio (for fins and ribs)
h/d_p	– relative perforated ratio
H	– height of the duct, m
H_j	– jet height, m
H_r	– rib height, m
k	– thermal conductivity, W/(m K)
L	– length of the test section in the duct, m
L/e	– relative long way length
$\dot{m}_f$	– mass flow rate of air, kg/s
N_g	– number of main gaps
Nu	– Nusselt number
O	– hole position from baffle base, m
p/P	– relative staggered pitch
P	– perimeter, roughness (rib) pitch, m
ΔP_o	– pressure drop, Pa
P/e	– relative roughness pitch
P/g	– relative pitch to gap ratio
P/H	– relative rib pitch
Pr	– Prandtl number
q	– solar irradiation, W/m ²
$\dot{Q}_u$	– useful heat gained by air, W

r_d	– depth of dimple cavity, m
r/e	– depth of dimple cavity, m
r_d/D_h	– relative dimple depth
$R(e^+)$	– friction roughness function
Re	– Reynolds number
S/e	– relative short way length
s/H	– relative baffle pitch
St	– Stanton number
T	– temperature, K
Tu	– turbulence intensity
V	– air velocity in the duct, m/s
x/r_d	– longitudinal pitch ratio
X/D_h	– streamwise pitch ratio
y/r_d	– transverse pitch ratio
Y/D_h	– spanwise pitch ratio
w	– rib width, m
w/e	– relative rib width
W	– duct width, m
W_{AC}	– width of discrete multi-arc shaped rib, m
W/H	– duct aspect ratio
W/w	– width of duct to width of collector

Greek symbols

α	– angle of arc, °
β	– ratio of orifice meter to pipe diameter
β_0	– open area ratio
ρ	– density, kg/m ³
ϕ	– wedge/chamfer angle, °

Subscripts and Superscripts

a	– air
avg	– average
B	– baffle
f	– fluid
$\hat{f}_i$	– air inlet
$\hat{f}_o$	– air outlet
m	– mean
np	– non-perforated
p	– plate, perforated
s	– smooth

Abbreviations and Acronyms

ANN	– artificial neural network
CCD	– central composite design
CFD	– computational fluid dynamics
LCT	– liquid crystal thermography
PCM	– phase change material
SAH	– solar air heater
SPR	– sinusoidal protrusion rib
STC	– solar thermal collector
THP	– thermo-hydraulic performance

expedited to meet the energy demand of the growing populations and other development works. Therefore, the governments emphasise using renewable energy sources to meet the demand at the required rate. Solar energy is an abundantly available, clean, green, and never-ending energy source that can meet the energy demand. Many solar thermal systems are available, such as solar space heating, solar water heating, concentrated solar

power, solar air conditioning, solar crop drying, solar cooking and solar ponds. These are solutions that utilise sunlight to convert it into thermal energy. There are several major types of solar thermal technologies in use. These are unglazed solar collectors, transpired solar air collectors, flat-plate solar collectors, evacuated tube solar collectors and concentrating solar systems. But due to high cost and sustainable use, the focus has always been

on solar air heaters. Many types of air heaters are available, such as duct heaters, enclosure heaters, forced air heaters, heat torches, flame heaters, radiators, space heaters and room heaters, and tubular heaters. The utilisation of solar air heaters is encouraged due to their easy installation, relatively low cost and portability. However, these systems have certain drawbacks. One significant limitation is the low heat transport capacity of the absorber sheet, which hinders efficient heat transfer. Another factor is the low thermal conductivity of the absorber sheet, which results in a reduced heat transfer rate between the absorber sheet and the flowing air. Consequently, the overall effectiveness and utilisation of solar air heaters are diminished. To overcome these limitations, researchers have focused on creating turbulence inside solar air heaters. The objective is to disrupt the laminar sublayer formed near the absorber plate and enhance the heat transfer rate. However, passive methods used to generate turbulence have disadvantages, such as increased fluid friction and the need for higher pumping power.

Researchers continue to explore alternative approaches to address these challenges and improve the performance of solar air heaters. By developing innovative methods and designs, the goal is to enhance heat transfer efficiency without significantly increasing fluid friction or energy consumption. There are many techniques used by various researchers, such as evaluations of solar air heaters (SAH) in terms of thermo-hydraulic performance [1], thermal performance [2], single pass [3], double pass [4], parallel pass [5] and longitudinal fins [6]. Limited researchers investigated experimentally and numerically various types of artificial roughness, such as arc shaped [7], apex up-down arc shaped [8], concave dimple [9], winglets [10], multi-V shaped [11], S-shaped [12], V-corrugated [13], aluminium cans as fins [14], porous fins [15], packed bed [16], porous bed [17], wire mesh [18], corrugated fins [19], wavy fins [20], turbulators [21] and SAH made from plastic-metal waste [22], etc. for increasing the performance of SAHs. The utilisation of irregular surfaces is a widely employed and effective technique for enhancing heat transfer in solar air heaters. This passive approach involves ap-

plying artificial roughness on the underside of the absorber plate using various methods, such as ribs, grooves, dimples, winglets, baffles, twisted tapes, mesh wires, and more. These modifications create disturbances in the flow, promoting turbulence and improving heat transfer. In Fig. 1, a schematic diagram shows different types of heat transfer augmentation techniques implemented in solar air heaters.

The artificial roughness is strategically placed on the underside of the blackened absorber plate to enhance heat transfer, while keeping an increase in pumping power to a minimum. This approach ensures that the disturbance or turbulence is created in close proximity to the heated surface, specifically targeting the laminar sublayer. By implementing these modifications, solar air heaters can achieve improved heat transfer performance without requiring significant additional pumping power. This technique allows for efficient utilisation of the available thermal energy and enhances the overall effectiveness of the solar air heating system.

The primary objective of this study is to review experimental and analytical work on the performance of solar air heaters using the concept of artificially roughened solar air heaters. This analysis provides researchers with an energy efficiency concept and a strategy for designing and operating industrial operations. In addition, this research work also involves an extensive examination of various roughness geometries employed in solar air heaters and their impact on the mechanisms of heat transfer and fluid flow. The study focused on understanding how different surface textures affect the exchange of heat and movement of fluids within SAH. Further, this paper identifies research gaps for future work and is beneficial for those working with artificially roughened solar air heaters.

From the literature review (Table 1), it has been observed that many review papers have been published on solar air heaters, particularly on roughened SAHs. But very few reviews have been conducted on the heat transfer and fluid flow of solar air heaters. In this area, there is a large opportunity in the near future for increasing the performance of SAHs with novel geometry

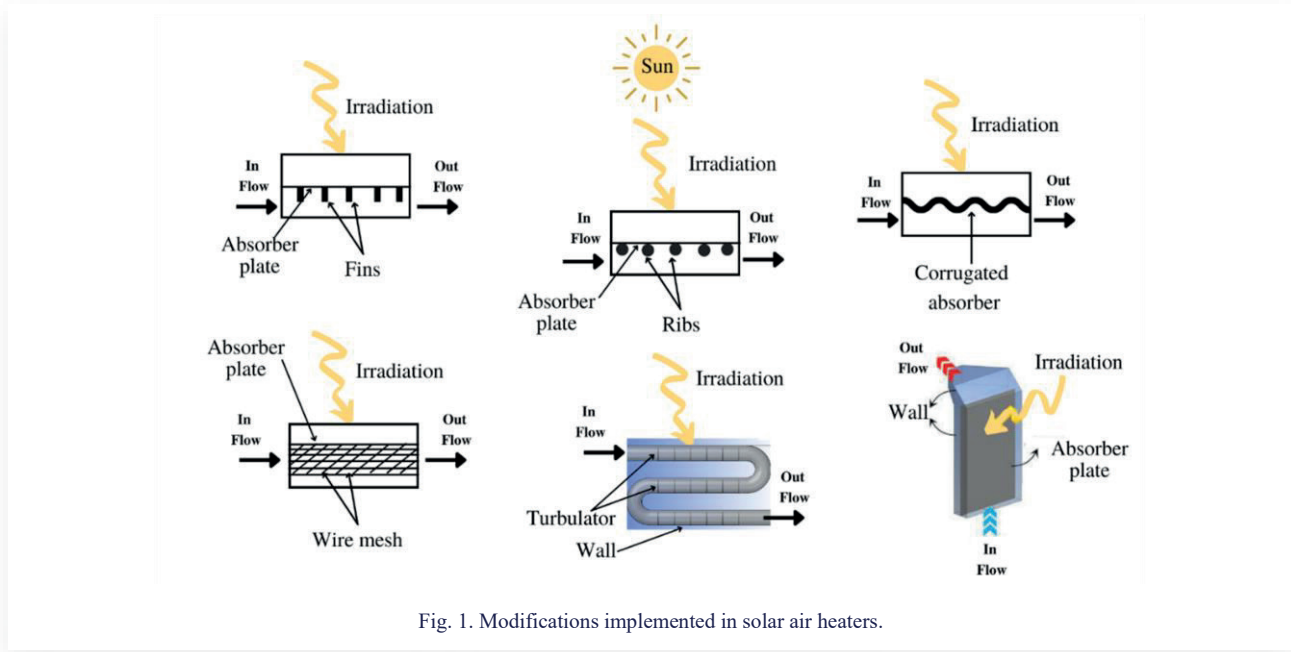


Fig. 1. Modifications implemented in solar air heaters.

Reference	Year	Studied issues
Varun et al. [23]	2007	Heat transfer (Nu) and friction (f) characteristics, correlations developed for Nu and f .
Gupta and Kaushik [24]	2009	Heat transfer, friction characteristics, energy and exergetic efficiency, effective efficiency.
Hans et al. [25]	2009	Heat transfer and friction characteristics, and correlations developed for Nu and f
Chamoli et al. [26]	2012	Heat transfer, friction characteristics and flow pattern.
Chamoli et al. [4]	2012	Fins, packed bed energy analysis, thermal efficiency, heat transfer enhancement.
El-Sebaili and Shalaby [27]	2012	Detailed description, fundamentals and previous work performed on solar dryers and solar air heaters.
Yadav and Bhagoria [28]	2013	CFD analysis, design parameters for improved performance include heat transfer, mass and momentum transfer, optimal design parameters and turbulence model.
Yadav and Thapak [29]	2014	Numerical analysis, developed codes in MATLAB-7, heat transfer, friction characteristics, and correlations developed for Nu and f .
Gawande et al. [30]	2014	Numerical and experimental study, thermal performance evaluation, heat transfer, friction characteristics, and correlations developed between Nu and f .
Kumar et al. [1]	2014	Exergetic efficiency, thermo-hydraulic performance (THP), thermal performance
Alam et al. [31]	2014	Numerical analysis, different turbulators used, heat transfer, friction characteristics and correlations developed for Nu and f .
Alam et al. [32]	2014	Experimental study, different turbulence generators, heat transfer, friction characteristics and correlations developed for Nu and f .
Mund et al. [33]	2014	Heat transfer, friction characteristics and correlations developed for Nu and f .
Anil Kumar Patil [34]	2015	Fluid turbulence and heat transfer mechanism, rib geometries, Reynolds number and temperature rise parameter, effective efficiency.
Sharma and Kalamkar [35]	2015	Different roughness geometrical elements, thermo-hydraulic performance, heat transfer and friction characteristics.
Kumar and Kim [36]	2015	Used different ribs, fins and baffles; heat transfer, friction characteristics, and correlations developed for Nu and f
Lanjewar et al. [37]	2015	Used different rib geometries; heat transfer, friction characteristics, and correlations developed for Nu and f
Sheikholeslami et al. [38]	2015	Different turbulators used on rib geometries; heat transfer, friction characteristics and heat transfer performance.
Singh et al. [39]	2015	Thermo-hydraulic performance comparison, heat transfer, friction factor, relative roughness pitch.
Saxena et al. [40]	2015	Optimising the dimensions, improving thermal performance.
Kumar et al. [41]	2016	Experimental and numerical analysis, triangular duct; heat transfer, friction characteristics and heat transfer performance.
Aravindh and Sreekumar [42]	2016	Used different modifications on rib geometries; heat transfer, friction characteristics and heat transfer performance.
Sharma and Kalamkar [43]	2016	CFD analysis, modelling, different meshing type, suitable turbulence model, thermo-hydraulic performance.
Kumar and Kim [44]	2016	Energy efficiency, heat transfer enhancement, turbulence models, thermo-hydraulic performance, RNG k - ϵ model.
Ravi and Saini [45]	2016	Experimental and theoretical investigations, performance enhancement techniques, a double pass system, thermal performance, and correlations developed.
Gawande et al. [46]	2016	CFD analysis, modelling, different meshing type, operating and system parameters, suitable turbulence model, thermo-hydraulic performance.
Yadav and Thapak [47]	2016	Experimental and CFD approaches on different roughness elements for heat transfer, friction characteristics and thermo-hydraulic performance.
Kabeel et al. [48]	2017	Experimental and theoretical study, different modifications on the absorber plate, recycling process for heat transfer, friction characteristics and thermo-hydraulic performance.
Alam and Kim [49]	2017	Experimental study on different roughness elements; heat transfer, friction characteristics and thermo-hydraulic performance.
Singh and Singh [50]	2018	Experimental and theoretical study, different modifications on the absorber plate, flow visualisation to asses heat transfer, friction characteristics and thermo-hydraulic performance.
Singh et al. [51]	2018	Experimental study on different roughness elements for heat transfer, friction characteristics thermo-hydraulic performance, thermal efficiency, effective efficiency and energy – exergy performance.
Kumar et al. [52]	2018	Energy efficiency, blockage height, blockage arrangement, thermal hydraulic performance, air passage.
Chauhan et al. [53]	2018	Jet impingement, heat transfer, friction factor, multi criteria decision making, thermodynamic behaviour, control factors on heat transfer and friction characteristics.

Reference	Year	Studied issues
Elsheikh et al. [54]	2019	Artificial neural network (ANN) method for optimising and predicting the performance of different solar energy devices.
Hu and Zhang [55]	2019	Correlation development on performance improvement method, reorganisation of flow of air stream.
Prakash and Saini [2]	2019	Experimental study, different modifications on absorber plate, flow visualisation for heat transfer, friction characteristics and thermo-hydraulic performance.
Chhapparwal et al. [56]	2019	Experimental, different modifications on absorber plate, physics involved in flow visualisation for heat transfer, friction characteristics and thermo-hydraulic performance.
Ghritlahre and Sahu [57]	2020	Experimental study on different roughness elements for heat transfer, friction characteristics, thermo-hydraulic performance, thermal performance, effective efficiency and energy – exergy performance.
Vengadesan and Senthil [58]	2020	Experimental study on different modification on SAH for heat transfer, friction characteristics, thermo-hydraulic performance, thermal performance.
Yadav and Prasad [59]	2020	Mathematical model, codes in MATLAB, thermal efficiency, thermo-hydraulic efficiency.
Arunkumar et al. [60]	2020	Thermal performance, design considerations, geometries used, flow conditions and their influence on turbulence; heat transfer, friction characteristics and thermo-hydraulic performance.
Nidhul et al. [61]	2021	Thermo-hydraulic performance, correlations for heat transfer and friction factor, effective efficiency and energy – exergy performance.
Chamarthi and Singh [62]	2021	Energy and exergy efficiencies, performance enhancement, solar air heater, testing standards.
Ghritlahre et al. [63]	2021	Solar air heaters, porous bed, artificial roughness, artificial neural network, learning algorithm (LA), multi-layer perceptron (MLP).
Ghritlahre et al. [64]	2022	Solar air heater, energy analysis, exergy analysis, thermal performance.

and various parameters. Thus, the present systematic review work on the heat transfer, fluid flow characteristics and thermo-hydraulic performance of roughened solar air heaters is necessary to identify existing research gaps, while also incorporating the latest published studies.

In the present paper, a comprehensive review on heat transfer, fluid flow characteristics and thermo-hydraulic performance of roughened solar air heaters has been reported. Various geometries of roughness, which were used by many researchers, have been discussed. The main objectives of the present review are as follows:

1. To study the heat transfer and fluid flow characteristics of various roughness geometries.
2. To find out the optimal thermo-hydraulic performances in various roughness geometries.
3. To report the developed Nusselt number and friction factor correlations of various roughened solar air heaters.
4. To find out the research gaps for future work.

2. History of development of a solar air heater

Since ancient times, the sun’s energy has been used for many purposes, such as crop drying, vegetable drying, cloth drying and making salts, etc. In 1877, Daniels and Duffie [65] were the first to report that solar energy can be stored in iron. The air passed through the iron and got heated, and then it was used for heating the room.

In 1881, an American researcher E. Morse, structured and developed an accredited SAH. This SAH was designed with a wooden-framed cabinet with a metallic absorber sheet, which was covered with a transparent glass cover. This SAH works on the concept of convection; the solar radiation is absorbed by

the steel sheet absorber plate, and air flows through the cabinet, where it absorbs heat from the absorber sheet and transfers it to the blowing air.

This system came into light, but still progress is needed on various modification options in SAH [66]. During this time, several solar-heated homes have been developed and operated in various places around the world. Between 1946 and 1949 in Massachusetts, SAH was modified using a chemical compound, which inherently has the ability to absorb heat and emit heat energy to the air. This technology is known as phase change material (PCM), which has wide applications in various industries and households. At that time, up until 1978, new and novel designs of houses were developed at the Massachusetts Institute of Technology (MIT). It is also observed that many stakeholders, i.e. industries, manufacturing, etc., are designing new solar air heaters for applications such as air heating, crop heating and vegetable heating. In the past few years, after detailed studies, many researchers have developed this solar system for many applications. These studies are summarised in Table 2.

3. Basic construction of a solar air heater

3.1. Basic design of a solar air heater

The basic design of a solar air heater is described in Table 3. SAH is mainly designed with a frame, absorber or collector plate, glass cover, insulation, black interior surfaces, and the inlet and outlet sections. Solar radiation falls upon the absorber plate, as shown in Fig. 2. The thermal energy of the solar radiation is absorbed by the plate, which is then conveyed to the following air through the duct.

Table 2. The developments in the past few years.

Researchers/ Inventors)	Year	City/Place/ Country	Novelty of the invention
Cabot [67]	1938	Boston	Groundwork for research on fully functional solar air heating took place at the Massachusetts Institute of Technology (MIT). The developed solar thermal system was designed to meet almost one-third of the total heating requirement in Boston.
Telkes and Raymond [68]	1949	Dover	Firstly, solar homes were defined at Dover (MIT). The solar homes worked on the principle of solar thermal energy storage by using fusion of NaSHs. The primary objective was to meet the heating load for consecutive 5 days.
Bliss [69]	1955	Arizona desert	A fully functional solar-heated house has been designed and developed in the desert of Arizona. The system comprises air stream flow through combined heater and thermal energy storage (TES) system.
Lof [70]	1959	Denver	Solar air heaters have been developed by using overlapped transparent glass as an absorber plate in a solar collector. They have also utilised pebble bed as solar thermal energy storage, while developing a dwelling in Denver. During 1963, 1964, 1976 and 1978, the researchers have examined the SAHs with all variations and reported the evidence.
Doherty [71]	2007	USA	The system utilised two novel features to increase the mean temperature of the system. They have given a solution to prior problems with mixing of cold fluid stream with hot air. They reported an increase in the system efficiency.
Vachon [72]	2010	USA	They achieved higher thermal efficiency by using the principle of heating and heat recovery in a solar heat collector. This system comprises glazing in contact with air. The spacing used in glazing over the back surface was the reason for higher performance of the system.
MacKay et al. [73]	2011	USA	Air heating was effective using a modular method. The modular system contained different sizes and angles to get maximum absorption of incoming thermal energy. The applications vary from industries, households and agriculture, etc.
Ryan [74]	2011	USA	The boundary of the heating system was defined at the verge between the absorber plate and the plenum. This system was employed to support the surface, a mechanism for flowing air stream and other parts of SAHs.
Hao [75]	2013	China	This novel SAH comprises a frame, a sheet of glass, supports and two ducts with a roof at the top, and there is a greenhouse at the bottom. The advantage of the design was that it achieved high thermal efficiency.

Table 3. The basic design of a solar air heater.

Component	General specification/ dimensions/materials	Purpose
Collector or absorber plate	An absorber plate is made of GI or aluminum and painted black to maximise radiation absorption.	This collects solar energy and transmits it to moving air.
Insulation	Generally, material used as glass wool or mineral wool up to 50 to 80 mm thickness.	It is provided to prevent heat losses from the bottom and sides of the ducts.
Matte black interior	It is matte black paint/ sticky powder.	The interior surfaces of test sections are painted with black matte paint to maximise solar absorption.
Glass cover	Glass covers with a thickness of 3 to 4 mm are commonly utilised.	The use of a glass cover over the top of the absorber plate reduces convection and re-radiation losses from the absorber surfaces to the environment.
Glazing	Glaze sheet is made of carbonate, acrylic and tempered glass materials.	Glazing is provided on the top surface of the glass cover to give adequate solar radiation absorption.
Inlet/outlet section	Bell mounted wooden inlet section and trapezoidal wooden outlet section.	The inlet and outlet sections are designed to make air intake and discharge easier.
Frame	Generally, the frame is made of woods or metal.	To provide support for the SAH body components.

3.2. Classification of SAHs

Solar air heaters are classified into various categories, such as flow patterns, applications, hybriding, collector (absorber) materials, glass covers, flow types and collector (absorber) surface type. The classification of SAHs is described in Table 4.

3.3. Applications of SAHs

Solar air heaters are mainly used for heating air for various purposes. Some important applications of SAHs are shown in Fig. 3.

Table 4. Classifications of SAHs.

Main classification	Subclassification			
	A	B	C	D
The collector cover (glass)	Bare plate	Single	Double	–
The absorber material	Metallic	Non-metallic	Matrix	–
The shape of absorbing surface	With slate	With porous media	With fins	–
Type of absorber flow pattern	Over	Under	On both side	–
Type of flow	Single pass	Double pass	Double flow	Triple pass
Hybrid collectors	Air	Water	Water-air	–
Applications	Drying	Space heating	Preheating	Cooling

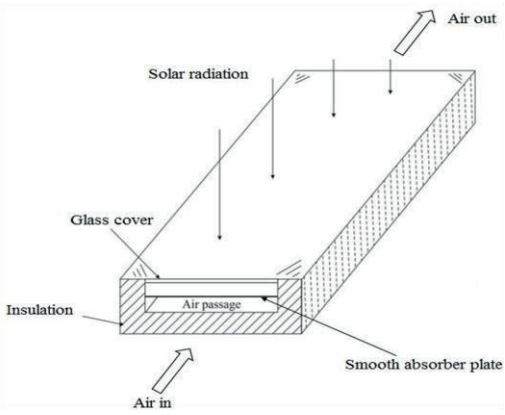


Fig. 2. Convectional solar air heater (drawn after [58]).



Fig. 3. Different applications of SAHs.

4. Performance evaluation of solar air heaters

Solar air heaters can be evaluated by both experimental and analytical approaches. The parameters under evaluation include the plate temperature of an absorber sheet, the air stream temperature, the rate of mass flow, and the thermal and thermo-hydraulic performance. These parameters are explained below in brief.

4.1. Mass flow rate

Mass flow rate is the function of the associated pressure drop through the SAH. The formula utilised for the mass flow rate of fluid is:

$$\dot{m}_f = C_d A_o \left[\frac{2 \rho_a (\Delta P_o)_a}{1 - \beta^4} \right]^{0.5}, \quad (1)$$

where: C_d – coefficient of discharge, A_o – area of an orifice meter, ΔP_o – pressure drop across an orifice meter, β – ratio of the orifice diameter to the pipe diameter.

4.2. Hydraulic diameter

$$D_h = \frac{4A_D}{P} = \frac{4(WH)}{2(W+H)}, \quad (2)$$

where: W – width of the duct, H – height of the duct, A_D – cross-sectional area of the duct, P – perimeter of the duct.

4.3. Velocity of air

$$V = \frac{\dot{m}_f}{\rho_a A_D}, \quad (3)$$

where $\rho = \rho_a$ denotes the density of air.

4.4. Reynolds number

$$\text{Re} = \frac{\rho V D_h}{\mu}, \quad (4)$$

where μ is the kinematic viscosity.

4.5. Heat transfer coefficient

The heat transfer coefficient (h) is a key factor in evaluating the performance of SAH. This coefficient is influenced by the nature and condition of the absorber surface and the flowing fluid. The heat transfer coefficient can be calculated using the following formula:

$$h = \frac{\dot{Q}_u}{A_c (T_{pm} - T_{fm})}, \quad (5)$$

where A_c represents the surface of a collector, and T_{pm} and T_{fm} are the mean temperatures of the plate and air, respectively.

The evaluation of useful heat gained ($\dot{Q}_u$) by the air is calculated using the formula:

$$\dot{Q}_u = \dot{m}_f C_{pf} \Delta T = \dot{m}_f C_{pf} (T_{fo} - T_{fi}), \quad (6)$$

where: C_{pf} – specific heat at constant pressure of fluid used, T_{fo} – outlet temperature of the fluid, T_{fi} – inlet temperature of the fluid.

4.6. Heat transfer

The ratio of convective heat transfer to conductive heat transfer is evaluated using the Nusselt number for a roughened duct. This dimensionless parameter can be calculated from:

$$Nu = \frac{hD_h}{k}, \quad (7)$$

where k represents the thermal conductivity of the fluid.

Furthermore, the Dittus-Boelter equation can be used to calculate the Nusselt number for a smooth duct of SAH:

$$Nu_s = 0.023Re^{0.8}Pr^{0.4}, \quad (8)$$

where Pr stands for the Prandtl number.

4.7. Fluid flow characteristics

The frictional characteristics in SAH is calculated using the pressure drop (ΔP). The Darcy equation formula can be used to evaluate friction factor for the roughened duct:

$$f = \frac{2\Delta PD_h}{4\rho L_t V^2}, \quad (9)$$

where L_t is the length of a test section. For a smooth duct, the friction factor can be calculated using the modified Blasius equation:

$$f_s = 0.085 Re^{-0.25}. \quad (10)$$

4.8. Thermo-hydraulic performance

The thermo-hydraulic performance in SAH can be calculated using the given formula:

$$THP = \frac{Nu}{Nu_s} \left(\frac{f}{f_s} \right)^{\frac{1}{3}}. \quad (11)$$

5. Roughness concept

Artificial roughness is a technique used to enhance fluid flow and heat transfer by introducing specially designed roughness elements on surfaces. The artificial roughness influences fluid flow patterns and improves heat transfer rates as compared to natural surface roughness, which has a minimal impact on convective heat transfer. The artificial roughness over the absorber plate or sheet has different shapes and sizes, such as ribs, dimples, grit, baffles, wings, protrusions, spherical balls, V-shapes, W-shapes and many others. These various roughness elements have different impacts on the flow behaviour and heat transfer characteristics in the SAH. Different researchers used different methodologies, such as experimental, analytical, and codes in advanced platforms like MATLAB, OpenFOAM, etc., to evaluate the performance of the SAH.

Much research has concluded that the reattachment of a fluid flow after the fluid passes the roughness element has a huge impact on the heat transfer coefficient. Sparrow et al. [76] demonstrated this relationship through experimental and numerical investigations. Additionally, Okamoto et al. [77] found that the maximum heat transfer rate occurred at the reattachment point.

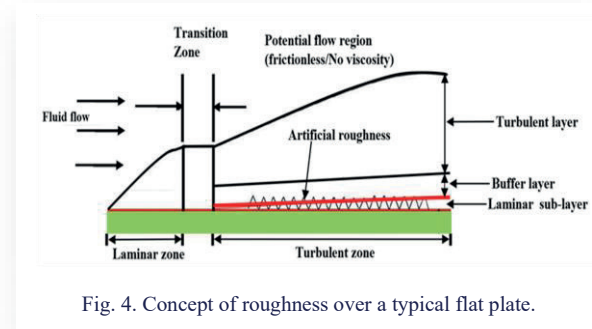


Fig. 4. Concept of roughness over a typical flat plate.

Researchers have also conducted analyses to develop correlations between different dimensionless geometrical governing parameters and operating parameters and heat transfer coefficients (Stanton number), friction factor and Nusselt number, based on their experimental studies. These correlations have been used as performance forecasting tools to evaluate roughened SAH performance, such as thermal and hydraulic performance.

The impact of artificial roughness elements on frictional characteristics, velocity profile and turbulence generation has been studied by various researchers. Nikuradse [78] examined the effect of roughness on friction by using grains of sand for fluid flow through a pipe, while Nunner [79] proposed a fluid flow model that highlights the influence of roughness elements on reducing the thermal resistance in the turbulent near-wall region. Dipprey and Sabersky [80] developed a correlation for heat transfer in a tube roughened by sand grains, and Lewis [81] used smooth, as well as roughened surfaces, to calculate thermal and hydraulic performance by firstly introducing the thermal efficiency. Also, experiment was conducted by Han et al. [82] on a roughened surface to evaluate heat transfer (Nu) and fluid flow characteristics (f), taking into account geometrical variables such as different attack angle, type of ribs and pitch to height ratio. Artificial roughness is a highly effective technique for inducing turbulence and enhancing forced convection heat transfer rates. By strategically placing roughness elements inside ducts and using fans or blowers to create turbulence, it is possible to increase heat transfer without excessive friction losses. The size and configuration of the roughness elements play a crucial role in achieving this balance.

In general, the relative roughness height (e/D), as well as the relative roughness pitch (P/e), are two highly influential parameters, which increase the rate of heat transfer for a roughened SAH. Various shapes, such as ribs, grooves, fins, and chamfers, can be utilised as roughness elements to enhance the heat transfer performance of conventional SAH.

In solar air heaters, improving the roughness of the absorber plate is essential for enhancing thermal efficiency. Artificial roughness techniques are particularly effective in generating turbulence on the absorber plate, resulting in improved heat transfer. While simple roughness methods like grinding tools are less effective, artificial roughness offers superior performance.

In summary, artificial roughness has a significant impact on fluid flow and heat transfer. By using specially designed roughness elements, it is possible to manipulate flow patterns, promote turbulence and enhance heat transfer rates. This technique

finds application in various fields, including solar air heaters, where improving roughness can lead to improved thermal efficiency, as shown in Fig. 4.

6. Impact of various roughness parameters

The solar air heaters can be modified using different roughness parameters. These roughness parameters include operating and control parameters such as height, pitch, angle of inclination, cross-sectional area, Reynold number, mass flow rate and other metrological parameters. These parameters have a significant influence on the thermal performance and thermo-hydraulic performance of SAH. Understanding how these factors affect the system can help optimise its efficiency.

Many researchers evaluated the performance of SAH and developed correlations for heat transfer (Nu) and friction factor (f) using various operating and geometrical parameters, such as Reynolds number, mass flow rate, etc. In a nutshell, heat transfer can be broadly expressed as a function of Reynolds number, relative roughness pitch, relative roughness height, angle of attack (α), different rib shapes and various duct shapes, the number of

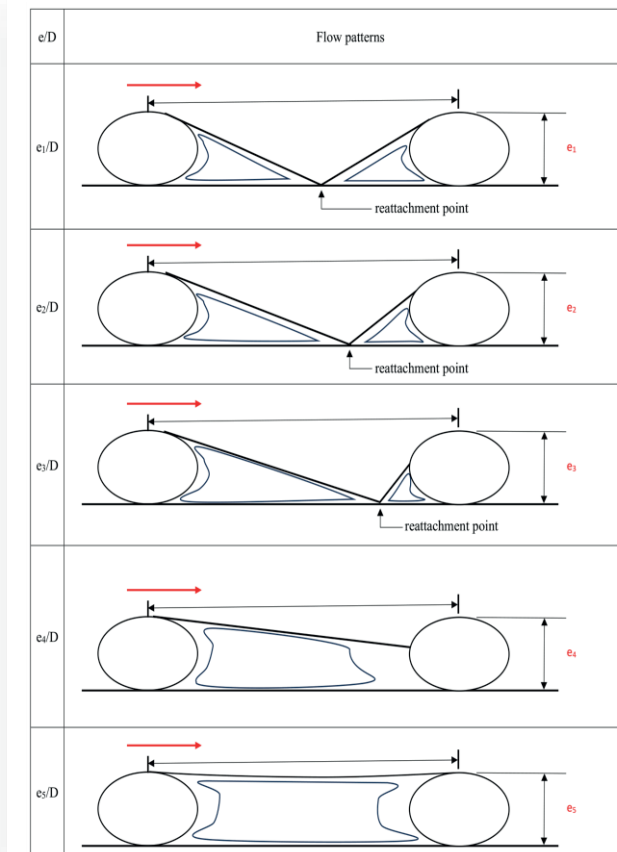


Fig. 5. Effect of relative roughness height on the flow patterns downstream (drawn after [84]).

roughened walls, etc. Furthermore, the same parameters also affect the friction factor.

Understanding and optimising these geometrical factors is essential for achieving enhanced thermal and thermo-hydraulic performance in SAH ducts. By carefully adjusting rib height, rib

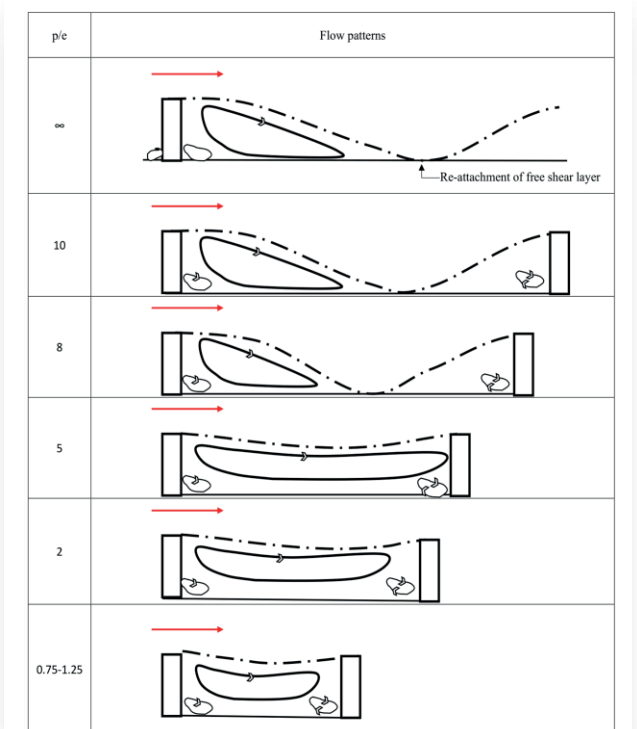


Fig. 6. Effect of relative roughness pitch (P/e) on the flow patterns downstream (drawn after [84]).

pitch, inclination, rib cross-section, and considering the appropriate Reynolds number, it is possible to achieve improved heat transfer efficiency while minimising pressure drop and pumping power requirements.

The geometrical parameters of artificial roughness elements play a crucial role in the performance of solar air heaters. These parameters include the relative roughness height, relative roughness pitch, angle of attack, shape, and arrangement of the roughness elements, etc.

6.1. Impact of relative roughness pitch and relative roughness height

Prasad and Saini [83] modified conventional SAH with circular wire ribs as artificial roughness elements. They used relative roughness height and relative roughness pitch firstly in the field of SAH as geometrical variable. They observed that increasing the relative roughness height leads to an increment in heat transfer and friction factor. For the relative roughness pitch values below 8, the flow reattachment did not occur, which led to a decrease in the rate of heat transfer. However, as the roughness pitch reduced, the friction factor kept increasing. Figure 5 illustrates the flow patterns in the downstream region of circular wire ribs, associated with the changes in relative roughness height and relative roughness pitch. The effect of relative roughness pitch, flow regimes and the development of flow vortices downstream are shown in Fig. 6.

6.2 Impact of angle of attack

Many researchers found the angle of inclination (α) to be the crucial parameter in the SAH performance. In general, when rib

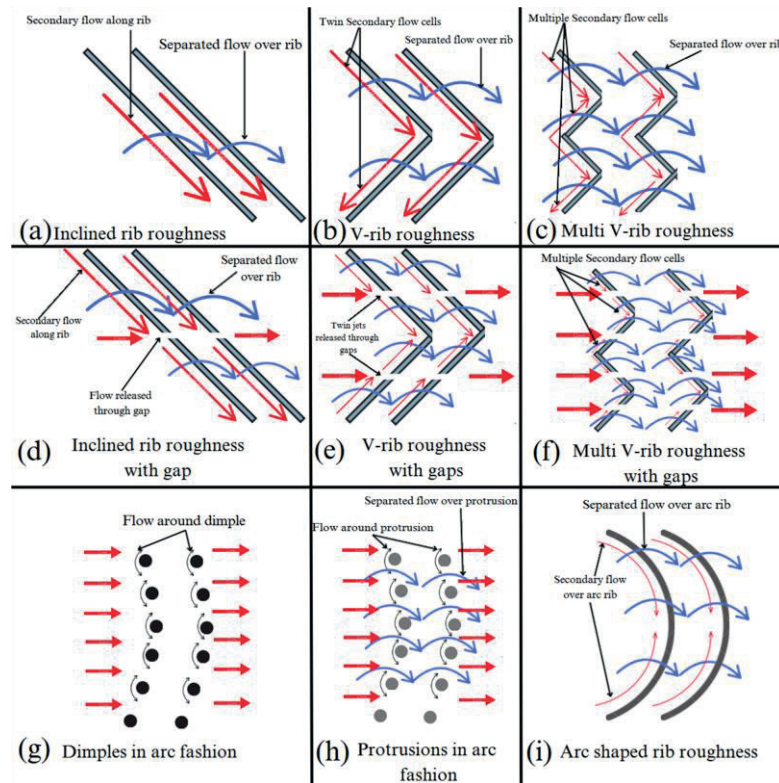


Fig. 7. Flow regime developed by introducing different roughness elements (drawn after [51]).

is inclined, it geometrically divides the main fluid flow into two parts, namely, the primary and the secondary stream flow. This segregation of the flow regime enhances the heat transfer and fluid flow characteristics. Taslim et al. [17] utilised inclined ribs as roughness elements to modify SAH. They observed that when fluid enters the leading edge of the rib and passes through the trailing edge, the whole flow regime acts as a high heat transfer zone. Secondary flow accelerates heat transfer and increases fluid friction.

6.3. Impact of V-shaped rib

Modifying conventional SAH by V-shaped rib roughness leads to creating spanwise counter-rotating flows. The general observation was that V-shaped ribs enhance heat transfer by inducing spanwise counter-rotating flows, which promote vigorous fluid mixing, suppress laminar flow and decrease the thermal resistance (flow obstruction) in the boundary layer, as shown in Fig. 7b.

6.4. Impact of multiple V-shaped rib roughness

Various researchers observed that numerous secondary flow cells are introduced by using multiple V-shaped ribs that interconnect with the main flow. Multiple V-shaped ribs enhance heat transfer by introducing chaotic fluid mixing, which leads to higher heat transfer rates, but also to increased frictional losses, as seen in Fig. 7.

6.5. Impact of gap in inclined, V- and multiple V-shaped rib roughness

V-shaped ribs with gaps enhance heat transfer rate by releasing

secondary flow. The secondary flow accelerates flow inside the boundary layer, which promotes better fluid mixing. The main reason behind the improvement in the heat transfer rate is a twin secondary flow that passes along the ribs. The release of jets through small gaps arranged along the longitudinal axis are shown in Fig. 7.

6.6. Impact of dimples and protrusions

Arc-shaped dimples and protrusions on the absorber plate enhance heat transfer by inducing flow impingement, flow separation and vortex formation. Sethi et al. [85] observed that flow vortices generated on both sides of the protrusion element over the absorber plate and the main flow impingement on the upstream side of the protrusion result in an enhancement of heat transfer in the modified SAH.

7. Various geometries used for solar air heaters performance improvements

The solar air heater (SAH) has been studied in order to improve its performance. The primary goal of these studies was to collect data on improvements in Nusselt number, friction factor, thermo-hydraulic performance and correlation developments. Researchers conducted research using a variety of methods, including not only experimental but also numerical and machine learning approaches. They used various roughness geometries and geometrical configurations of SAH to achieve the aforementioned goal, as summarised in Table 5.

7.1. Transverse ribs

Prasad and Saini [86] conducted an experimental study of SAHs to determine the effect of a protrusion wire (small diameter) over the flat plate (absorber sheet). The objective was to evaluate heat transfer enhancement between the flowing air stream and the heated absorber sheet, as shown in Fig. 8. They considered the design parameters, such as a P/e ratio ranging from 10 to 20, an e/D ratio ranging from 0.020 to 0.033, and Re ranging from 5000 to 50 000. Following the completion of the experimental investigation, they obtained an increase of 2.38 in Nu and an increase of 4.25 in f for a P/e ratio of 10. The researchers reported an increasing effect of the P/e ratio in increasing Nu and f . Additionally, when increasing f , a decrease in Nu was observed while increasing the relative roughness pitch.

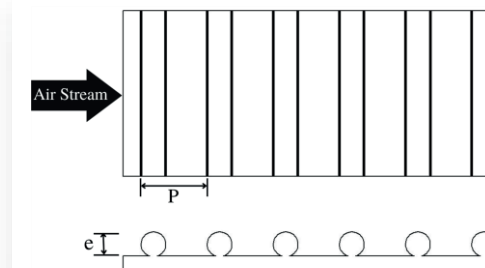


Fig. 8. Transverse wire rib (drawn after [86]).

Gupta [87] conducted an experimental study on a conventional-cum-modified solar air heater with a rectangular duct to investigate the heat transfer mechanism and flow behaviour over the absorber plate. He employed transverse wire ribs as artificial roughness to find the effect of rib roughness on the thermal performance of the modified SAH in the transitional rough region, as depicted in Fig. 9. He used design and flow parameters, such as the aspect ratios of 6.7 to 11.5, P/e ratio of 10, e/D ratio ranging from 0.018 to 0.052, and Re within 3000–18 000. As a result, he reported that a correlation between heat and momentum transfer roughness functions $R(e^+)$ and $G(e^+)$ for a varying range of roughness Reynolds numbers of 5 to 70. He concluded that the correlation between $R(e^+)$ and $G(e^+)$ can be used to design a SAH. He also found a strong correlation between Nu and f , based on theoretical and experimental values.

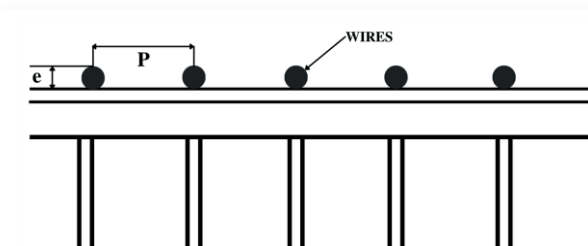


Fig. 9. Transverse wire rib (drawn after [87]).

Karwa [88] performed an experimental study on the rectangular duct in view of the performance improvement of SAH using different types of repeated ribs, as seen in Fig. 10. These ribs

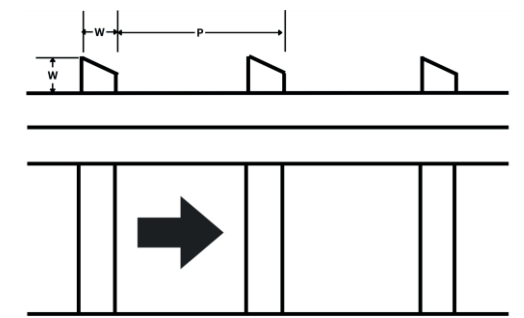


Fig. 10. Transverse wire rib (drawn after [88]).

are transverse, V-discrete, V-continuous, and inclined types of patterns were tested. The parameters involved in the study were a P/e ratio of 10 (fixed), an e/D ratio ranging from 0.0467 to 0.050, an angle of inclination fixed at 60°, and the working parameters such as Re varying between 2800 and 15 000. The author reported that the V-down discrete pattern of augmented absorber plate-based SAH provided the maximum increment in heat transfer under unique conditions. In addition, the author reported significant correlations for Nu and f in the performance evaluation of SAH.

Layek et al. [89] experimentally investigated the repeated transverse chamfered rib-grooved roughened SAH (Fig. 11) to understand heat flow and entropy generation inside the duct. The physical parameters taken included the chamfer angle from 5° to 30°, a relative roughness height of 0.03 (fixed), the relative groove position (g/P), a relative roughness pitch of 10 (fixed) and Reynolds number varying between 3000 and 21 000. The authors found that the Nusselt number and friction factor for the roughened SAH increased 2.6 and 3.35 times, respectively, compared to the smooth SAH at an optimised chamfer angle of 18°. Also, they reported that the maximum thermo-hydraulic performance achieved was 1.76.

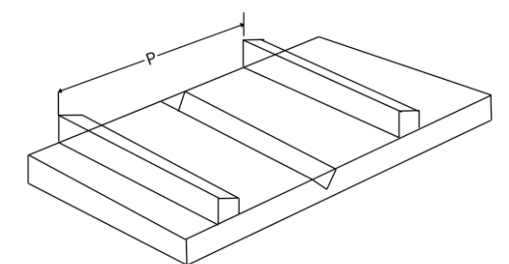
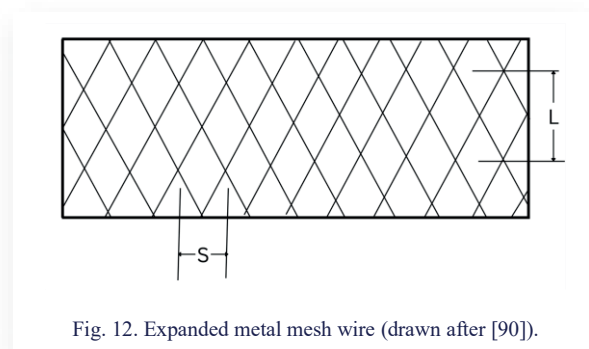


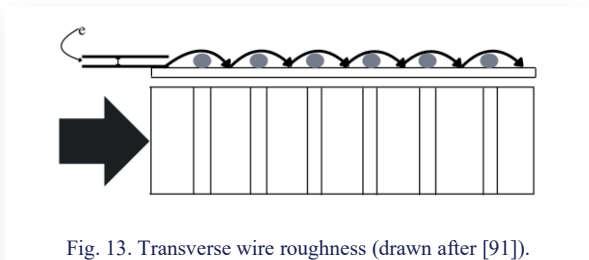
Fig. 11. Transverse wire rib (drawn after [89]).

Saini and Saini [90] conducted an experimental study to see the effect of expanded metal mesh wire based artificially roughened SAH on the heat transfer phenomena, as seen in Fig. 12. They utilised the system parameters such as the relative long-way length of a mesh (L/e) of 25 to 71.87, the relative short-way length of a mesh (S/e) of 15.62 to 46.87 and the relative height of a mesh (e/D) of 0.012 to 0.039, and the Reynolds number ranging from 1900 to 13 000. In the case of the modified SAH with the mesh wire based rib roughness, they obtained a maxi-



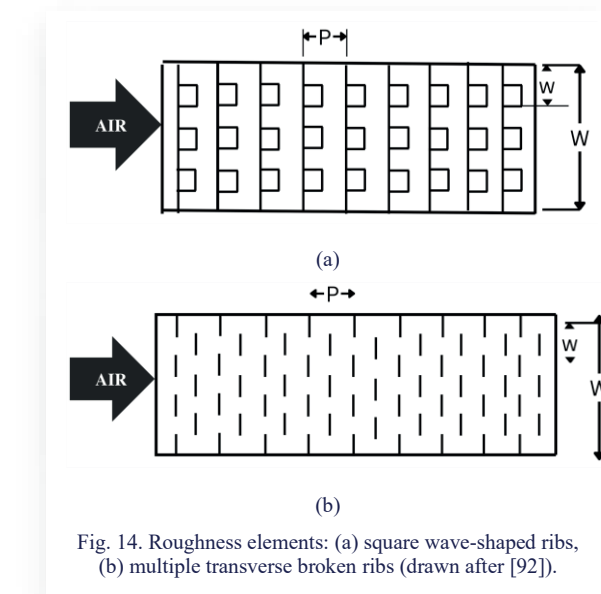
imum increase in Nu and f of 4 and 5, respectively, compared to a smooth SAH. They also reported a strong correlation between Nu and f for the accurate prediction of fluid flow behaviour.

Verma and Prasad [91] performed an experiment to evaluate the impact of transverse wire artificial roughness on the fluid flow characteristics and the rate of heat transfer along with the flow behaviour – the flow mechanism inside the SAH is shown in Fig. 13. They employed the design parameters and working parameters, such as a P/e ratio of 10–40, an e/D ratio ranging from 0.01 to 0.03, and Re ranging from 5000 to 20 000. They also considered the roughness Reynolds number (e^+) for the study. For a roughened duct, they obtained an enhancement in Nu ranging from 1.25 to 2.08 compared to a smooth duct-based SAH, and also reported an optimum thermo-hydraulic performance (THP) of 1.71.



Singh et al. [92] performed experimental and numerical studies of multiple transverse broken and square wave-shaped ribs as roughness elements, depicted in Fig. 14, for evaluation of the performance characteristics of SAH. They utilised design parameters such as $e/D = 0.043$ (fixed), $W/w = 7$ (fixed), $P/e = 10$ (fixed), and the Reynolds number as the working parameter, varied from 3000–18 000 (six levels). They concluded that SAH with multiple broken ribs performed better, achieving 2.38 times higher THP under identical conditions compared to the solar air heater with broken transverse ribs. Furthermore, they found that at Re of 15 000, THP for multiple broken transverse ribs was 2.10 times higher, and for square wave-shaped ribs, 1.62 times higher, compared to smooth duct SAH. Also, they obtained increased values of Nu and f – 2.50 times and 3.92 times, respectively, for multiple transverse broken ribs, and 3.24 times and 3.85 times, respectively, for square wave-shaped ribs.

Singh et al. [93] studied the impact of artificially roughened SAH using square-wave shaped and non-uniform ribs (shown in Fig. 15) to evaluate an enhancement in the thermal performance.



They employed a CFD (computational fluid dynamics) technique to save resources and periodic gaps to minimise eddies just after the ribs. They also used the following design and working parameters: a relative roughness width of 10–310; a relative roughness pitch of 10 (fixed); a relative roughness height of 0.043 (fixed); the Reynolds number ranging from 3000 to 15 000. The authors quoted a maximum augmentation of the Nusselt number of 2.22 and a maximum augmentation of the friction factor of 3.44. They also reported a maximum thermo-hydraulic performance value of 1.49. Finally, they concluded that non-uniform rib-based SAH gives better performance compared to uniform rib-based SAH.

Sharma et al. [94] analysed SAH by employing six distinct roughness geometries, such as transverse ribs with and without an inclination angle, dimples with and without an inclination angle, arc-based rib and sine-wave type rib, using a CFD approach to improve heat transfer and fluid flow characteristics. The design parameters used were $P/e = 10$ (fixed) and $e/D_h = 0.271$ (fixed), and the working parameter was the Reynolds number varying from 3000 to 18 000 to capture the behaviour of SAH exactly. The authors reported maximum enhancement in Nu of 3.60 at the optimum Re of 15 000 and maximum augmentation in f of 15.5 at $Re = 3000$ for sine wave-shaped baffles compared

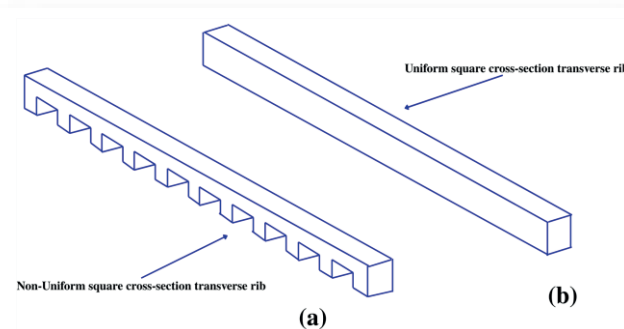


Fig. 15. Roughness elements: (a) non-uniform ribs, (b) uniform ribs (drawn after [93]).

to the smooth case. They also found the maximum THP value for the sine wave-shaped baffles based SAH amounting to 2.05 at the optimal Re of 15 000.

Mahanand and Senapati [95] performed a CFD analysis of a SAH for the prediction of heat transfer and fluid flow characteristics using transverse inverted-T shaped ribs shown in Fig. 16 as roughness elements. As control parameters, they used an e/D ratio of 0.042 (fixed), a P/e ratio varying from 7.14 to 17.86, and Re varied from 3800 to 18 000 over a two-dimensional (2D) fluid flow SAH domain. Additionally, boundary conditions were implemented utilising an aluminium absorber plate along with the RNG (renormalization group theory) $k-\epsilon$ model with an enhanced wall treatment for capturing a real-time view inside SAH. The authors obtained a maximum augmentation of Nu for SAH with an absorber plate roughened with the inverted-T ribs by a factor of 2.747 compared to a smooth plate SAH. The optimum design parameters for improved Nu were $P/e = 7.14$ and $Re = 15 000$. Also, for P/e of 7.14 and Re of 3800, the maximum f achieved was 3.404 times that of a smooth duct. They also reported a THP of 1.87 for an inverted T-shaped ribbed SAH, under the following operating conditions: P/e ratio of 7.14 and e/D_h ratio of 0.042 at $Re = 15 000$. They also found sound correlations between Nu and f .

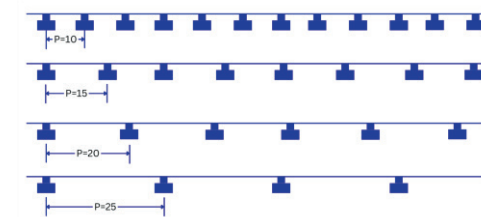


Fig. 16. Roughness elements in the form of transverse inverted-T shaped ribs (drawn after [95]).

Gawande et al. [96] studied two novel roughness geometries, such as combined square and equilateral triangular ribs over a flat absorber plate (Fig. 17) for thermal and thermo-hydraulic performance enhancement, using both experimental and CFD techniques. The design and flow parameters utilised were: a P/e ratio varying from 7.14 to 17.86, an e/D ratio of 0.042 (fixed) and Re varying from 3800 to 18 000. In this study, a constant heat flux of 1000 W/m^2 and the RNG $k-\epsilon$ turbulence model were employed to minimise the deviation between the experimental and CFD results. The highest increments in Nu and f reported were 2.877 and 3.849 for a P/e ratio of 7.14, an e/D ratio of 0.042

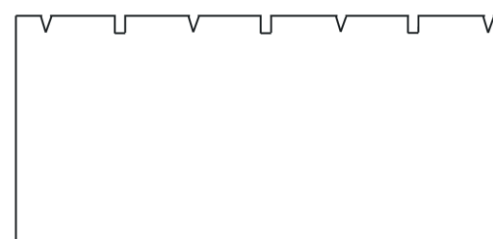


Fig. 17. Roughness geometry of combined square and equilateral triangular ribs (drawn after [96]).

and Re of 18 000. Also, a THP of 1.86 was obtained at the optimum operating parameters, such as $P/e = 7.14$, $e/D = 0.042$ and $Re = 18 000$. It was observed that combined ribs give better SAH performance compared to individual ribs.

Kumar et al. [97] studied experimentally the double-pass SAH consisting of a flat plate kind absorber to enhance thermal performance compared to conventional SAH, as depicted in Fig. 18. They employed asymmetric semi-circular and symmetric circular shapes as roughness over the absorber plate. They found that asymmetric semi-circular based roughness performed better in SAH compared to a symmetrical shape based SAH. Moreover, novel correlations were developed for Nu and f as a function of Reynolds number and relative roughness height. This correlation showed strong prediction capability.

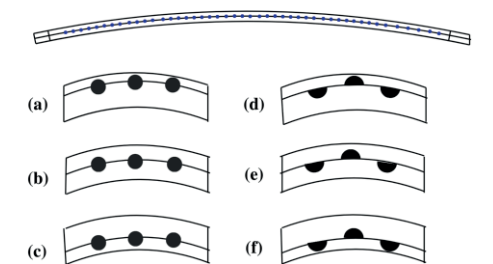


Fig. 18. Symmetric circular (a – c) and asymmetric semi-circular (d – f) shapes as roughness (drawn after [97]).

Kumar and Layek [98] combined two techniques to emphasise the rate of heat transfer and fluid flow characteristics of a modified SAH with roughness geometry of circular ribs over an absorber plate, using liquid crystal thermography (LCT) and CFD technique. The roughness geometry is represented in Fig. 19. The design parameters involved were: a P/e ratio taken as 10 and 20, a constant e/D_h ratio and Re varying from 8551 to 11 149. Liquid crystal thermography produced graphical results of Nu and f . The maximum enhancement in the Nusselt number obtained through the combined approach was 1.97 compared to the smooth duct.

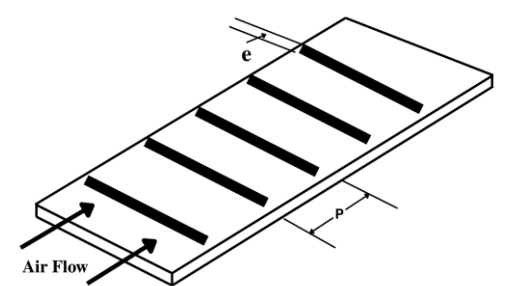
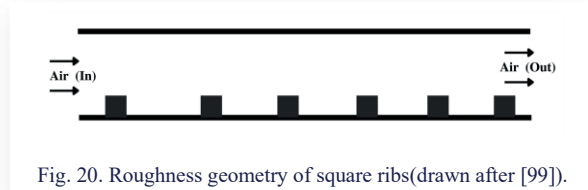


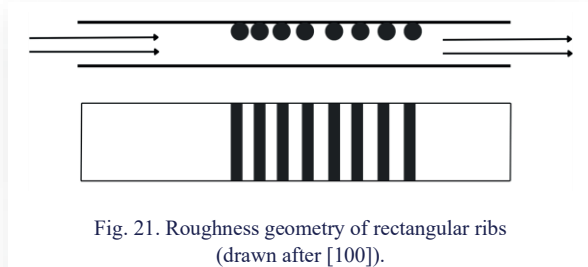
Fig. 19. Roughness geometry of circular ribs (drawn after [98]).

Kumar et al. [99] modified SAH with square ribs and also modified the cross section of the duct to capture an impact on thermal and thermo-hydraulic performance of the SAH compared to a conventional SAH, as illustrated in Fig. 20. Two sets

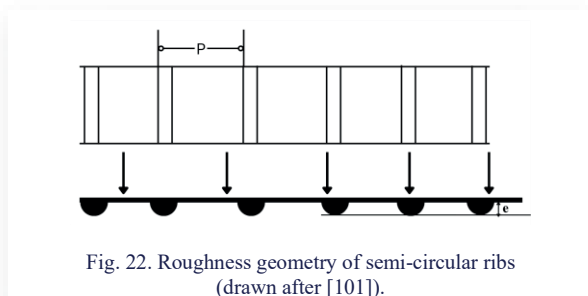


of design parameters were tested by the CFD technique, i.e. the P/e ratio ranging from 5 to 13 and the e/D ratio ranging from 0.013 to 0.05. The authors also mentioned Re varying from 3900 to 17 900. They found that the $k-\epsilon$ based turbulent model performed significantly better than other methods tested. The enhancement in performance evaluated after modification showed that Nu increased by 1.97 times and f increased by 2.53 times compared to the smooth duct. The highest THP calculated was 1.44 at the optimum parameters of $P/e = 10$ and $e/D = 0.05$ at Re of 17 900. Their study revealed a strong correlation for Nu and f , with a maximum deviation of about 15%.

Kumar et al. [100] studied the mechanism of heat transfer and fluid flow phenomena of SAH using rectangular ribs over the absorber plate, as depicted in Fig. 21. The duct was triangular in cross-section, created for CFD evaluations. The physical design parameters employed for the study were the aspect ratio (e/w) in the range of 0.25 to 4.0, e/D ratio in the range of 0.02 to 0.04, P/e ratio in the range of 5 to 15; Re varied between 4000 and 18 000. The authors reported a maximum THP of 1.89 at optimum parameters: a P/e ratio of 10, an e/w ratio of 4.0, an e/D ratio of 0.04 and Re of 15 000.



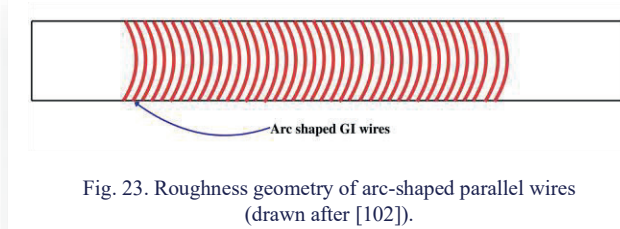
Kumar et al. [101] evaluated the performance of SAH with three novel models of SAH and roughness over the absorber plate. The roughness geometry is shown in Fig. 22. The design parameters used were the apex angle of 60, number of rounded corners 0 to 1, a curvature radius of 0 to 0.393 H (where H is the duct height) and operating parameter Re varied from 4000 to 18 000. The minimum enhancement in the performance of SAH



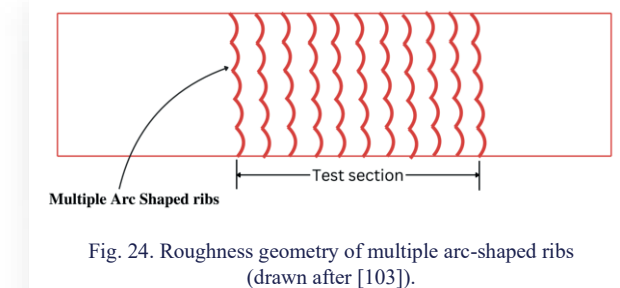
reported was 1.87 times that of the conventional model with no roughness element over the absorber plate.

7.2. Arc-shaped rib roughness

Saini and Saini [102] performed an experimental investigation on modified SAH for performance enhancement. The modification of SAH was carried out using an arc-shape parallel wire as the roughness element (Fig. 23). The design and operating parameters considered in the study were P/e ratio of 10 (fixed), a relative roughness height (e/D) ranging from 0.0213 to 0.0422, a relative angle of attack ($\alpha/90^\circ$) varying between 0.3333 and 0.6666, and Re ranging from 2000 to 17 000. It was found that, compared to the smooth duct of SAH, Nu increased by 3.80 times and f increased by 1.75 times. Based on the experimental datasheet of Nu and f , strong correlations were developed with good accuracy.

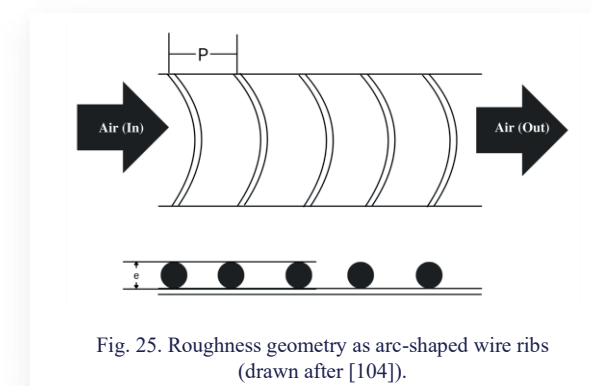


Singh et al. [103] used multiple arc-shaped ribs as roughness elements for the modification of SAH (Fig. 24) to improve its thermal performance. The parameters of practical and operating conditions, such as Re ranging from 2200 to 22 000, e/D ratio ranging from 0.018 to 0.045, W/w ratio ranging from 1 to 7, P/e ratio varied between 4 and 16, and arc angle (α) within the range of 30° to 75° . They reported the maximum increase in Nu and f of 5.07 and 3.71, respectively, compared to the smooth duct. The optimum parameters they obtained were: Re of 22 300, W/w ratio of 5, e/D ratio of 0.045, P/e ratio of 8 and α of 60° . The maximum THP reported at optimum parameters was 3.4.

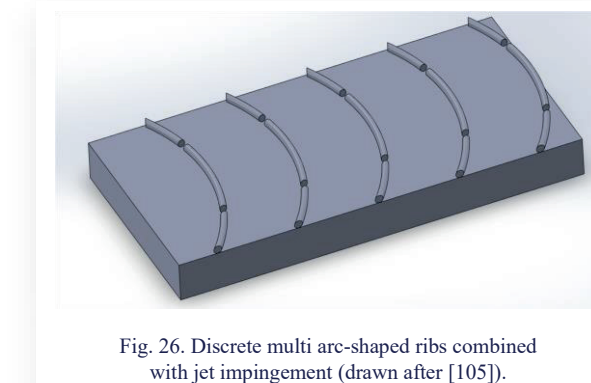


Sahu and Prasad [104] investigated a modified SAH for exergetic performance enhancement using roughness geometry in the form of arc-shaped wire ribs (Fig. 25), using an analytical approach. The mathematical model was developed to validate the results. The maximum exergetic performance improvement compared to a smooth duct was found to be 56%.

Kumar et al. [105] modified the SAH with a novel roughness geometry consisting of discrete multi arc-shaped ribs combined

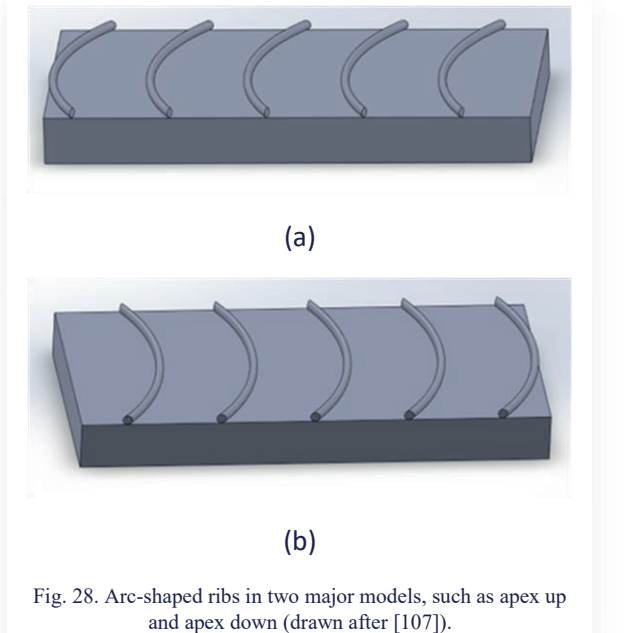
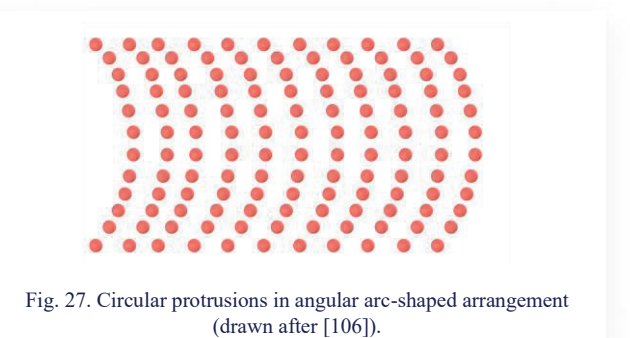


with jet impingement for an enhancement in heat transfer and fluid flow characteristics, as seen in Fig. 26. The design and operating parameters utilised were: the relative discrete distance of 0.27–0.86, the relative discrete width of 0.32–1.72 (varying), the relative rib height of 0.025–0.047 (varying), the relative rib pitch of 0.58–3.1 (varying), the arc angle of 35° – 65° (varying), the stream wise pitch ratio of 0.47–1.72 (varying), the spanwise pitch ratio of 0.46–0.82 (varying) and the Reynolds number varying from 3000 to 19 000. They reported an increment in Nu and f for a modified SAH of 7.61 and 6.48, respectively, as compared to the smooth SAH. In addition, the THP for optimised design parameters was 4.1. Correlations were developed for Nu and f to predict thermal and fluid characteristics inside the SAH.



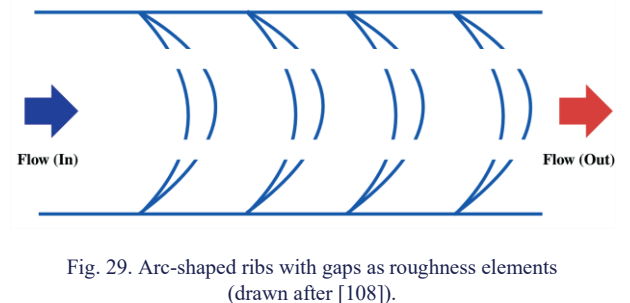
Yadav et al. [106] studied experimentally the effect of heat transfer and fluid flow mechanism of a modified SAH, incorporating circular protrusions in angular arc-shaped arrangement as roughness elements, as seen in Fig. 27. The physical and operating variables used included: a P/e ratio varying from 12 to 24, an e/D ratio varying from 0.015 to 0.03, the arc angle of protrusions arrangement varying from 45° to 75° and Re ranged from 3600 to 18 100. The maximum augmentation of Nu and f was 2.89 and 2.93 times, respectively, in comparison to a smooth SAH. The authors also developed correlations for Nu and f .

Ghritlahre [107] performed an experimental study for augmentation of heat transfer and fluid flow characteristics by modifying conventional SAH with a novel arc-shaped artificial roughness in two major configurations, such as apex up and apex down, as shown in Fig. 28. They employed the design parameters as a P/e ratio of 10 (fixed), an e/D ratio of 0.0395 (fixed), a rib roughness of 2.5, and an arc angle of 60° . The operating



parameter they used was the mass flow rate, ranging from 0.007 to 0.022 kg/s. They observed the maximum enhancement of 1.42 in heat transfer compared to smooth SAH in the apex upstream flow.

Srivastava et al. [108] performed CFD analysis of modified SAH, using artificial roughness in the form of dissimilar repeated ribs. These ribs were arc-shaped with a gap and V-shaped with a gap (Fig. 29) for the enhancement of thermal and thermo-hydraulic performance of the SAH. The boundary conditions were set as: a g/e ratio of 1, an e/D ratio of 0.04, a P/e ratio of 10, a p/P ratio of 0.65, a relative gap distance (d/W) of 0.69, a number of gaps – 1, an arc angle of 60° , a constant heat flux of 1000 W/m^2 , and Re ranging from 3000 to 15 000. The Nu and f for arc-shaped ribs with gaps were increased by 2.4 and



3.94 times, respectively, at $Re = 3000$. The maximum THP values reported were 2.21 and 1.5 for V-shaped ribs and arc-shaped ribs, respectively.

Gill et al. [109] conducted experimental studies on modified SAH using broken arc ribs combined with staggered rib for enhancement of thermal and thermo-hydraulic performance of the SAH, as seen in Fig. 30. They utilised the following design parameters: a fixed relative staggered rib position of 0.4, an e/D ratio of 0.043, a relative gap size of 0.65, an arc angle of 60° , and a P/e ratio of 10 (fixed). They used Re ranging from 2000 to 16 000 as the operating variable. The maximum augmentation of Nu and f was 3.06 and 2.50, respectively, compared to the smooth duct. They also reported the highest THP of 2.27 in the whole study.

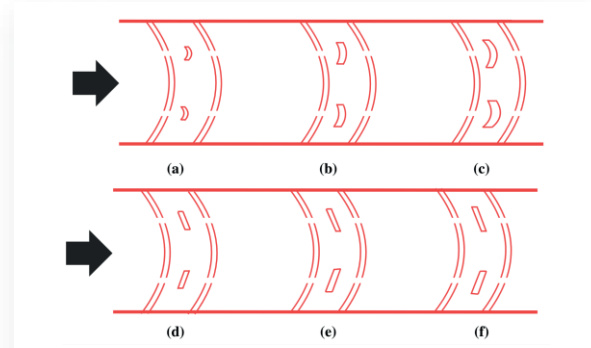


Fig. 30. Roughness in the form of the broken arc ribs combined with staggered rib (drawn after [109]).

Bhuvad et al. [110] performed an experimental study on the enhancement of heat transfer and fluid flow characteristics of modified SAH using roughness elements as novel apex-up discrete arc rib, as depicted in Fig. 31. They modified SAH by augmenting parameters, such as the angle of attack (α) in three-step (30° , 40° , 60°), the g/e ratio of 4 (fixed), the e/D ratio of 0.045 (fixed), P/e set to 10, and the number of gaps set to 3. They used Re ranging from 3000 to 14 000 as the operating parameter. The authors obtained the maximum increase in Nu and f of 2.92 and 3.04 times, respectively, compared to smooth SAH. The maximum THP found was 2.01 across all cases of studied SAH. The correlations for Nu and f were developed for the prediction of similar cases in SAH.

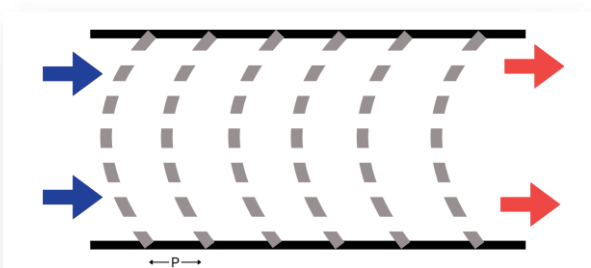


Fig. 31. Apex-up discrete arc rib as a roughness element (drawn after [110]).

Korpale et al. [111] developed a modified SAH using rectangular section ribs, shown in Fig. 32, to enhance the heat transfer capability of SAH. They employed the following design parameters: a P/e ratio range of 5–60, an e/D ratio in the range of

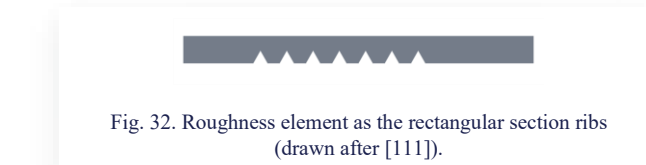


Fig. 32. Roughness element as the rectangular section ribs (drawn after [111]).

0.065–0.252, a w/W ratio in the range of 0.5–10 and Re varying from 4000–20 000. The maximum THP found for modified SAH was 2.77. They also validated the experimental results with CFD results for optimisation.

Kumar et al. [112] developed SAH with a novel geometry consisting of multiple-arc shaped ribs with gaps (Fig. 33), designed for better heat transfer and fluid flow under the absorber plate of a new modified SAH. The parameters they used for the study were: a number of gaps of 3, a relative gap width of 0.5–1.5, a relative gap position of 0.3–0.9, and varying Reynolds number of 4000–20 000. They reported an enhancement of the Nusselt number and an increase in friction factor, with deviation of $\pm 6.15\%$ and $\pm 5.74\%$, respectively, compared to the smooth-duct-based SAH. Additionally, the maximum THP obtained for the modified SAH was 2.77 at optimum parameters. These were: a relative pitch of 17.22, a relative rib height of 0.044, a relative gap width 0.5 and the Reynolds number of 20 000.

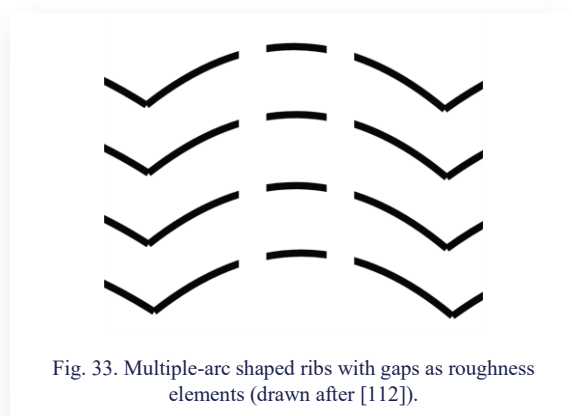


Fig. 33. Multiple-arc shaped ribs with gaps as roughness elements (drawn after [112]).

7.3. V-shaped ribs

Singh et al. [113] experimentally investigated SAH with discrete V-shaped ribs as artificial roughness, with the view to heat trans-

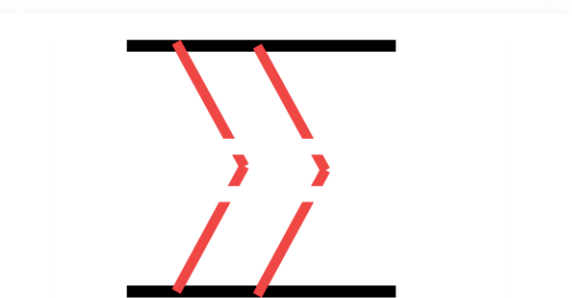


Fig. 34. Roughness elements in the form of discrete V-shaped ribs (drawn after [113]).

fer and fluid flow enhancement (Fig. 34). They used various design parameters, such as a P/e ratio ranging from 4 to 12, an e/D ratio ranging from 0.015 to 0.043, α of 30° to 75° , a g/e ratio varying between 0.5 and 2.0, and a relative gap position (d/w) ranging from 0.20 to 0.80. They also used Re varying from 3000 to 15 000 as an operating parameter. The authors found the maximum augmentation in Nu and f of 3.04 and 3.11, respectively, as compared to the smooth duct. They also reported the optimal parameters to achieve the highest Nu and f using the discrete V-shaped rib roughness, which were: a d/w ratio of 0.65, a g/e ratio of 1.0, a P/e of 8.0, α of 60° and an e/D ratio of 0.043. They also developed the correlations for Nu and f in terms of Re and other rib variables.

Momin et al. [114] reported an experimental analysis of a SAH modified with V-shaped rib roughness (Fig. 35) for evaluating an enhancement of heat transfer and fluid flow behaviour. The design and operating parameters used by the authors were: an e/D ratio of 0.02 to 0.034, α of 30° to 90° (fixed), a P/e ratio of 10, and varying Re ranging from 2500 to 18 000. The reported maximum increments across all configurations of modified SAH were: an enhancement in Nu by 2.30 times and an increase in f by 2.83 times compared to the smooth SAH. They also developed correlations for Nu and f , with the conclusion that Re has more impact on the performance than α .

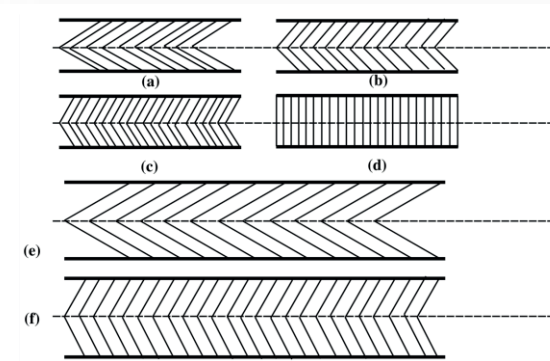


Fig. 35. Roughness element formed by the V-shaped rib (drawn after [114]).

Hans et al. [25] performed an experimental investigation on a SAH, modifying it with an artificial roughness in the form of different types of multiple V-shaped ribs for amelioration in heat transfer and fluid flow behaviour in the SAH. They utilised the design parameters, such as an e/D ratio varying from 0.019 to 0.043, a P/e ratio varying from 6 to 12, α varying from 30° to 75° , a W/w ratio varying from 1 to 10 and Re ranging from 2000 to 20 000 as operating parameter. By modification, they obtained the highest increase in Nu and f – of 6 and 5 times, respectively – compared to a smooth duct. They also developed a statistically based correlation for Nu and f for predicting the SAH performance.

Deo et al. [115] conducted an experimental study on modification of SAH by using artificial roughness in the form of multi-gap V-down ribs combined with staggered ribs for performance enhancement of conventional SAH, as portrayed in Fig. 36. The variables they used for design and operation were: a P/e ratio of 4–14, an e/D ratio of 0.026–0.057, α of 40° – 80° , a g/e ratio of

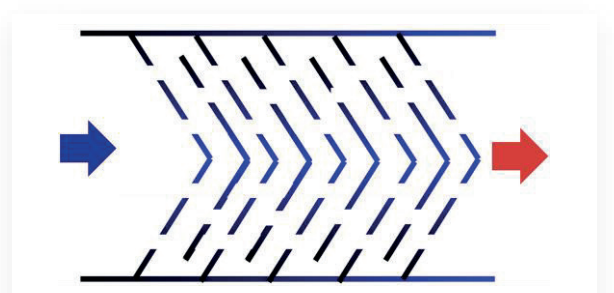


Fig. 36. Roughness elements in the form of multi-gap V-down ribs combined with staggered ribs (drawn after [115]).

1 (gap width to rib height as fixed), a w/e ratio of 4.5 (staggered rib length to rib height ratio as fixed), a p/P ratio of 0.65 (relative staggered rib pitch as fixed), a N_p of 2 (number of gaps), and Re in the range of 4000 to 12 000. The obtained maximum augmentation in Nu was 3.34 and the maximum THP found was 2.45.

Singh et al. [116] developed an experimental setup of modified SAH with V-down ribs with a gap as roughness elements to examine an enhancement of thermal and thermo-hydraulic performance of conventional SAH, as shown in Fig. 37. They utilised design parameters and operating parameters, including an angle of attack of 60° (fixed), a relative roughness height of 0.043 (fixed), a duct aspect ratio of 12, a relative roughness pitch of 4–12 (five step) and the Reynolds number varying from 3000 to 12 000. For the modified SAH, they reported the maximum augmentation in the Nusselt number and friction factor of 2.70 and 2.86, respectively. They also found that the highest THP for the modified SAH was 1.93.

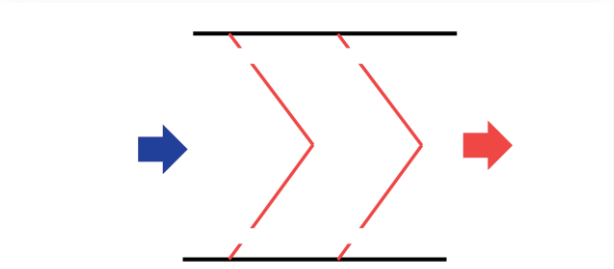


Fig. 37. V-down ribs with a gap as roughness elements (drawn after [116]).

Kumar et al. [117] accomplished an experimental study on the modified SAH, using artificial roughness in the form of multiple V-shaped ribs with gaps to improve heat transfer and fluid flow behaviour of a SAH. They used the design and run parameters, such as a P/e ratio of 10 (fixed), an e/D ratio of 0.043 (fixed), a W/w ratio of 6 (fixed), a relative gap distance (G_d/L_v) within a range of 0.24 to 0.80, a g/e ratio ranging from 0.5 to 1.5, an α of 60° (fixed), and Re varying between 2000 to 20 000. They observed the maximum increase in Nu and f of 6.32 and 6.12, respectively, for modified SAH as compared to a smooth duct. They also observed an improvement in THP of 3.6 for the modified duct compared to conventional smooth SAH.

Patel and Lanjewar [118] developed the modified SAH using artificial roughness as a V-geometry with symmetrical gap and staggered elements, as depicted in Fig. 38. To achieve the objec-

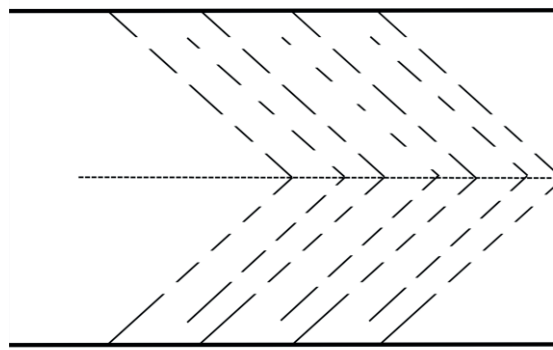


Fig. 38. Roughness formed by a V-geometry with symmetrical gap and staggered elements (drawn after [118]).

tive they employed the governing design parameters, such as a duct aspect ratio (W/H) of 12 (fixed), a P/e ratio of 10, an e/D_h ratio of 0.043, an α of 60° , a ratio of staggered rib position to pitch (P'/P) of 0.4 (fixed), a ratio of gap width to rib height (g/e) of 4, a number of main gaps (N_g) equal to 4, a ratio of staggered size to rib height (r/e) of 4, a relative gap size of additional gap in the symmetrical rib elements (g'/e) in the range of 0.5 to 2, the relative gap position of additional gap in the symmetrical rib elements (d/w) in the range of 0.25 to 0.85, and Re varying between 4000 to 15 000. The maximum obtained enhancement in Nu and f for modified SAH was 2.51 and 2.70, respectively, compared to the smooth SAH. The maximum THP found was 1.82 for the optimum variables, such as Re of 12 524, a g'/e ratio of 1 and a d/w ratio of 0.65.

Srivastava et al. [108] performed a CFD analysis of the SAH modified using dissimilar repeated ribs as artificial roughness to enhance the SAH's thermal and thermo-hydraulic performance. The ribs were of V-shape with a gap, as illustrated in Fig. 39. The boundary conditions used were: a g/e ratio of 1, an e/D ratio of 0.04, a P/e ratio of 10, a p/P ratio of 0.65, a d/W ratio of 0.69, a number of gaps of 1, arc angle of 60° , constant heat flux of 1000 W/m^2 , and Re ranging from 3000 to 15 000. The maximum enhancement in Nu and f they obtained for V-shaped ribs was 2.4 and 3.4, respectively, at Re = 15 000. The maximum THP values reported were 2.21 and 1.5 for V-shaped ribs and arc-shaped ribs, respectively.

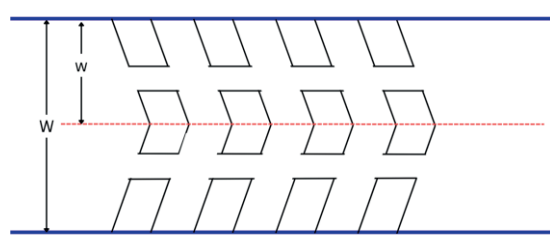


Fig. 39. Roughness element in the form of a V-shaped rib with a gap (drawn after [108]).

7.4. Dimple or protrusion ribs

Kumar and Prasad [119] extensively conducted an experimental study on a modified SAH roughened with concave dimples for better performance of conventional SAH. They compared a 3-sided roughened SAH with a 1-sided conventional SAH. They utilised various design and operating parameters, such as a P/e ratio in the range of 8 to 15, an e/D ratio in the range of 0.018 to 0.045, an e/d ratio in the range of 1 to 2, and Re varied from 2500 to 13 500. They reported that Nu and f were affected by roughness parameters. The maximum increase in Nu and f across all cases investigated was 3.94 and 2.79.

Saini and Verma [120], aimed at improving the performance of SAH, performed an experimental investigation artificially roughened SAH with dimple shape roughness elements, as shown in Fig. 40. The employed design and operating parameters were: a relative roughness height (e/D) in the range of 0.018–0.037, a relative pitch (p/e) in the range of 8–12, and the Reynolds number ranging within 2000–12 000. They observed that, under all studied cases, the maximum Nu and f were 7.58 and 4.68, respectively, compared to a smooth duct. They also developed correlations for Nu and f for the performance prediction of SAH.

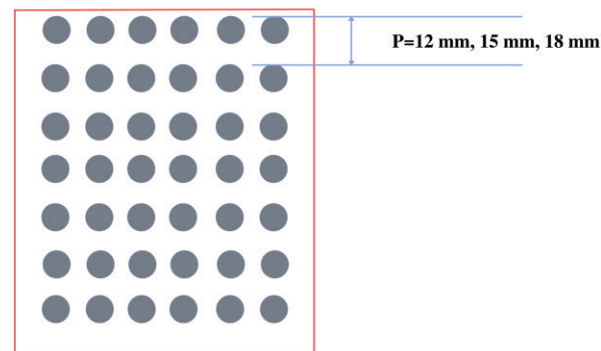


Fig. 40. Dimple-shaped roughness elements (drawn after [120]).

Kumar and Verma [121] performed extensive experimental, as well as numerical investigation, on sinusoidal-based SAH for performance enhancement. They employed the design and operating parameters, such as a hydraulic diameter (D_h) of 33.33 mm, a roughness pitch (P) of 10–18 mm (in three steps), a roughness height (e) of 1 mm, an aspect ratio (W/H) of 5, and the Reynolds number varied from 4000 to 15 000. Compared to a smooth duct, they obtained maximum augmentation in Nu and f of 2.48 and 1.76, respectively. The highest calculated THP was 2.05 at the optimum parameters of $P = 10 \text{ mm}$ and Re = 4000. The authors also concluded that THP obtained from experimental analysis deviated by 6.77% compared to the CFD results.

Bhusan and Singh [122] developed a modified SAH with protrusion roughness for an enhancement of SAH performance, as illustrated in Fig. 41. The employed various design and operating parameters included: a relative short way length (S/e) in the range of 18.75 to 37.50, a relative long way length (L/e) in the range of 25.00 to 37.50, a relative print diameter (d/D) ranging from 0.147 to 0.367, a relative roughness height (e/D) of

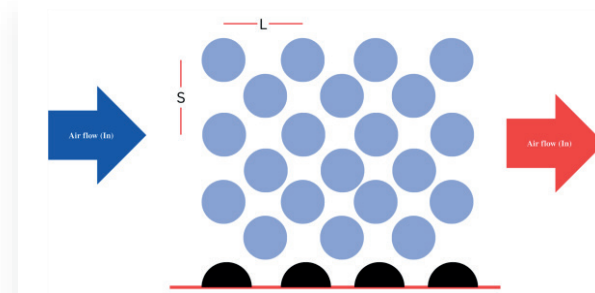


Fig. 41. Protrusions as roughness elements (drawn after [122]).

0.03 (fixed), and the Reynolds number ranging between 4000 and 20 000 during experimentation. The maximum Nu and f obtained were 3.8 and 2.2, respectively, compared to a smooth duct. The maximum performance was evaluated at the optimum parameters, as the S/e ratio of 31.25, L/e ratio of 31.25 and d/D ratio of 0.294. They also developed correlations for Nu and f .

Hassan et al. [123] performed an experimental investigation on modified SAH by using artificial roughness as dimple roughened in multiple arc pattern in the view of performance enhancement as seen in Fig. 42. The employed design and operating parameters included a relative roughness pitch (p/e) in the range of 4 to 16, a relative roughness height (e/D_h) in the range of 0.018 to 0.045, a relative roughness width (W/w) in range of 1 to 5, an angle of attack in the range of 30° to 75° , and the Reynolds number of 2000 – 18 000 as governing parameters for the modified SAH. The maximum increase in the Nusselt number and friction factor was found to be 5.61 and 2.27 times, respectively, compared to a smooth duct. The optimum parameters in that case were: Re of 17 984, a W/w ratio of 4, an e/D_h ratio of 0.036, a P/e ratio of 12 and α of 60° .

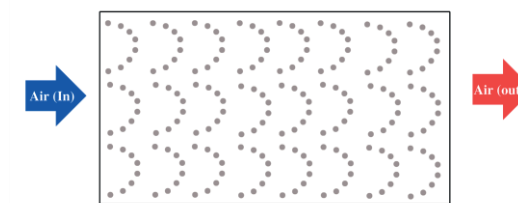


Fig. 42. Roughness elements in the form of dimples in multiple arc pattern (drawn after [123]).

Kumar [9] performed an experimental study on 1- and 3-sided concave dimple-roughened modified SAH (Fig. 43) aimed at improvement of thermal and thermo-hydraulic performance. The author employed the roughness design and run parameters, such as a relative dimple pitch in the range of 8 to 15, a relative dimple height of 0.018 – 0.045, a relative dimple depth in the range of 1 to 2, and a Reynolds number in the range of 2000 to 13 500. The maximum Nu and f were found to be 3.94 times and 2.79 times that of a smooth duct. Additionally, statistical correlations were developed for Nusselt number and friction factor. The percentage deviation between experimental and data collected through correlations was within the range of 5–6%.

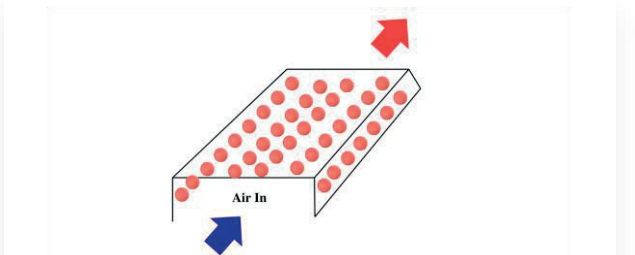


Fig. 43. Concave dimples as roughness elements (drawn after [9]).

Alam and Kim [124] fabricated a modified SAH using conical protrusion ribs as roughness elements for gaining higher thermal and thermo-hydraulic performance, as seen in Fig. 44. The design parameters employed were: a relative rib pitch (P/e) ranging between 6 and 12, a relative rib height (e/D) ranging from 0.200 to 0.044, and a Reynolds number of 4000 to 16 000. The authors developed correlations for Nu and f . They found maximum augmentation in Nu and f of 2.49 and 7.29, respectively, compared to a smooth SAH.

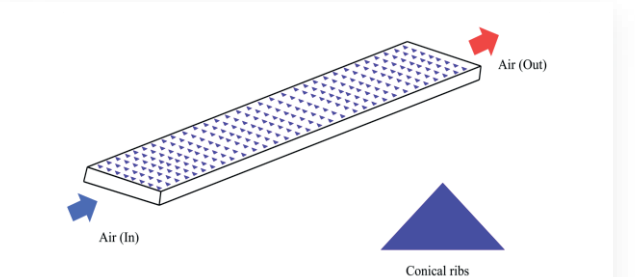


Fig. 44. Conical protrusion rib as a roughness element (drawn after [124]).

7.5. W-shaped ribs

Lanjewar et al. [125] developed modified SAH using W-shaped ribs with inclination as artificial roughness to optimise the performance of a conventional SAH. There were two major models tested with respect to flow as downstream W-down and upstream W-up, as shown in Fig. 45. They utilised various design parameters to evaluate proposed performance enhancement, such as a duct aspect ratio (W/H) of 8, a P/e ratio of 10 (fixed), an e/D of 0.03375 (fixed), α of 30° to 75° , and used Reynolds number varying from 2300 to 14 000 as a run parameter. The

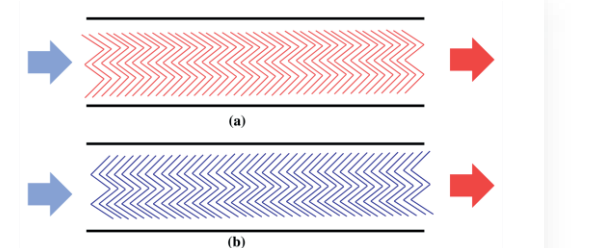


Fig. 45. Downstream W-down (a) and upstream W-up (b) roughness elements (drawn after [125]).

maximum increase in Nu and f obtained in the study was 2.36 and 2.01, respectively, for W-down ribs at the optimum parameters. These parameters were $\alpha = 60^\circ$, $e/D = 0.03375$ and $Re = 14\,000$. They also developed correlations for Nu and f , which were acceptable within the range of 5 to 10% in any case.

Kumar et al. [126] conducted an experimental analysis on modified SAH using discrete W-shaped elements as artificial roughness for performance enhancement of conventional SAH, as illustrated in Fig. 46. They employed various design parameters, such as the duct aspect ratio of 8, Reynolds number varied from 3000 to 15 000, relative roughness height of 0.0168 to 0.0338, relative roughness pitch of 10 and the angle of attack (α) varied from 30° to 75° . They found that the modified SAH has the maximum increase of the Nusselt number and friction factor of 2.16 and 2.75 times, respectively, for $\alpha = 60^\circ$. They also developed correlations for Nu and f for SAH performance prediction.

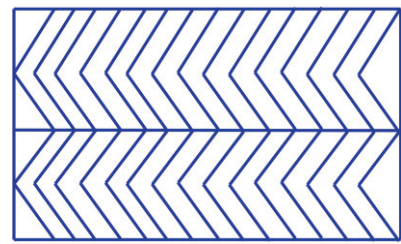


Fig. 46. Discrete W-shaped ribs as roughness elements (drawn after [126]).

Thakur and Thakur [127,128] developed modified SAH using multi-staggered rib parameters of the W-rib roughness as novel roughness geometry for the performance enhancement of conventional SAH, as illustrated in Fig. 47. The employed design and operating parameters included: a duct aspect ratio (W/H) of 10 (fixed value), an approach angle (α) 60° (as fixed value), a Reynolds number (Re) in range of 4000–14 000, a relative roughness height (e/D_h) 0.0423 (as fixed value), relative roughness pitch (P/e) in the range of 10 to 14, number of gaps (N_g) of 1 to 2, relative gap width (g/e) in the range of 0.5 to 2.0, relative staggered rib length (w/e) in the range of 1.5 to 6.0 and the relative staggered pitch (p/P) in the range of 0.45 to 0.75. They obtained the maximum Nu and f of 4.10 and 2.4, respectively, at the optimum parameters, which were: $g/e = 1$, $N_g = 2$, $Re = 12\,000$, $p/P = 0.65$, $P/e = 12$ and $w/e = 4.5$. The calculated THP for the studied case was 3.24.

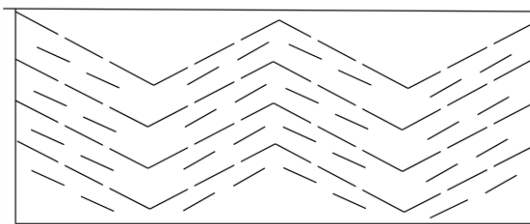


Fig. 47. Roughness in the form of W-shaped ribs with symmetrical gaps with staggered ribs patterns (drawn after [127,128]).

7.6. Spherical ball roughness

Murmu et al. [129] experimentally investigated the effects of artificial roughness on heat transfer and fluid flow characteristics of modified SAH using the spherical ball as roughness elements. The operating and design parameters they employed were: a relative roughness pitch (P/e) varied from 9 to 18, a relative roughness height (e/D_h) ranging from 0.024 to 0.040, a ball's height-diameter ratio (e/d_b) varied from 0.5 to 2, an angle of attack (α) ranging from 35° to 75° and a Reynolds number varied from 2500 to 18 500. In their study, the maximum increase in Nu and f were 3.56 and 2.67, respectively, compared with the smooth duct-based SAH. They also developed a correlation to predict the performance of SAH in terms of Nu and f .

7.7. Non-uniform ribs

Kumar et al. [100] performed a numerical study on a ribbed solar air heater to enhance heat transfer and thermo-hydraulic performance. Forward chamfered rib was in centre of their investigation, as shown in Fig. 48. They performed the analysis using Ansys CFD for the geometrical configuration and variables, such as a relative roughness height (e/D) of 0.018–0.043, a rib chamfered height ratio (e'/e) of 0 to 1, a rib aspect ratio (e/w) of 0.24 to 1.5 and Reynolds number ranging from 4000 to 17 000. The maximum increment in Nu and f reported in this research was 2.88 and 3.52 times, respectively. They also developed correlations for predicting the SAH performance.

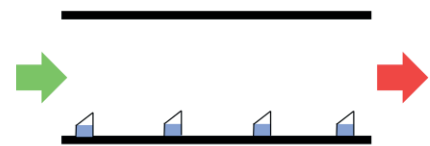


Fig. 48. Forward chamfered ribs as roughness elements (drawn after [100]).

Bhagoria et al. [130] conducted an experimental investigation to improve heat transfer in an SAH modified using wedge-shaped integral transverse ribs (Fig. 49), and developed correlations for heat transfer and friction factor. The utilised variables and physical parameters were: a Reynolds number varying from 3000 to 18 000, a relative roughness height varied between 0.015 and 0.033 and a rib wedge angle of 8° , 10° , 12° and 15° .

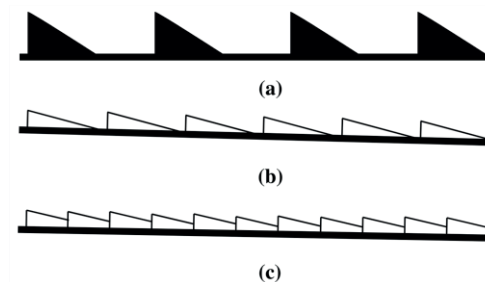


Fig. 49. Wedge-shaped ribs as roughness elements (drawn after [130]).

The authors reported a modified SAH to yield a Nusselt number up to 2.4 times higher and friction factor 5.3 times higher compared to a smooth duct. They also developed correlations for the Nusselt number and friction factor within an error band of $\pm 15\%$, using functional parameters such as rib spacing, rib height, rib wedge angle and Reynolds number.

Barik et al. [131] modified SAH using six different types of ribs: T-shaped rib, left arm inclined T-shaped rib, right arm inclined T-shaped rib, Y-shaped rib, and concave and convex T-shaped ribs, to enhance the performance of SAH, as shown in Fig. 50. They analysed a modified SAH through experimentation and numerically. They developed a physical model using various parameters, such as the pitch from 20 mm to 90 mm, stem height from 3 mm to 11 mm, an inclination angle of the rib from 0° to 60° and a Reynolds number varying from 5000 to 18 000. They obtained the highest increment of Nu and f for a T-shaped rib. They also reported the highest THP of 1.7 for a T-shaped rib.

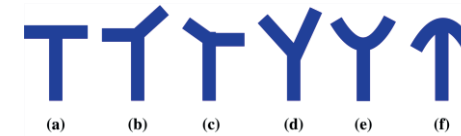


Fig. 50. Roughness element in the form of: T-shaped rib (a), right arm inclined T-shaped rib (b), left arm inclined T-shaped rib (c), Y-shaped rib (d), concave (e) and convex (f) T-shaped ribs (drawn after [131]).

Kumar and Goel [132] numerically investigated SAH considering various shapes of roughness elements, such as semi-circular, circular, triangular, square and rectangular, as depicted in Fig. 51, to enhance heat transfer through the absorber plate. The triangular passage was used with different parameters, such as Reynolds number varying from 3700 to 18 700, a relative rib-height of 0.0433, a relative rib pitch of 11, a rib aspect ratio (e/w) of 0.67 and 2.0, and a relative chamfered height of 0.75. They reported the highest THP value of 2.75 for a forward-chamfered rectangular rib at the optimum parameters of $Re = 18\,700$ and $e/w = 2.0$.

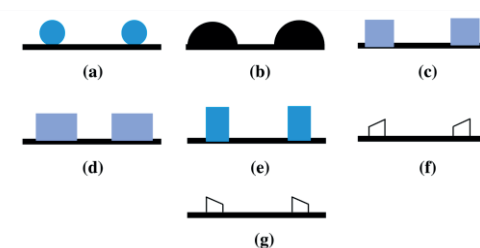


Fig. 51. Semi-circular, circular, triangular, square and rectangular shapes as roughness elements (drawn after [132]).

Zina et al. [133] analysed conventional SAH, modifying an absorber plate with triangular and semi-circular nooks with variable height of a three-rib set (Fig. 52) for performance enhancement. They performed a numerical analysis under changing pitch from 10 to 20, rib height from 0.7 to 2.2 and Reynolds

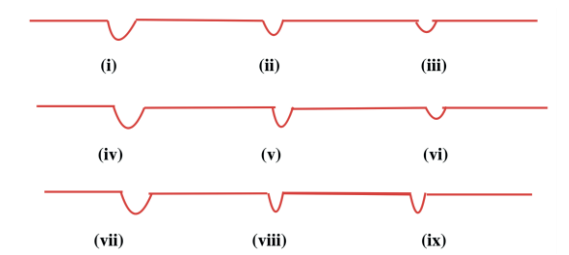


Fig. 52. Roughness element in the form of triangular and semi-circular nooks with variable height of three ribs (drawn after [133]).

number varying between 3800 and 18 000. The maximum enhancement in Nu and f found under that study were 2.55 and 0.22, respectively. They also reported an optimum THP of 1.93 for $Re = 5000$.

Gawande et al. [134] investigated SAH numerically, as well as experimentally, using various ribs such as 20° chamfered rib, right-angle triangular and reverse L-shaped ribs, shown in Fig. 53, in terms of increasing Nu and f . They employed operating and control parameters, such as a rib height (e) of 0.7 to 1.4, a relative roughness pitch (P/e) of 7.14 to 35.71, a relative roughness height (e/D) of 0.021 to 0.042 and a Reynolds number ranging from 3800 to 18 000. They reported a maximum augmentation in heat transfer and fluid flow characteristics of 3.218 and 4.047, respectively, for 20° chamfered ribs based modified SAH. The calculated THP values were 2.047, 2.035 and 1.903, for 20° chamfered rib, right-angle triangular and reverse L-shaped ribs, respectively.

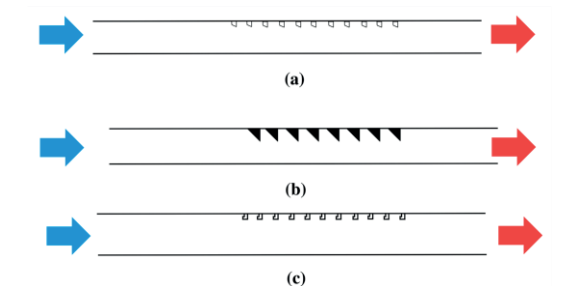


Fig. 53. Roughness elements in the form of 20° chamfered rib, right-angle triangular and reverse L-shaped ribs (drawn after [134]).

7.8. Inclined baffles

Aharwal et al. [135] performed an experimental investigation of a solar air heater modified with roughness of repeated square cross-section split-rib with gap for evaluation of heat transfer and friction behaviour of the fluid, as depicted in Fig. 54. They employed various process control parameters, including a relative roughness pitch of 10 (fixed), a relative roughness height of 0.0377, an angle of attack of 90° (fixed), a relative gap width ranging from 0.5 to 2.0, a relative gap position varying from 0.1667 to 0.667 and a Reynolds number varying from 3000 to 18 000. They obtained a maximum increment in Nu and f of 2.59 and 2.87, respectively.

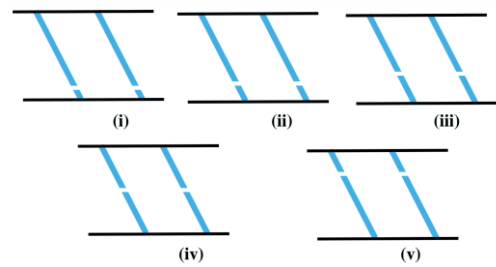


Fig. 54. Repeated square cross-section split-rib with gap as roughness elements (drawn after [135]).

Aharwal et al. [136] performed experimental investigation on modified SAH using integral repeated discrete square ribs as roughness element, as shown in Fig. 55. The used operating and physical parameters were: a relative roughness pitch of 4 to 10 (varying), a relative roughness height of 0.018 to 0.0377, an angle of attack of 30° to 90° (varying), a relative gap width ranging from 0.5 to 2.0, a relative gap position varying from 0.16 to 0.5 and a Reynolds number varying from 3000 to 18 000. In the study, the maximum obtained increase in Nu and f was 2.83 and 3.63 times, respectively, compared to a smooth SAH. The optimum parameters were: a relative gap position of 0.25 for a relative gap width of 1.0, a relative roughness pitch of 8.0, an angle of attack of 60° and a relative roughness height of 0.037.

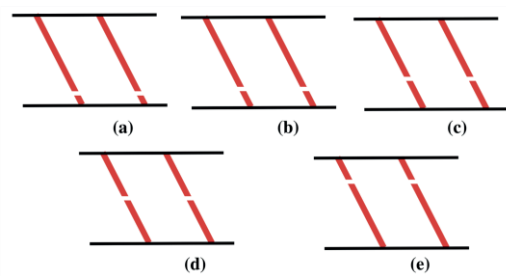


Fig. 55. Roughness element as integral repeated discrete square ribs (drawn after [136]).

Kumar et al. [137] modified SAH by using angled perforated baffles (Fig. 56) for improvement in heat transfer and fluid flow characteristics. They employed various parameters for evaluations, such as a relative baffle height of 0.5, a relative baffle pitch of 10, a relative hole position of 0.266, an open area ratio (β_0) of 12% and a Reynolds number varying from 5000 to 17 000. In the case under study, they obtained an enhancement in Nu and f of 3.82 and 7.14 times, respectively. The maximum determined value of THP for modified SAH was 1.94.

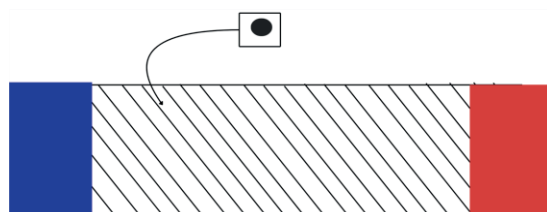


Fig. 56. Angled perforated baffles as roughness elements (drawn after [137]).

Kumar and Layek [138] experimentally analysed modified SAH using winglet type rib for enhancement in heat transfer and frictional flow behaviour (Fig. 57). They used operating parameters, such as a relative roughness pitch varying from 5 to 12, an attack angle changing from 30° to 75° , a relative roughness width varying from 3 to 7, a Reynolds number varying between 3000 to 22 000. They obtained an improvement in Nu and an increase in f of 2.83 and 3.63 times, respectively.

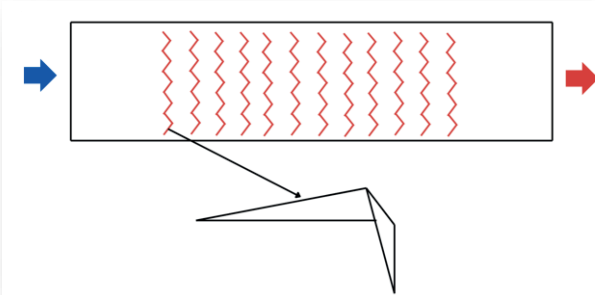


Fig. 57. Winglet type rib as a roughness element (drawn after [138]).

Kottayat et al. [139] performed experimental and numerical analysis of a modified SAH using discrete multiple inclined baffles, as shown in Fig. 58. They employed various parameters, including a relative baffle height of 0.1 (fixed), a relative baffle pitch varying from 0.6 to 1 and a Reynolds number varying from 6000 to 14 000. The obtained increments in Nu and f were 2.47 and 3.07 times, respectively, compared to a smooth SAH.

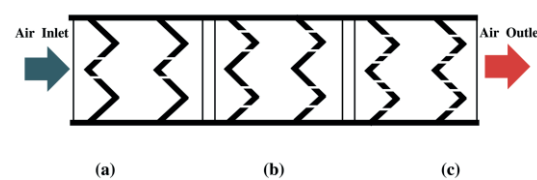


Fig. 58. Roughness elements in the form of multiple discrete inclined baffles (drawn after [139]).

Parsa et al. [140] conducted a numerical study of a modified SAH configuration using novel staggered cuboid baffles (shown in Fig. 59) to enhance heat transfer and fluid flow characteristics. The operating parameters employed were: relative length ranging from 1 to 3, relative width from 0.2 to 1.0, relative height from 0.1 to 0.9, relative pitch from 1.5 to 3.5, attack angle ranging from 0° to 90° , and Reynolds number varying from 5080 to 10 160. The highest obtained THP value was 3.43.



Fig. 59. Roughness element in the form of novel staggered cuboid baffles (drawn after [140]).

7.9. S-shaped ribs

Kumar et al. [12] carried out an experimental analysis of heat transfer and friction factor in a solar air heater duct with the S-shaped wire rib roughness, as shown in Fig. 60. They used parameters such as the relative roughness pitch (P/e) between 4 and 16, the relative roughness height (e/D_h) from 0.022 to 0.054, arc angles (α) ranging from 30° to 75° , and relative roughness width (W/w) between 1 and 4. Tests were conducted across a Reynolds number range of 2400 to 20 000. The findings reveal that the roughened duct exhibits superior performance compared to the smooth duct under the examined conditions. The optimal values of Nu and f were recorded at $W/w = 3$, $P/e = 8$, $\alpha = 60^\circ$, and $e/D_h = 0.043$. Additionally, predictive correlations for Nu and f were developed using the experimental results, accounting for the effects of both roughness and operating parameters.

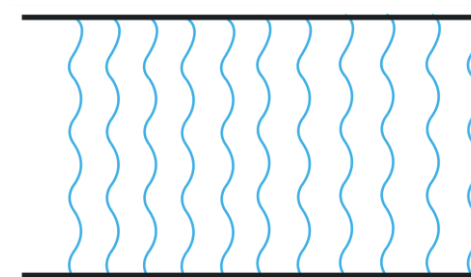


Fig. 60. S-shaped wire ribs as roughness elements (drawn after [12]).

7.10. Other roughness geometries

Jaurker et al. [141] experimentally investigated a modified SAH using rib-grooved artificial roughness, as displayed in Fig. 61. They employed operating parameters, such as Reynolds number ranging from 3000 to 21 000, relative roughness height varying from 0.0181 to 0.0363, relative roughness pitch range of 4.5 to 10, and groove position to pitch ratio of 0.3 to 0.7. They obtained a maximum performance of a modified SAH, such as a Nusselt number up to 2.7 times higher, while the friction factor rising up to 3.6 times compared to a smooth SAH. They also developed predictive correlations for estimating the Nusselt number and friction factor.

Karmare and Tikekar [142,143] experimentally studied SAH using metal grit ribs for enhancement of heat transfer and friction factor, as shown in Fig. 62 and Fig. 63. The parameters they

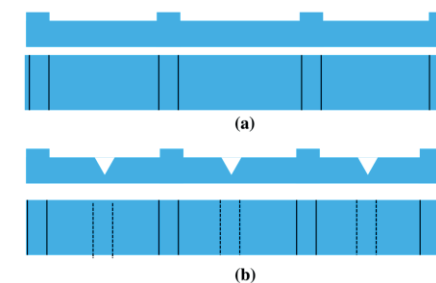


Fig. 61. Roughness element as rib-grooved [141].

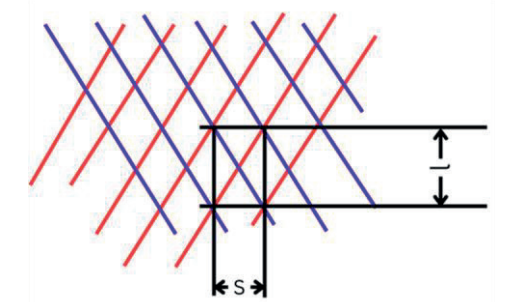


Fig. 62. Schematic of inclined roughness elements showing the spacing details of the ribs used.



Fig. 63. Roughness element in the form of metal grit ribs (drawn after [143]).

employed for the study were: a relative roughness height of grit between 0.035 and 0.044, a relative roughness pitch of grit between 12.5 and 36, a relative length of grit of 12.5 to 36, and Reynolds number varied from 4000 to 17 000. They found that the Nusselt number and friction factor increased by a maximum of 1.87 and 2.13 times, respectively, compared to a smooth SAH. They also made an effort to determine the predictive correlations for Nu and f .

Bopche and Tandale [144] experimentally investigated SAH using artificial roughness as an inverted U-shaped rib for better heat transfer and friction factor, as illustrated in Fig. 64. They employed various parameters for this investigation, such as e/D varied from 0.0186 to 0.03986, P/e changing from 6.667 to 57.14, α fixed at 90° , and Re varied from 3800 to 18 000. They obtained a maximum increase in Nu and f of 2.82 and 3.72, respectively.

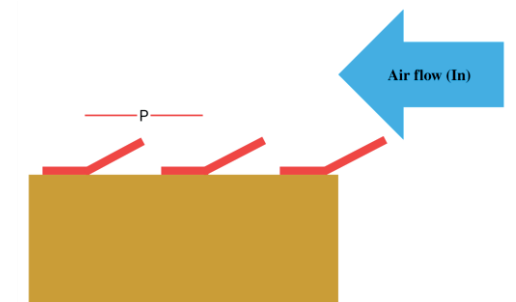
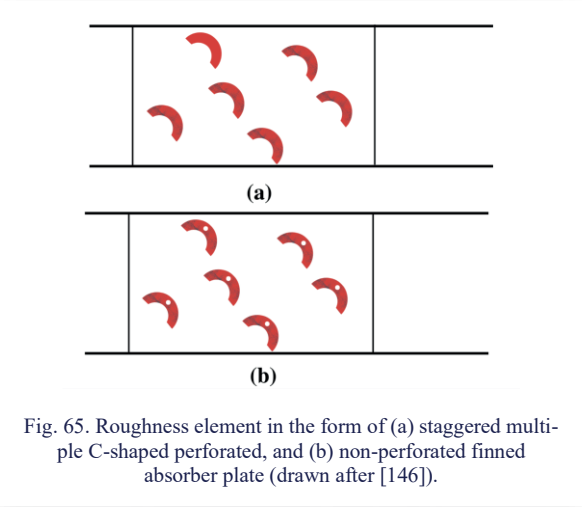


Fig. 64. Inverted U-shaped rib as a roughness element (drawn after [144]).

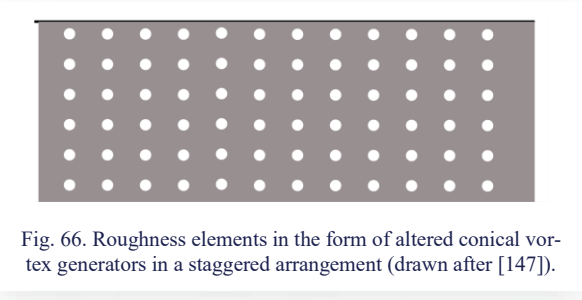
Kumar and Verma [145] numerically investigated modified SAH using artificial roughness as a sinusoidal protrusion rib with the view of higher Nusselt number and friction factor. They used various operating and control parameters, such as e of 1 mm, P varied from 10 to 18 mm and Re in the range of

4000 to 15 000. They obtained a maximum THP of 2.02 under optimum operating condition of $P = 10$ mm and $Re = 4000$.

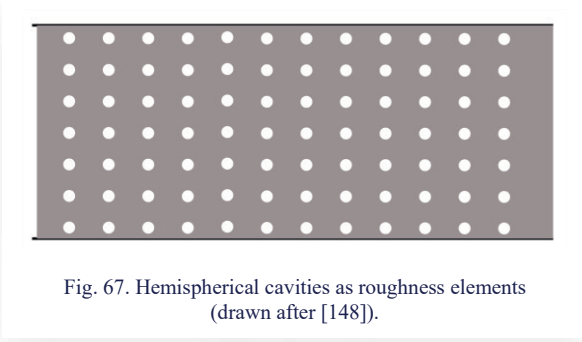
Saravanan et al. [146] conducted an experimental investigation to study the thermo-hydraulic performance of a solar air heater with staggered multiple C-shaped perforated and non-perforated finned absorber plate, as depicted in Fig. 65. The experiments were carried out for a relevant range of flow parameters, i.e. Reynolds number ranging from 3000 to 27 000, and geometric parameters, such as the relative pitch to gap ratio from 3.4 to 3.8, the relative height ratio from 0.3 to 0.7 and the relative perforated ratio from 1.5 to 3, with a focus on their effects on the maximum heat transfer enhancement. They reported a maximum THP of 1.64. They also developed correlations to predict and match the experimentally obtained Nu and f for non-perforated and perforated fins.



Bezbaruah et al. [147] analysed SAH numerically using altered conical vortex generators in a staggered arrangement (Fig. 66). The operating parameters used included a relative pitch ratio ranging from 2.67 to 15, a relative height ratio changing from 0.17 to 0.34 and a Reynolds number varied from 3500 to 16 000. The highest THP they obtained under these investigations was 2.04.

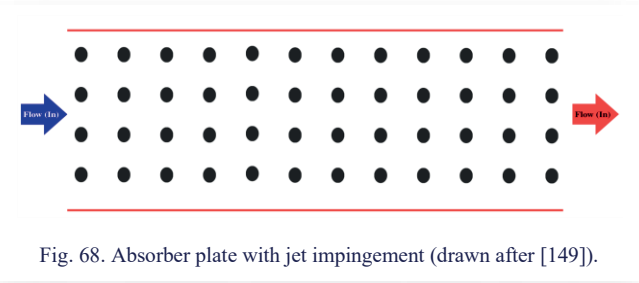


Goel et al. [148] conducted a comprehensive parametric investigation on SAH using hemispherical cavities as artificial roughness elements, displayed in Fig. 67, for an enhancement in heat transfer and fluid flow characteristics. They obtained maximum enhancement in the Nusselt number by a factor of 5.33 compared to a smooth SAH under the optimum operating condition of a transverse-pitch ratio, a longitudinal-pitch ratio and



a relative dimple depth of 11, 11 and 0.039, respectively, and a Reynolds number of 2160.

Yadav and Saini [149] performed a numerical study on a modified SAH using jet impingement technique for increasing the heat transfer, as shown in Fig. 68. They used a jet diameter ratio ranging from 0.065 to 0.195, a jet height ratio ranging from 0 to 0.433 and the Reynold number from 3500 to 17 500. They reported maximum increases in Nu and f of 7.58 and 9.01 times, respectively, compared to a smooth SAH under optimum conditions, such as a jet diameter ratio of 0.0650 and a jet height ratio of 0.216 at a Reynolds number of 17 500. They also quoted a maximum THP of 3.66 at a jet height ratio of 0.216 (fixed), jet diameter ratio of 0.065 (fixed) and a Reynolds number of 15 500 (fixed).



Wang et al. [150] investigated SAH modified with S-shaped ribs with gap to improve heat transfer (Fig. 69). They employed various parameters for investigations, such as a gap width ($g = 2$ to 6 mm), a rib width ($w = 30$ to 50 mm), a rib spacing ($p = 40$ to 60 mm), a rib height ($e = 4$ mm), a rib thickness ($s = 2$ mm), $P/e = 20$ to 30, $W/w = 3$ to 5, $e/D = 0.023$ to 0.036, $g/e = 1$ to 2, and Re varied from 2000 to 20 000. The reported maximum augmentation achieved for Nu and f was 5.42 and 5.87, respectively, at $Re = 19\,258$, $W/w = 4$, $P/e = 20$ and $g/e = 1.5$.

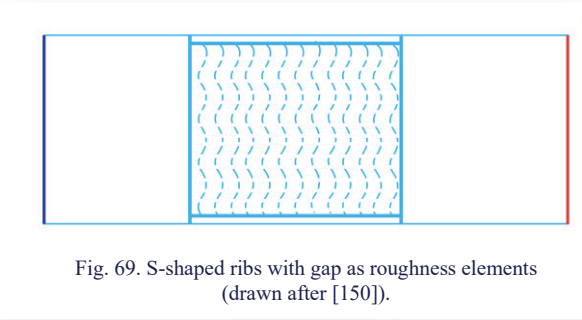


Table 5. Summary report.

Author(s) Ref.	Year	Roughness geometry	Reynolds number	Type of study	Main findings
Transverse ribs					
Prasad and Saini [84]	1988	Transverse rib	5000–50 000	Experimental	$Nu/Nu_s=2.38$ and $ff/f_s=4.25$
Gupta et al. [151]	1993	Transverse rib with small diameter	3000–18 000	Experimental	Studied the effects of e/D on heat transfer and friction factor at fixed $P/e=10$
Karwa [88]	2003	Transverse rib Inclined rib V continuous rib V discrete rib	2800–15 000	Experimental	Stanton number enhancement for: Continuous rib: V-up = 102–137%, V-down = 110–147% Discrete rib: V-up = 93–134%, V-down = 102–142%
Verma and Prasad [91]	2000	Rectangular duct with thin wires	5000–20 000	Experimental	Thermo-hydraulic optimisation of the roughness and other parameters for the maximum heat transfer and minimum pressure drop was made.
Layek et al. [89]	2009	Repeated transverse chamfered rib-groove roughness	3000–21 000	Experimental and analytical	Maximum Nu and f found were 2.6 and 3.5 times the smooth SAH. Also, the maximum THP reported was 1.76.
Saini and Saini [142]	1996	Expanded metal mesh	2000–13 500	Experimental	Correlations for the Nusselt number and friction factor in terms of these parameters were developed that can correlate the experimental data reasonably well.
Singh et al. [92]	2019	Multiple broken transverse ribs and square-wave shaped rib	3000–15 000	Experimental and CFD analysis	The maximum increment in Nu was 2.22, and the consequent value for f increment of 3.44 was obtained. The highest THP value of 1.49 was obtained, while the maximum thermal efficiency of 68.6% was achieved.
Singh et al. [93]	2021	Uniform/non-uniform square-wave profile transverse roughness rib	3000–15 000	CFD analysis	The non-uniform ribs were found to yield better heat transfer enhancement as compared to uniform ribs.
Sharma et al. [94]	2021	Transverse, inclined transverse, dimple, inclined dimple, arc and sine wave	3000–18 000	Experimental and numerical	The results revealed that the positioning and shape of baffle unswervingly affect THP of the solar air heater.
Mahanand and Senapati [95]	2020	Transverse inverted T-shaped rib	3800–18 000	Numerical	A thermal enhancement factor of 1.87 was obtained as a result of enhancement in heat transfer for the SAH with an inverted T-shaped rib roughened absorber plate.
Gawande et al. [96]	2018	Combined square and equilateral triangular ribs	3800–18 000	Numerical and experimental	A value of 1.86 was recorded as an optimum value of THP for the range of parameters investigated.
Kumar et al. [152]	2017	Circular and semicircular rib	4000–24 000	Numerical and experimental	Two new correlations were developed for the Nusselt number and friction factor as a function of Reynolds number and relative roughness height. The values obtained from the developed correlations agree well with data from the model.
Kumar and Layek [98]	2019	Circular rib	8551–11 149	LCT technique and CFD technique	The maximum values of heat transfer rate were obtained at a roughness pitch ratio of 10, with a minimum penalty of frictional losses.
Kumar et al. [99]	2019	Square rib	3900–17 900	Numerical	Correlations were developed for both friction factor and Nusselt number based on observed results.
Kumar et al. [153]	2017	Rectangular rib roughened	4000–18 000	Numerical	The maximum value of thermo-hydraulic performance parameter (THP), 1.89, was found in the case of relative roughness pitch (P/e), rib aspect ratio (e/w) and relative roughness height (e/D) of 10, 4.0 and 0.04, respectively, at the Reynolds number of 15 000.
Kumar et al. [97]	2017	Transverse semicircular rib	5000–10 000	Numerical	The maximum thermo-hydraulic performance was observed for a higher relative roughness height value ($e/D=0.067$) and a lower relative roughness pitch ($P/e=7.5$) in a semi-circular rib roughened equilateral triangular duct is of the order of 1.7.
Arc-shaped roughness					
Saini and Saini [102]	2008	Arc-shaped wire ribs	2000–17 000	Experimental	An enhancement of 3.80 times in heat transfer and 1.80 times in friction factor was observed at the optimum parameters compared to a smooth duct.
Singh et al. [103]	2014	Multiple arc-shaped ribs	2200–22 000	Experimental	Developed Nu and f correlations; observed effects of roughness on Nu and f

Table 5. Summary report (continued).

Author(s) Ref.	Year	Roughness geometry	Reynolds number	Type of study	Main findings
Sahu and Prasad [104]	2016	Arc-shaped ribs	2000–40 000	Analytical	Investigated the exergetic and effective efficiency
Gill et al. [109]	2017	Staggered piece and broken arc rib	2000–16 000	Experimental and CFD analyses	Examined the heat transfer and friction factor
Sahu and Prasad [154]	2017	Arc-shaped ribs	2900–35 000	Analytical	Examined the thermo-hydraulic performance.
Sara-vanakumar et al. [155]	2019	Arc shaped ribs Integrated with fins and baffles	2900–17 000	Analytical	The maximum exergy was obtained for 8 fins, a baffle length of 0.2 m, a baffle width of 0.015 m and a mass flow rate of 0.012 kg/s.
Bhuvad et al. [110]	2022	Apex-up discrete arc ribs	3000–14 000	Experimental	The results showed that the new apex-up discrete arc rib absorber plates had superior thermal performance to the existing best apex-down rib geometries. Maximum augmentation in Nu, achieved for an attack angle of 30°, was 2.92 times that of a smooth plate.
Kumar et al. [105]	2022	Discrete multiple-arc ribs	3000–19 000	Experimental	The Nusselt number and friction factor for a roughened solar thermal collector were found to be 7.61 and 6.48 times higher than those of a smooth channel.
Ghritlahre [107]	2021	Arc-shaped wire rib	No data*	Experimental	The above result reveals that apex upstream flow is more efficient than the apex downstream flow.
Korpale et al. [111]	2020	Rectangular section ribs	4000–20 000	Experimental and numerical	The maximum THP obtained was 2.77 at Reynolds number 20 000, relative rib pitch of 17.22, relative rib height of 0.044 and relative rib height of 0.5.
Kumar et al. [112]	2019	Discrete multiple arc-shaped ribs	11000–19 000	Experimental and numerical	A remarkable improvement in the thermal performance of the solar air heater was observed by varying roughness parameters, but it led to an increase in the pumping power of the solar air heater.
V-shaped ribs					
Momin et al. [114]	2002	V-shaped ribs	2500–18 000	Experimental	Found an optimal thermo-hydraulic performance at an angle of attack = 60°.
Patel and Lanjewar [118]	2021	V-geometry roughened	4000–15 000	Experimental	The maximum augmentation in Nu and f was found to be 2.51 and 2.70 times that of a smooth plate, respectively, for $g'/e=1$ and $d/w=0.65$. Maximum value of η_{THP} was found to be 1.82 for $g'/e=1$ and $d/w=0.65$.
Srivastava et al. [108]	2020	Arc-shaped with gap and V-shaped with gap	3000–15 000	Numerical	The maximum THP values for the V-shaped ribs and arc-shaped ribs were 2.21 and 1.5, respectively, at Re = 3000.
Dimple or protrusion ribs					
Kumar and Prasad [119]	2019	Concave dimple	2000–13 500	Experimental	The maximum enhancement in Nusselt number for varying relative dimple pitch, relative dimple height and relative dimple depth was, respectively, of the order of 2.6–3.55, 1.91 to 3.42, 3.09 to 3.94 times, and that of the friction factor was of the order of 1.62–2.79, 1.52 to 2.34 and 2.21 to 2.56 times over those of the 1-side roughened ones.
Sethi et al. [85]	2012	Dimple roughness in arc-shaped arranged	3600–18 000	Experimental	Studied the roughness effect on Nu and f .
Kumar and Verma [145]	2022	Sinusoidal protrusion rib (SPR)	5000–10 000	CFD analysis	The thermal efficiency of the SPR model was found to be 69% at Re = 15000 in the range of Re investigated.
W-shaped ribs					
Lanjewar et al. [125]	2011	W-shaped ribs	2300–14 000	Experimental	Found optimal thermo-hydraulic performance at an attack angle of 60° for the W-down rib.
Spherical ball roughness					
Murmu et al. [156]	2019	Inclined spherical ball	2500–18 500	Experimental	Developed Nu and f correlations.
Kumar and Murmu [157]	2021	Inclined spherical ball	2500–18 500	Experimental	Maximum thermal efficiency for roughened ducts was found to be 81% compared to 38% for a smooth duct under the parameters: relative roughness pitch = 12, relative roughness height = 0.036, height-diameter ratio = 1 and angle of attack = 55° at a mass flow rate of 0.0319 kg/s.

Table 5. Summary report (continued).

Author(s) Ref.	Year	Roughness geometry	Reynolds number	Type of study	Main findings
Non-uniform ribs					
Karwa et al. [158]	1999	Chamfer rib	3000–20 000	Experimental	Developed correlations for heat transfer and friction factor; additionally observed the effects of chamfer angle
Bhagoria et al. [130]	2002	Wedge-shaped ribs	3000–18 000	Experimental	Obtained $Nu_x=1.38$, and developed correlations for Nu and f
Karmare and Tikekar [143]	2007	Wire ribs-grid shape	4000–17 000	Experimental	Developed correlations for Nu and f
Barik et al. [131]	2021	T-rib, left arm inclined T-rib, right arm inclined T-rib, Y-rib, concave and convex T-ribs	5000–18 000	Numerical	Among all the considered ribs, the T-rib was found to be the best heat transfer enhancer, keeping the friction factor at a low value.
Kumar and Goel [132]	2021	Semi-circular, circular, triangular, square, rectangular	3700–18 700	Numerical	The performance of SAH was strongly influenced by both shapes of the roughness element and the cross-sectional area of flow-passage through which air flows.
Inclined baffles					
Kumar et al. [137]	2022	Angled perforated baffles	5000–17 000	Experimental, CCD, ANN technique	The STC with roughened angled perforated baffles enhanced Nu and f by 3.82 and 7.14 times, respectively, as compared to the STC without baffles. The angled perforated baffles at α of 55° provided the highest THP at Re of 9000. A THP value of 1.94 was obtained for the array of designed parameters.
Kottayat et al. [139]	2022	Discrete multiple inclined baffle	6000–14 000	CFD and experimental	A maximum thermo-hydraulic performance of 2.69 was obtained for the gap at the trailing apex. The proposed design yielded a higher collector efficiency, 55–70%, compared to the ribbed rectangular SAH design exhibiting 30–55%. With lower exergy losses, the proposed SAH design had higher exergetic efficiency (1.5–2.2%) than the ribbed rectangular SAH (0.9–1.7%) for the range of Re studied.
Parsa et al. [140]	2021	Staggered cuboid baffles	5080, 7620, 10 160	Numerical	It is indicated that the relative baffle height (c/H) and relative baffle pitch (s/H) are the most influential parameters on the efficiency and thermo-hydraulic performance factor.
Kumar and Layek [138]	2020	Winglet shaped ribs	No data**	Experimental	By comparing with the smooth surface, the studied roughened surface could significantly improve the thermal efficiency of a solar air heater system. The highest value of thermal efficiency of the collector surface was 69% at the maximum mass flow rate.
Other roughness geometries					
Jaurker et al. [141]	2006	Rib-grooved roughness	3000–21 000	Experimental	Developed the correlations for Nu and f .
Patel et al. [159]	2021	Discrete reverse NACA profile ribs	6000–18 000	Experimental	Maximum Nu and THP were found to be 102.08 and 2.578, respectively.
Saravanan at al. [146]	2021	Staggered C-shape finned	3000–27 000	Experimental	The thermal performance and friction factor the for C-shape finned absorber plate without a perforated hole for a relative pitch-to-gap ratio of 3.8 and a relative height ratio of 0.6 were 2.61 and 5.93 times more than for the smooth plate absorber plate, respectively. The perforated fin with the same relative ratios gave thermal performance and friction factor 2.67 and 5.34 times higher than the smooth absorber plate solar air heater, respectively.
Bezbaruah et al. [147]	2021	Staggered conical vortex generators	3500–16 000	Experimental and numerical	The highest thermo-hydraulic performance parameter of 2.04 was achieved.
Goel et al. [148]	2021	Hemispherical dimple cavities	2100–18 500	Experimental and numerical analysis	The best augmentation in Nusselt number obtained was of the order of 5.33 which was observed for a transverse pitch ratio, a longitudinal-pitch ratio and a relative dimple depth values of 11, 11 and 0.039, respectively, at a Reynolds number of 2160.
Arunkumar et al. [160]	2021	Rectangular perforated duct inserts	3000–18 000	Experimental	It was found based on the experiments that thermal efficiency was maximum for the height ratio (h/H) of 0.66 when compared to height ratios of 0.50 and 0.83.
Yadav and Saini [149]	2020	Impinging jet	3500–17 500	Numerical	The maximum obtained thermo-hydraulic performance parameter value of 3.66 corresponded to a jet height ratio of 0.216 and a jet diameter ratio of 0.065 for the Reynolds number of 15 500.

Table 5. Summary report (continued).

Authors	Year	Roughness geometry	Reynolds number	Type of study	Main findings
Other roughness geometries					
Wang et al. [150]	2020	“S”-shaped ribs with gap	2000–20 000	Experimental	Compared with a smooth plate, the thermal efficiency of a solar air heater with artificial roughness was improved by 13% to 48% under different conditions. However, the work section pressure drop of this new solar air heater was within the range of 15.8–30 Pa.
Kumar and Verma [121]	2020	Sinusoidal rib	4000–15 000	Numerical	The results are also compared with the triangular and square rib and found that the optimal THP of the proposed rib was 20.06% and 33.31% higher.
Gawande et al. [134]	2020	Chamfered rib	3800–18 000	Experimental and numerical	The chamfered square rib with a 20° chamfer angle was found to provide the best thermo-hydraulic performance and, hence, was selected for further thermal analysis.
Zina et al. [133]	2020	Triangular ribs with semi-circular nooks	3800–18 000	Numerical	The optimum THP value of 1.93 could be obtained fo case C1 and Re of 5000.
Kumar et al. [100]	2018	Forward facing chamfered rectangular ribs	4000–17 000	Numerical	A remarkable change in f and Nu values was observed by varying both Re and roughness parameters.
Kumar et al. [161]	2020	Forward and backward facing arced ribs	2300–23 000	Experimental	Thermo-hydraulic performance improved by 3.61 times compared to the smooth flat plate solar air heater, for a curved-ribbed SAH for $W/w=3$, $P/e=8$, $g/e=1$, $e/D=0.045$, and $d/x=0.65$ at Re = 23 000.

* Mass flow rate $\dot{m} = 0.007\text{--}0.022$ kg/s.
** Mass flow rate $\dot{m} = 0.00531\text{--}0.04041$ kg/s.

8. Correlations for Nusselt number and friction factor

In the field of solar air heaters, the development of correlations is pioneering in performance prediction and optimisation, design improvement, scaling and adaptation, generalisation of results, energy efficiency and sustainability, cost reduction and customisable solutions. The first correlation, reported in 1991, involved transverse ribs to develop correlations for the Nusselt number and friction factor in a modified solar air heater. Various geometries have been explored since then, including arc-shaped ribs, V-shaped wire ribs, wire rib grid shapes, wedge-shaped ribs, chamfered rectangular ribs, angled perforated baffles, dimpled and protruded surface geometries and spherical ball-shaped ribs. The parameters used in these studies include e/D_h , P/e , Re, e/D , α , B/S , W/H , g/P , ϕ , e^+ , y , d/H , p/H , y/H , P , e/w , d/w , g/e , $\alpha/90^\circ$, N_g , D/L , g/H , P/H , X/D , Y/D , d/x , W/w , w/e , G/L , l/s , L/e , S/e , e'/e , O/H , β , and V . The error range associated with the correlations developed in these studies typically falls between 5% and 30%, depending on the complexity of the geometry and the experimental conditions. Table 6 presents the numerous correlations obtained by various investigators with the ranges of parameters selected for their studies.

9. Nusselt number and friction factor enhancement

In the field of renewable energy conservation systems, the performance of the system plays a vital role. In the flat plate solar air heater, the performance evaluation has been reported by many researchers. The conventional SAH has been modified using artificial roughness. The size, orientation, shape, material used, flow rate of fluid, gaps, and many other factors affect the values of Nu and f . The comparison of various geometrical configurations used has been discussed in Table 7. As the enhancement of heat transfer and fluid friction in comparison with a smooth duct are subjected to various operating and control parameters, the maximum values can be obtained at the optimum parameters in the examined range of Reynolds number (mass flow rate), as listed in Table 7 as Nu/Nu_s and f/f_s , respectively. In this review study, the effort has been put on major geometries, such as transverse ribs, arc-shaped ribs, dimple or protrusion ribs, W-shaped ribs, V-shaped ribs, spherical ball roughness, non-uniform ribs, inclined baffles, and other roughness geometries.

Table 6. Details of correlations developed.

Author(s)	Novel geometry	Parameters	Correlation developed
Transverse ribs			
Hong and Hsieh [162]	Transverse ribs	$P/e = 5.31$ $e/D_h = 0.19$	Nu = 0.056Re _{<i>h</i>} ^{0.74} for Tu = 0.04 Nu = 0.03Re _{<i>h</i>} ^{0.79} for Tu = 0.07 Nu = 0.019Re _{<i>h</i>} ^{0.83} for Tu = 0.07
Prasad and Saini [84]	Small diameter protrusion wire	$P/e = 10 - 20$ $e/D_h = 0.020 - 0.033$ Re = 500 – 50000	$\overline{St} = \bar{f}/2 \left[1 + \sqrt{\bar{f}/2} (4.5(e^+)^{0.28} Pr^{0.57} - 0.95(P/e)^{0.53}) \right]$ $f = 2/[0.95(P/e)^{0.53} + 2.5\ln(D_h/2e) - 3.75]^2$
Gupta et al. [163]	Small diameter transverse ribs	$e/D = 0.018 - 0.52$ Re = 3000 – 18 000	Nu = 0.000824(e/D) ^{−0.178} (W/H) ^{0.284} Re ^{1.062} , for $e^+ < 35$ Nu = 0.000307(e/D) ^{−0.469} (W/H) ^{0.245} Re ^{1.062} , for $e^+ \geq 35$ $f = 0.06412(e/D)^{0.019}(W/H)^{0.237}Re^{-0.185}$
Karwa [88]	Inclined, discrete and continuous wire rib	$e/D_h = 0.467 - 0.05$ $P/e = 10$ $\alpha = 60^\circ, 90^\circ$ $B/S = 3$ $W/H = 7.19 - 7.75$ Re = 2500 – 18 000	$G = 32.2e^{-0.006}(W/H)^{0.5}(P/e)^{2.56} \exp[0.7343\ln(P/e)^2](e^+)^{0.08}$ $f = 1.66e^{-0.0078}(W/H)^{-0.4}(P/e)^{0.2695} \exp[-0.719\ln(P/e)^2(e^+)^{-0.075}]$ for $7 \leq e^+ \leq 20R$ $f = 1.325e^{-0.0078}(W/H)^{-0.4}(P/e)^{0.2695} \exp[-0.719\ln(P/e)^2(e^+)^{-0.075}]$ for $20 \leq e^+ \leq 60R$
Layek et al. [164]	Transverse chamfered rib groove combination	$P/e = 4.5 - 10$ $\Phi = 5 - 300$ $e/D = 0.022 - 0.04$ Re = 3000 – 18 000 $g/p = 0.3 - 0.6$	Nu = 0.00206Re ^{0.936} (e/D) ^{0.349} (P/e) ^{3.0318} × exp [−0.868(ln (P/e)) ²](g/p) ^{1.108} × exp [2.486 {ln (g/p)) ² + 1.406{ln (g/p)) ³ }] $f = 0.001227Re^{-0.199}(e/D)^{0.585}(P/e)^{7.19}$ × exp [−1.854{ln (P/e)) ²](g/p) ^{0.645} × exp [1.513{ln (g/p)) ² + 0.8662{ln (g/p)) ³ }]
Verma and Prasad [91]	Small diameter transverse protrusion wire	$e/D = 0.01 - 0.03$ $P/e = 10 - 40$ $e^+ = 8 - 42$ Re = 5000 – 20 000	Nu = 0.08596(P/e) ^{−0.54} (e/D) ^{0.072} Re ^{0.723} , for $e^+ \leq 24$ Nu = 0.0245(P/e) ^{−0.016} (e/D) ^{0.021} Re ^{0.802} , for $e^+ > 24$ $f = 0.0245(P/e)^{-0.0206}(e/D)^{0.021}Re^{-1.25}$
Mahand and Senapati [95]	Transverse inverted-T shaped rib	$e/D_h = 0.042$ $P/e = 7.14 - 17.86$ Re = 3800 – 18 000	$\overline{Nu} = 0.052927Re^{0.8379}(P/e)^{-0.1357}$ $\bar{f} = 0.329536Re^{-0.2325}(P/e)^{-0.1305}$
Gawande et al. [134]	Chamfered rib, right angle triangular and reverse L-shaped ribs, invented from square ribs	$q = 1000$ W/m ² Re = 3800 – 18 000 $P/e = 7.14 - 35.71$ $e/D = 0.021 - 0.042$ $\alpha = 0^\circ - 40^\circ$	For 20° chamfered rib Nu = 0.0943Re ^{0.8248} (P/e) ^{−0.3022} For right-angle triangular rib Nu = 133.36Re ^{0.8183} (e/D) ^{3.6305} exp[0.4480ln (e/D) ²](P/e) ^{−0.4807} × exp[0.055ln (P/e) ²] For reverse L-shaped rib Nu = 0.032Re ^{0.8332} (P/e) ^{0.3479} exp[−0.1004ln(P/e) ²] For 20° chamfered rib $f = 0.653Re^{-0.2883}(P/e)^{-0.2087}$ For right-angle triangular rib $f = 42.979Re^{-0.2778}(e/D)^{1.1517} \exp[0.1105\ln(e/D)^2](P/e)^{-1.7453}$ × [0.3133ln (P/e) ²] For reverse L-shaped rib $f = 0.2805Re^{-0.2617}(P/e)^{0.0815} \exp[-0.0319\ln(P/e)^2]$
Kumar et al. [152]	Circular and semicircular rib	$L = 1600$ mm $H = 40$ mm $D = 77$ mm $d = 4, 6, 8, 10$ mm $P = 30$ mm $y = 10, 20, 30$ mm $d/H = 0.1, 0.15, 0.20, 0.25$ $P/H = 0.75$ $y/H = 0.25, 0.5, 0.75$ $q = 800$ W/m ² Pr = 0.72 Re = 5000 – 10 000 (6 values) $\alpha = 25^\circ$	Nu = 0.44D _{<i>h</i>} ^{0.69} (d/H) ^{0.05} exp{−0.0165[ln (d/H)) ²]} $f = 16.4D_h^{-0.406}(d/H)^{2.5} \exp\{0.44[\ln(d/H)]^2\}$
Kumar et al. [99]	Square rib	$P/e = 10$ $e/D = 0.013 - 0.05$ Re = 4000 – 18000	Nu = 0.00415Re ^{0.95} (P/e) ^{1.217} (e/D) ^{0.243} exp{−0.265[ln (P/e)) ²]} $f = 6.436Re^{-0.4}(P/e)^{-1.033}(e/D)^{0.278} \exp\{-0.224[\ln(P/e)]^2\}$
Kumar et al. [153]	Rectangular rib roughened	$P/e = 5 - 15$ $e/D = 0.02 - 0.4$ $e/w = 0.25 - 4.0$ Re = 4000 – 18 000	Nu _{avg} = 0.365{Re ^{0.83} (P/e) ^{−1.322} (e/w) ^{0.122} (e/D) ^{0.216} × exp[0.265(ln(P/e)) ²]} $f_{avg} = 3.394 \times 10^2 \{Re^{-0.56}(P/e)^{-2.294}(e/w)^{0.130}(e/D)^{0.378}$ × exp[0.408(ln(P/e)) ²]}

Table 6. Details of correlations developed (*continued*).

Author(s)	Novel geometry	Parameters	Correlation developed
Aharwal et al. [136]	Inclined and transverse ribs	$e/D_h = 0.0377$ $P/e = 10$ $\alpha = 60^\circ$ $d/W = 0.167 - 0.5$ $W/H = 5.87$ $Re = 3000 - 18\,000$	$Nu = 0.0102Re^{e^{1.148}(e/D)^{0.51}\{1 - [0.25 - (d/W)]^2(0.01[1 - (g/e)]^2\}}}$ $f = 0.5(e/D)^{0.72}Re^{-0.0836}$
Varun et al. [165]	Inclined and transverse wire ribs	$P/e = 10$ $e/D_h = 0.030$ $Re = 2000 - 20\,000$	$Nu = 0.0006(P/e)^{0.0104}Re^{1.213}$ $f = 1.0858(P/\bar{e})^{0.0114}Re^{-0.3685}$
Arc-shaped ribs			
Saini and Saini [102]	Arc-shaped wire ribs	$g/e = 0.5 - 2$ $P/e = 10$ $\alpha/90 = 0.333 - 0.666$ $d/w = 0.167 - 0.5$ $W/H = 5.87$ $Re = 2000 - 17\,000$	$Nu = 0.001047(e/D)^{0.3772}(\alpha/90)^{-0.1198}Re^{1.123}$ $f = 0.14408(e/D)^{-0.1765}(\alpha/90)^{0.1185}Re^{-0.17103}$
Bhuvad, Azad and Lanjewar [110]	Apex-up discrete arc ribs	$W = 200\text{ mm}$ $W/H = 8.0$ $\alpha = 30^\circ, 45^\circ, 60^\circ$ $e/D_h = 0.045$ $P/e = 10$ $N_g = 3$ $g/e = 3$	$R = \sqrt{(2/l)} + 2.5\ln(2e/D_h) + 3.75$ $e^+ = \sqrt{(f/2)}Re(e/D_h)$ $G = (f/2St) - 1\sqrt{(2/f)} + R$
Kumar et al. [105]	Discrete multi-arc ribs	$D_d/L_v = 0.27 - 0.86$ $g_w/H_r = 0.32 - 1.72$ $H_r/H = 0.025 - 0.047$ $P_r/H = 0.58 - 3.1$ $\alpha = 35^\circ - 65^\circ$ $X/D_h = 0.47 - 1.72$ $Y/D_h = 0.46 - 0.82$ $D_j/D_h = 0.065$ $W/W_{AC} = 5.0$ $Re = 3000 - 19\,000$	$Nu = 4.03 \times 10^{-18}Re^{0.4674}(D_d/L_v)^{-0.3183}$ $\times \exp\{-0.1854[\ln(D_d/L_v)]^2\}(g_w/H_r)^{-0.0566}$ $\times \exp\{-0.0172[\ln(g_w/H_r)]^2\}(H_r/H)^{-2.7231}$ $\times \exp\{-0.4179[\ln(H_r/H)]^2\}(P_r/H)^{-0.0137}$ $\times \exp\{-0.1373[\ln(P_r/H)]^2\}\alpha_a^{2.0055}$ $\times \exp\{-1.1569[\ln(\alpha_a)]^2\}(X/D_h)^{-0.0254}$ $\times \exp\{-0.0888[\ln(X/D_h)]^2\}(Y/D_h)^{-0.4181}$ $\times \exp\{-0.4506[\ln(Y/D_h)]^2\}$ $f = 4.03 \times 10^{-18}Re^{-0.338}\exp\{0.0239\{\ln(Re)\}^2\}(D_d/L_v)^{-0.9527}$ $\times \exp\{-0.6782[\ln(D_d/L_v)]^2\}(g_w/H_r)^{-0.167}$ $\times \exp\{-0.01021[\ln(g_w/H_r)]^2\}(H_r/H)^{-6.5696}$ $\times \exp\{-1.0137[\ln(H_r/H)]^2\}(P_r/H)^{-0.00067}$ $\times \exp\{-0.1612[\ln(P_r/H)]^2\}\alpha_a^{15.909}$ $\times \exp\{-2.0566[\ln(\alpha_a)]^2\}(X/D_h)^{-0.0409}$ $\times \exp\{-0.2004[\ln(X/D_h)]^2\}(Y/D_h)^{-0.4866}$ $\times \exp\{-0.6556[\ln(Y/D_h)]^2\}$
Kumar et al. [112]	Discrete multiple arc-shaped ribs	$g/e = 0.5 - 1.5$ $N_g = 1 - 3$ $d/x = 0.3 - 0.9$ $Re = 11000 - 19000$	$Nu = -619.14 + 161.14(g/e) + 632.75(d/x) + 0.054Re + 5.45N_g + 8.37 \times 10^{-5}(g/e)Re + -2.79 \times 10^{-4}(d/x)Re + 0.72(d/x)N_g + 5.39 \times 10^{-4}ReN_g - 77.01(g/e)^2 + -495.62(d/x)^2 - 1.28 \times 10^{-6}Re^2 - 1.66N_g^2$ $f = -5.06 \times 10^{-3} + 0.014(g/e) + 0.0775(d/x) + 1.51 \times 10^{-6}Re + 4.69 \times 10^{-4}N_g + 7.83 \times 10^{-4}(g/e)(d/x) + 3.65 \times 10^{-4}(g/e)N_g + -2.25 \times 10^{-7}(d/x)Re - 1.79 \times 10^{-7}ReN_g + -7.05 \times 10^{-3}(g/e)^2 - 0.059(d/x)^2 + -1.023 \times 10^{-10}Re^2 + 9.53 \times 10^{-4}N_g^2$
Kumar et al. [161]	Innovatively arranged arc (combination of alternately arranged forward and backward facing arced ribs)	$Re = 2300 - 23000$ $W/H = 11$ $e/D = 0.018 - 0.045$ $\alpha = 30^\circ - 75^\circ$ $g/e = 0.5 - 2.0$ $W/w = 1 - 4$ $d/x = 0.25 - 0.85$ $P/e = 4 - 16$	$Nu_{avg} = 2 \times 10^{-7}Re^{1.4079}(g/e)^{-0.1264}(d/x)^{-0.1596}$ $\times \alpha^{3.5008}(W/w)^{0.5932}(P/e)^{0.2322}(e/D)^{0.2958}$ $\times \exp\{-0.2429[\ln(g/e)]^2\}\exp\{-0.2484[\ln(d/x)]^2\}$ $\times \exp\{-0.4123[\ln\alpha]^2\}\exp\{-0.2534[\ln(W/w)]^2\}$ $\times \exp\{-0.0479[\ln(P/e)]^2\}$ $f = 0.004Re^{-0.217}(g/e)^{-0.1152}(d/x)^{-0.2848}$ $\times \alpha^{1.8628}(W/w)^{0.3868}(P/e)^{0.6392}(e/D)^{0.1914}$ $\times \exp\{-0.1877[\ln(g/e)]^2\}\exp\{-0.2182[\ln(d/x)]^2\}$ $\times \exp\{-0.2304[\ln\alpha]^2\}\exp\{-0.1814[\ln(W/w)]^2\}$ $\times \exp\{-0.1668[\ln(P/e)]^2\}$

Table 6. Details of correlations developed (*continued*).

Author(s)	Novel geometry	Parameters	Correlation developed
Korpale et al. [111]	Rectangular section ribs	$Re = 4000 - 20\,000$ $P/e = 1 - 60$ $e/D = 0.017 - 0.252$ $w/e = 0.5 - 10$	$Nu = 0.024Re^{0.8032}(P/e)^{1.4013}(e/D)^{0.6762}(w/e)^{-0.5388}e^A$ $A = -0.1728\log(P/e)\log(e/D) + 0.1986\log(P/e)\log(w/e) + 0.0658\log(e/D)\log(w/e) - 0.3235[\log(P/e)]^2$ $f = 24.167Re^{-0.1953}(P/e)^{0.1906}(e/D)^{2.5305}(w/e)^{-0.3140}e^B$ $B = -0.2073\log(P/e)\log(e/D) + 0.1293\log(P/e)\log(w/e) + 0.0497\log(e/D)\log(w/e) - 0.1880[\log(P/e)]^2 + 0.2287[\log(e/D)]^2$
V-shaped ribs			
Gupta et al. [151]	V-shaped wire ribs and V-shaped inclined wire ribs	$e/D = 0.02 - 0.053$ $P/e = 7.5 - 10$ $\alpha = 30^\circ - 90^\circ$ $Re = 5000 - 20\,000$	$Nu = 0.00824(e/D)^{-0.178}(W/H)^{0.288}Re^{1.062}$, for $e^+ < 35$ $Nu = 0.00307(e/D)^{-0.469}(W/H)^{0.245}Re^{0.812}$, for $e^+ \geq 35$ $f = 0.06412(e/D)^{0.019}(W/H)^{0.0237}Re^{-0.185}$
Mulu-work et al. [166]	V-shaped staggered discrete wire ribs	$e/D = 0.02$ $B/S = 3 - 9$ $\alpha = 60^\circ$ $Re = 2000 - 15500$	$Nu = 0.00534(B/S)^{1.3496}Re^{1.2991}$ $f = 0.7117(B/S)^{0.0636}Re^{-2.991}$
Momin et al. [114]	V-shaped continuous wire rib	$e/D_h = 0.02 - 0.034$ $P/e = 10$ $\alpha = 30^\circ - 90^\circ$ $Re = 2500 - 18\,000$	$Nu = 0.067Re^{0.888}(e/D_h)^{0.424}(\alpha/60^\circ)^{-0.077}\exp[-0.782(\ln\alpha/60^\circ)^2]$ $f = 6.266Re^{-0.425}(e/D_h)^{0.565}(\alpha/60^\circ)^{-0.093}\exp[-0.719(\ln\alpha/60^\circ)^2]$
Hans et al. [167]	Multiple V-shaped ribs	$P/e = 6 - 12$ $c/D = 0.0213 - 0.0422$ $\alpha = 30^\circ - 75^\circ$ $W/w = 1 - 10$ $Re = 2000 - 20\,000$	$Nu = 3.35 \times 10^{-5}Re^{0.92}(e/D)^{0.77}(W/w)^{0.43}(\alpha/90^\circ)^{-0.49}$ $\times \exp\{-0.1177(\ln(W/W))^2\}$ $\times \exp\{-0.61[\ln(\alpha/90^\circ)]^2\}(P/e)^{8.54}\exp\{-2.040[\ln(P/e)]^2\}$ $f = 4.47 \times 10^{-4}Re^{-0.3188}(e/D)^{0.73}(W/w)^{0.22}(\alpha/90^\circ)^{-0.39}$ $\times \exp\{-0.52[\ln(\alpha/90^\circ)]^2\}(P/e)^{8.9}\exp\{-2.133[\ln(P/e)]^2\}$
Singh et al. [113]	Discrete V-shaped	$P/e = 4 - 12$ $g/e = 0.5 - 2$ $\alpha = 30^\circ - 75^\circ$ $d/w = 0.2 - 0.8$ $e/D_h = 0.015 - 0.043$ $Re = 3000 - 15\,000$	$Nu = 2.36 \times 10^{-5}Re^{0.90}(P/e)^{3.5}(\alpha/60^\circ)^{-0.023}(g/e)^{-0.014}(e/D_h)^{0.47}$ $\times \exp\{-0.84[\ln(p/e)]^2\}\exp\{-0.72[\ln(\alpha/60^\circ)]^2\}$ $\times \exp\{-0.05[\ln(d/w)]^2\}\exp\{-0.15[\ln(g/e)]^2\}$ $f = 44.13 \times 10^{-2}Re^{-0.126}(P/e)^{2.74}(\alpha/60^\circ)^{-0.034}$ $\times (d/w)^{-0.058}(g/e)^{-0.031}(e/D_h)^{0.70}$ $\times \exp\{-0.685[\ln(P/e)]^2\}\exp\{-0.93[\ln(\alpha/60^\circ)]^2\}$ $\times \exp\{-0.058[\ln(d/w)]^2\}\exp\{-0.21[\ln(g/e)]^2\}$
Kumar et al. [117]	Multiple V-shaped ribs with gap	$P/e = 6 - 12$ $g/e = 0.5 - 1.5$ $e/D = 0.022 - 0.043$ $W/w = 1 - 10$ $G_d/L_v = 0.24 - 0.80$ $\alpha = 30^\circ - 75^\circ$ $Re = 2000 - 20\,000$	$Nu = 8.534 \times 10^{-3}Re^{-0.932}(e/D)^{0.175}(W/w)^{0.506}$ $\times \exp\{-0.753[\ln(W/w)]^2\}(G_d/L_v)^{-0.0348}$ $\times \exp\{-0.0653[\ln(G_d/L_v)]^2\}(g/e)^{-0.0708}$ $\times \exp\{-0.223[\ln(g/e)]^2\}(\alpha/60^\circ)^{-0.0239}$ $\times \exp\{0.1153[\ln(\alpha/60^\circ)]^2\}(P/e)^{1.196} \times \exp\{-0.2805[\ln(P/e)]^2\}$ $f = 3.1934Re^{-0.351}(e/D)^{0.268}(W/w)^{0.1132}$ $\times \exp\{-0.974[\ln(W/w)]^2\}(G_d/L_v)^{0.0610}$ $\times \exp\{-1.065[\ln(G_d/L_v)]^2\}(g/e)^{-0.1769}$ $\times \exp\{-0.6349[\ln(g/e)]^2\}(\alpha/60^\circ)^{0.1553}$ $\times \exp\{-0.1527[\ln(\alpha/60^\circ)]^2\}(P/e)^{-0.7941} \times \exp\{0.1486[\ln(P/e)]^2\}$
Spherical ball roughness			
Murmu et al. [156]	Inclined spherical ball	$P/e = 15$ $e/D_h = 0.036$ $e/d_b = 1$ $\alpha = 55^\circ$	$Nu = 5.8 \times 10^{-7}Re^{1.0459}(e/D_h)^{2.8326}(p/e)^{1.56}(e/d_b)^{-0.231}(\alpha/55)^{-1.089}$ $\times \exp(-0.87)[\ln(e/D_h)]^2\exp(-4.86)[\ln(P/e)]^2$ $\times \exp(-0.1817)[\ln(e/d_b)]^2\exp(-2.6542)[\ln(\alpha/55)]^2$ $f = 2.38 \times Re^{-0.05629}(e/D_h)^{0.017}(P/e)^{1.89}(e/d_b)^{-1.101}(\alpha/55)^{-0.1127}$ $\times \exp(0.0664)[\ln(e/D_h)]^2\exp(-0.0429)[\ln(P/e)]^2$ $\times \exp(-0.526)[\ln(e/d_b)]^2\exp(-2.352)[\ln(\alpha/55)]^2$
Non uniform rib roughness			
Karmare and Tikekar [143]	Wire ribs-grid shape	$e/D_h = 0.035 - 0.044$ $P/e = 12.5 - 36$ $l/s = 1.72 - 1$ $Re = 4000 - 17000$ $l/e = 25 - 71.87$	$Nu = 2.4(e/D_h)^{0.42}(l/s)^{-0.146}Re^{1/3}(P/e)^{-0.27}$ $f = 15.55(e/D_h)^{0.91}(l/s)^{-0.27}Re^{-0.26}(P/e)^{-0.51}$
Saini and Saini [142]	Expanded metal mesh	$s/e = 15.62 - 46.87$ $e/D_h = 0.012 - 0.0390$ $Re = 1900 - 13\,000$	$Nu = 4.0 \times 10^{-4}Re^{1.22}(e/D)^{0.625}(s/10e)^{2.22}$ $\times \exp\{1.25[\ln(s/10e)]^2\}(l/10e)^{2.66}\exp\{0.824[\ln(l/10e)]^2\}$ $f = 0.815Re^{0.316}(l/e)^{0.266}(s/10e)^{0.19}Re^{-0.26}(10e/d)^{0.591}$

Table 6. Details of correlations developed (*continued*).

Author(s)	Novel geometry	Parameters	Correlation developed
Karwa et al. [158]	Machined ribs	$P/e = 4.5 - 8.5$ $e/D_h = 0.014 - 0.0320$ $d/w = 0.167 - 0.5$ $W/H = 5.87$ $Re = 3000 - 20\ 000$	For $7 \leq e^+ < 20$ $G = 103.77e^{-0.006}(W/H)^{0.5}(P/e)^{2.56} \exp[0.7343 \ln(P/e)^2] (e^+)^{-0.31}$ For $20 \leq e^+ \leq 60$ $G = 32.26e^{-0.006}(W/H)^{0.5}(+/e)^{2.56} \exp[0.7343 \ln(P/e)^2] (e^+)^{0.8}$ For $7 \leq e^+ < 20R$ $f = 1.66e^{-0.0078}(W/H)^{-0.4}(P/e)^{0.2695} \exp[-0.762 \ln(P/e)^2 (e^+)^{-0.075}]$ For $20 \leq e^+ \leq 60R$ $f = 1.325e^{-0.0078}(W/H)^{-0.4}(P/e)^{0.2695} \exp[-0.762 \ln(P/e)^2 (e^+)^{-0.075}]$
Bhagoria et al. [130]	Wedge shaped ribs	$60.17 \phi^{-1.0264} < P/e < 12.12$ $\phi = 8^\circ - 15^\circ$ $e/D_h = 0.015 - 0.033$ $Re = 3000 - 18000$	$Nu = 1.89 \times 10^{-4} Re^{1.21} (e/D_h)^{0.426} (P/e)^{2.94} (\phi/10)^{-0.018}$ $\times \exp\{-0.71[\ln(P/e)]^2\} \exp\{-1.50[\ln(\phi/10)]^2\}$ $f = 12.44 Re^{-0.18} (e/D_h)^{0.99} (P/e)^{-0.52} (\phi/10)^{0.49}$
Kumar et al. [100]	Forward facing chamfered rectangular ribs	$e/D = 0.018 - 0.043$ $P/e = 12$ $e/w = 1.5$ $e'/e = 0.75$ $Re = 4000 - 17\ 000$	$Nu = 0.066 Re^{0.858} (e'/e)^{0.022} (e/w)^{0.12} (e/D)^{0.241} \exp\{0.039[\ln(e'/e)]^2\}$ $f = 11.79 Re^{-0.58} (e'/e)^{0.169} (e/w)^{0.131} (e/D)^{0.318} \exp\{0.203[\ln(e'/e)]^2\}$
Inclined baffles			
Kumar et al. [137]	Angled perforated baffles	$W/H = 10$ $H_B/H = 0.50$ $P_B/H_B = 10$ $O_B/H_B = 0.266$ $\alpha = 35^\circ - 65^\circ$ $\beta_0 = 12\%$ $Re = 5000 - 17\ 000$	$Nu = 1.0826 \times 10^{-18} Re^{1.5805} 15.694 \alpha_a \exp[-1.9599(\ln \alpha_a)^2]$ $f = 2.48 \times 10^{-6} Re^{-0.137} 5.6517 \alpha_a \exp[-0.7113(\ln \alpha_a)^2]$
S-shaped ribs			
Kumar et al. [12]	S-shaped wire ribs	$P/e = 4 - 16$ $e/D_h = 0.022 - 0.054$ $\alpha = 30^\circ - 75^\circ$ $W/w = 1 - 4$ $Re = 2400 - 20\ 000$	$Nu = 1.4332 \times 10^{-4} Re^{1.2764} (W/w)^{0.2748}$ $\times \exp\{-0.1084[\ln(W/w)]^2\} (P/e)^{0.4876} \exp\{-0.1107[\ln(P/e)]^2\} (\alpha/90^\circ)^{-0.0468}$ $\times \exp\{-0.0642[\ln(P/e)]^2\} (e/D_h)^{-0.7653} \exp\{-0.1257[\ln(e/D_h)]^2\}$ $f = 1.430 \times 10^{-1} Re^{-0.224} (W/w)^{0.1424} (P/e)^{0.7657}$ $\times \exp\{-0.187[\ln(P/e)]^2\} (\alpha/90^\circ)^{0.2129} (e/D_h)^{0.2159}$
Other roughness geometries			
Saini and Verma [120]	Dimple protrusions	$e/D_h = 0.018 - 0.037$ $Re = 2000 - 12000$ $P/e = 8 - 12$	$Nu = 5.2 \times 10^{-4} Re^{1.27} (e/D)^{0.33} (P/e)^{3.15}$ $\times \exp\{-2.2[\log(P/e)]^2\} \exp\{-1.30[\log(e/D)]^2\}$ $f = 0.0642 Re^{-0.423} (e/D)^{-0.0214} (P/e)^{-0.465}$ $\times \exp\{0.054[\log(P/e)]^2\} \exp\{-0.840[\log(e/D)]^2\}$
Bhusan and Singh [122]	Dimple shaped roughness	$L/e = 25 - 37.5$ $S/e = 18.75 - 37.5$ $e/D = 0.03$ $d/D = 0.147 - 0.367$ $Re = 4000 - 20000$	$Nu = 2.1 \times 10^{-88} Re^{1.452} (S/e)^{12.94} (L/e)^{992} (d/D)^{-3.9}$ $\times \exp\{-10.4[\log(S/e)]^2\} \exp\{-77.2[\log(L/e)]^2\} \exp\{-7.83[\log(d/D)]^2\}$ $f = 2.32 Re^{-0.201} (S/e)^{-0.383} (L/e)^{-0.484} (d/D)^{0.133}$
Yadav et al. [106]	Circular protrusions	$P/e = 12 - 24$ $e/D = 0.015 - 0.03$ $W/H = 11$ $\alpha = 45^\circ - 75^\circ$ $Re = 3600 - 18100$	$Nu = 0.154 Re^{1.017} (P/e)^{-0.38} (e/D)^{0.521} (\alpha/60^\circ)^{-0.213}$ $\times \exp\{-2.0233[\ln(\alpha/60^\circ)]^2\}$ $f = 7.207 Re^{-0.56} (P/e)^{-0.18} (e/D)^{0.176} (\alpha/60^\circ)^{0.038}$ $\times \exp\{-1.412[\ln(\alpha/60^\circ)]^2\}$
Saravanan at al. [146]	Staggered C-shape finned	$V = 0.57 - 4.4\ m/s$ $P/g = 3.2 - 3.8$ $h/H = 0.3 - 0.7$ $h/d_p = 1.5 - 3$ $Re = 3000 - 24000$	$Nu_p = 0.527 Re^{0.85} (P/g)^{1.009} (h/H)^{1.076} (h/d_p)^{-2.484}$ $f_p = 0.21 Re^{-0.21} (P/g)^{-0.782} (h/H)^{-0.502} (h/d_p)^{1.084}$ $Nu_{np} = 0.009 Re^{0.861} (P/g)^{1.009} (h/H)^{0.482}$ $f_{np} = 1.126 Re^{-0.194} (P/g)^{-0.757} (h/H)^{0.593}$
Bezbaruah et al. [147]	Staggered conical vortex generators	$Re = 3500 - 16\ 000$ $e = 0.0075\ m$ $d = 0.015\ m$ $P = 0.020\ m$ $W = 0.08\ m$ $H = 0.03\ m$ $L_1 = 0.15\ m$ (inlet lenght) $L_2 = 1.1\ m$ (test section lenght) $L_3 = 0.2\ m$ (outlet lenght) $D = 0.044\ m$	$Nu = C_0 Re^{0.7209} (e/D)^{0.1718} (P/e)^{-0.127} \exp\{-0.0225[\ln(P/e)]^2\}$ $f = C_1 Re^{-0.4091} (e/D)^{0.4665} (P/e)^{-0.6123}$ $C_0 = 0.22805, C_1 = 24.5497$

Table 7. Details of enhancement of heat transfer and fluid friction with optimum parameters used in the literature.

Author(s)	Geometry configuration	Reynolds number	Optimum value	Enhancement	
				Nu/Nu _s	f/f _s
Transverse ribs					
Prasad and Saini [86]	Small diameter protrusion wire	1500–24 000	$P/e = 10$ e/D_h or $e/D = 0.027$	2.38	4.25
Karwa et al. [158]	Transverse ribs	2800–15 000	$P/e = 5.5$ e/D_h or $e/D = 0.0143$ $W/H = 7.75$ $\Phi = 15^\circ$	2.30	3.42
	Inclined ribs			2.12	3.92
	V continuous ribs			2.37	3.65
	V discrete ribs			2.47	2.47
Singh et al. [93]	Uniform/non-uniform square-wave profile transverse roughness rib	3000–15 000	$W_c/w = 85$	2.22	2.44
Sharma et al. [94]	Transverse, inclined transverse, dimple, inclined dimple, arc and sine wave	3000–18 000	For sine wave Re = 15000	3.60	–
Mahanand and Senapati [95]	Transverse inverted T-shaped ribs	3800–18 000	$e/D_h = 0.042$ $P/e = 7.14$ Re = 15 000	2.747	3.263
Gawande et al. [96]	Combined square and equilateral triangular ribs	3800–18 000	$e = 1.4$ $e/D = 0.042$ $P = 10$ $P/e = 7.14$ Re = 18 000	2.877	3.661
Kumar and Layek [98]	Circular ribs	8551–11 149	$P/e = 10$ $e = 1.2$	1.97	–
Kumar et al. [99]	Square ribs	4000–18 000	$P/e = 10$ $e/D = 0.05$ Re = 17 900	1.97	2.53
Kumar et al. [117]	Rectangular ribs roughened	4000–18 000	$P/e = 10$ $e/D = 0.04$ $e/w = 4.0$ Re = 15 000	2.70	4.93
Kumar et al. [152]	Semicircular ribs	4000–24 000	$e = 0.067$ $P/e = 25$ $P = 100$ Re = 24 000	2.20	2.50
Arc-shaped roughness					
Saini and Saini [102]	Arc-shaped wire ribs	2000–17 000	$P/e = 10$ e/D_h or $e/D = 0.0422$ $\alpha/90^\circ = 0.3333$	3.80	1.75
Kumar et al. [105]	Discrete multiple-arc ribs	3000–19 000	$W/W_{Ac} = 5.0$ $D_d/L_v = 0.67$ $g_w/H_r = 0.87$ $H_r/H = 0.047$ $P_r/H = 1.7$ $\alpha = 60^\circ$ $X/D_h = 1.72$ $Y/D_h = 0.82$	7.61	6.48
Yadav et al. [106]	Circular protrusions in arc shape	3600–18 100	$P/e = 12$ e/D_h or $e/D = 0.03$ $\alpha = 60^\circ$	2.89	2.93
Ghritlahre [107]	Arc-shaped wire ribs with apex upstream	No data*	$P/e = 10$ $e/D = 0.0395$ $e = 2.5$ $\alpha = 60^\circ$	2.56	–
	Arc-shaped wire ribs with apex downstream			1.81	–
Srivastava et al. [108]	Arc-shaped ribs with gap	3000–15 000	$P/e = 10$ $e/D = 0.043$ $g/e = 1$ $d/W = 0.67$ $\alpha = 60^\circ$ Re = 3000	2.40	3.94
	V-shaped ribs with gap		$P/e = 10$ $e/D = 0.043$ $g/e = 1$ $d/W = 0.67$ $\alpha = 60^\circ$ Re = 15 000	3.40	4.05

Table 7. Details of enhancement of heat transfer and fluid friction with optimum parameters used in the literature (continued).

Author(s)	Geometry configuration	Reynolds number	Optimum value	Enhancement	
				Nu/Nu _s	f/f _s
V-shaped ribs					
Singh et al. [113]	Discrete V down ribs	3000–15 000	$P/e = 10$ e/D_h or $e/D = 0.033$ $\alpha = 60^\circ$	3.04	3.11
Momin et al. [114]	V-shaped continuous wire ribs	17 034	$P/e = 10$ e/D_h or $e/D = 0.034$ $\alpha = 60^\circ$	2.30	2.83
Hans et al. [167]	Multiple V-shaped ribs	2000–20 000	$P/e = 8$ e/D_h or $e/D = 0.03$ $\alpha = 60^\circ$ $W/w = 10$	6.00	5.00
Singh et al. [168]	Discrete V-shaped ribs with gap	3000–15 000	$P/e = 8$ $e/D = 0.043$ $\alpha = 60^\circ$ $W/w = 12$	–	–
Kumar et al. [117]	Multiple V-shaped ribs with gap	2000–20 000	$P/e = 10$ $e/D = 0.043$ $\alpha = 60^\circ$ $G_d/L_v = 0.69$ $g/e = 1$	6.32	6.12
Kumar et al. [169]	Multiple V-shaped ribs with gap	2000–20 000	$P/e = 10$ $e/D = 0.043$ $\alpha = 60^\circ$ $G_d/L_v = 0.69$ $g/e = 1$ $W/w = 6$	6.74	6.37
Dimple or protrusion ribs					
Saini and Verma [120]	Dimple shape	2000–12 000	$P/e = 10$ e/D_h or $e/D = 0.0379$	7.58	4.68
Kumar and Verma [145]	Sinusoidal ribs	4000–15 000	$D_h = 33.33$ mm $P = 10, 14, 18$ mm $e = 1$ mm $W/H = 5$ $q = 1000$ W/m ² Re = 4000 – 15 000	2.48	1.76
Bhushan and Singh [170]	Protrusion	1900–13 000	$S/e = 31.25$ $L/e = 31.25$ $d/D = 0.294$	3.80	2.20
Hassan et al. [123]	Dimple	2000–18 000		5.56	1.62
Vikash Kumar [119]	Concave dimple	2000–13 500	$P/e = 8 - 15$ $e/D_h = 0.018 - 0.045$ $e/d = 1 - 2$	3.94	2.56
Tabish and Kim [124]	Conical protrusion rib over spherical ribs	4000–16 000	$P/e = 10$ $e/D = 0.044$	2.49	7.29
W-shaped ribs					
Kumar et al. [126]	Discritised W-shaped ribs	3000–15 000	$P/e = 10$ e/D_h or $e/D = 0.038$ $\alpha = 60^\circ$	2.16	2.75
Lanjewar et al. [171]	W-shaped ribs	2300–14 000	$P/e = 10$ $e/D = 0.0375$ $\alpha = 60^\circ$ $W/H = 8$	2.36	2.01
Thakur and Thakur [128]	Multi-staggered rib parameters of the W-rib roughness	4000–14 000	$P/e = 12$ $e/D_h = 0.0423$ $\alpha = 60^\circ$ $g/e = 1$ $n = 2$ $p/P = 0.65$ $w/e = 4.5$	4.10	2.50
Spherical ball roughness					
Murmu et al. [156]	Inclined spherical ball	2500–18 500	$p/e = 15$ $e/D_h = 0.036$ $e/d_b = 1$ $\alpha = 55^\circ$	3.56	2.34

Table 7. Details of enhancement of heat transfer and fluid friction with optimum parameters used in the literature (continued).

Author(s)	Geometry configuration	Reynolds number	Optimum value	Enhancement	
				Nu/Nu _s	f/f _s
Non-uniform ribs					
Layek et al. [89]	Chamfered compound rib	3000–21 000	$P/e = 10$ e/D_h or $e/D = 0.03$ $\phi = 18^\circ$	2.60	3.35
Kumar et al. [100]	Forward facing chamfered rectangular ribs	4000–17 000	$e/D = 0.043$ $P/e = 12$ $e/w = 1.5$ $e'/e = 0.75$ Re = 17000	2.88	3.52
Bhagoria et al. [130]	Wedge shaped ribs	3000–18 000	$P/e = 7.57$ e/D_h or $e/D = 0.033$ $\phi = 10^\circ$	2.40	5.30
Barik et al. [131]	T-rib, left arm inclined T-rib, right arm inclined T-rib, Y-rib, concave and convex T-ribs	5000–20 000	$P = 80$ mm Re = 20000	1.91	–
Kumar and Goel [132]	Semi-circular, circular, triangular, square, rectangular	3700–18 700	For a forward-chamfered rectangular rib: $e/D = 0.0433$ $e/w = 1.67, 2$ $e'/e = 0.75$ $P/e = 11$ Re = 3700 – 18700	3.82	–
Zina et al. [133]	Triangular ribs with semi-circular nooks	3800–18 000	Re = 5000 $P = 10$ mm $e = 2.2, 1.8, 1.4$ mm	2.55	2.42
Inclined baffles					
Aharwal et al. [135]	Inclined continuous with gap ribs	3000–18 000	$P/e = 10$ e/D_h or $e/D = 0.0377$ $\alpha = 60^\circ$ $W/H = 5.84$ $g/e = 1$ $d/W = 0.25$	2.95	2.87
Aharwal et al. [136]	Integral inclined discrete rib	3000–18 000	$P/e = 8$ e/D_h or $e/D = 0.037$ $\alpha = 60^\circ$ $d/W = 0.25$ $g/e = 1$	2.83	3.60
Kumar et al. [137]	Angled perforated baffles	5000–17 000	$\alpha = 55^\circ$ Re = 9000	3.82	7.14
Kumar and Layek [138]	Winglet shaped ribs	No data**	$P/e = 8$ $\alpha = 60^\circ$ $W/w = 5$ Re = 22 000	2.83	3.63
Other roughness geometries					
Saini and Saini [142]	Expanded metal mesh	1900–13 000	e/D_h or $e/D = 0.037$ $\alpha = 60^\circ$ $L/e = 46.87$ $S/e = 25$ $W/H = 11$	4.00	5.00
Chandra et al. [172]	One ribbed wall	10 000–80 000	$P/e = 8$ e/D_h or $e/D = 0.0625$ $L/D_h = 20$	2.43	3.14
	Two ribbed walls			2.64	5.39
	Three ribbed walls			2.81	3.14
	Four ribbed walls			2.99	9.50
Jaurker et al. [141]	Rib grooved combination	3000–21 000	$P/e = 6$ e/D_h or $e/D = 0.0363$ $g/p = 0.4$	2.75	3.61
Bopche and Tandale [144]	U-shaped ribs	3800–18 000	$P/e = 6.67$ $e/D = 0.03986$ $\alpha = 90^\circ$	2.82	3.72
Saravanan at al. [146]	Staggered C-shape finned	3000–24 000	$V = 0.57 - 4.4$ m/s $P/g = 3.2 - 3.8$ $h/H = 0.3 - 0.7$ $h/d_p = 1.5 - 3$ Re = 3000 – 24 000	2.67	5.34
Bezbaruah et al. [147]	Staggered conical vortex generators	3500–16 000	$P/e = 2.67$ $e/D = 0.34$ Re = 16 000	2.57	–

Table 7. Details of enhancement of heat transfer and fluid friction with optimum parameters used in the literature (*continued*).

Author(s)	Geometry configuration	Reynolds number	Optimum value	Enhancement	
				Nu/Nu _s	<i>f</i> / <i>f</i> _s
Goel et al. [148]	Hemispherical cavities	2100–18 500	$x/r_d = 11$ $y/r_d = 11$ $r_d/D_h = 0.039$ Re = 2160	5.33	3.60
Yadav and Saini [149]	Impinging jet	3500–17 500	$D_j/D_h = 0.065 - 0.195$ $H_j/D_h = 0 - 0.433$ Re = 3500 – 17 500	7.18	8.48
Wang et al. [150]	S-shaped ribs with gap	2300–23 000	Re = 19258 $P/e = 20$ $W/w = 4$ $e/D = 0.023 - 0.036$ $g/e = 1.5$ $q = 450 - 650 \text{ W/m}^2$	5.42	5.87
Gawande et al. [134]	20° chamfered ribs	3800–18 000	$P/e = 7.14$ $e/D = 0.042$ $\alpha = 0^\circ - 40^\circ$	3.218	4.04
	Right-angle triangular ribs		$W/H = 5$ Re = 15 000	3.100	3.80
	Reverse L-shaped ribs			2.827	3.42

*Mass flow rate $\dot{m} = 0.007\text{--}0.022 \text{ kg/s}$.
**Mass flow rate $\dot{m} = 0.00531\text{--}0.04041 \text{ kg/s}$.

10. Thermo-hydraulic performance

The performance of a solar air heater can be evaluated using artificial roughness on the underside of the absorber plate surface. The use of artificial roughness increases heat transfer, as well as causes a pressure drop due to friction. The performance of SAH can be enhanced when heat transfer increases simultaneously, allowing a minimum possible friction factor.

The overall performance of SAH is judged by thermo-hydraulic performance as $THP = (Nu/Nu_s)/(f/f_s)^{1/3}$. The term ‘thermo-hydraulic performance’ was first used by Webb and Eckert [173], who considered parameters, such as heat transfer and the friction factor caused by fluid pressure drop.

In this paper, various research works are referred to in order to get an overview of the THP value determined based on experimental studies conducted by researchers. It is found that the value of THP ranges from 1.38 to 4.73 under various arrangements utilised over the absorber plate and environmental conditions, as enlisted in Table 8.

11. Discussion

11.1. Correlations

In the field of correlation development, Prasad and Saini [86] used P/e , e/D and Re as influencing parameters to modify conventional SAH, and they developed correlations for Nu and f . Similarly, Hong and Hsieh [162], Verma and Prasad [91], Kumar et al. [99] and Varun et al. [165] developed correlations for Nu and f for the performance prediction of SAH. There are many other authors who have been associated in the same field of interest and developed correlations for Nu and f . The parameters considered were: e/D , w/H and Re used by Gupta et al. [151], w/H , P/e , e^+ and e used by Karwa [88], P/e , e/D , g/P and Re

adopted by Layek et al. [89], P/e and Re taken by Varun et al. [165], Mahanand and Senapati [95] and Gawande et al. [134], d/H and D_h utilised by Kumar et al. [152], P/e , e/w , e/D and Re used by Kumar et al.[153], e/D , d/w , g/e and Re adopted by Aharwal et al. [136] for developing correlation and modification of conventional SAH.

The correlations for Nu and f using arc-shaped rib roughness were first developed by Saini and Saini [102] with e/D , α , Re as variable parameters in a modified SAH. Bhuvad et al. [110] used f , e/D and Re for apex-up discrete arc rib roughness; Kumar et al. [105] used Re, D_d/L_v , g_w/H_r , H_r/H , R_r/H , α , X/D_h and Y/D_h parameters for discrete multiple-arc rib roughness; Kumar et al. [112] used g/e , d/x , Re and N_g parameters for correlation development of Nu and f for discrete multiple arc-shaped ribs. Kumar et al. [161] arranged arcs in forward and backwards facing, using parameters such as Re, g/e , d/x , W/w , P/e , e/D and g/e for correlation development, whereas Korpale et al. [111] developed correlations using Re, P/e , e/D and w/e as roughness factors for a rectangular-section rib configuration.

Gupta et al. [163] firstly reported correlations for Nu and f based on V-shaped wire ribs with and without inclination angle through e/D , W/H , α and K/D parameters. Muluwork et al. [166] additionally used the B/S ratio to develop the correlation. Momin et al. [114] arranged wire ribs in a continuous manner to develop correlations by using e/D , α and Re as governing parameters. Hans et al [167] used multiple V-shaped ribs to modify the correlation with parameters such as e/D , W/w , Re, P/e and α . Singh et al. [116] used discrete V-shaped ribs with parameters such as Re, P/e , α , g/e , e/D , d/w to develop correlations for Nu and f . Kumar et al. [169] developed correlations for Nu and f using Re, e/D , W/w , G_d/L_v , g/e , α , P/e for multiple V-shaped ribs with a gap.

Table 8. Details of thermo-hydraulic performance of solar air heaters reported to date.

Authors	Roughness type	Roughness parameters	$THP = \frac{Nu/Nu_s}{(f/f_s)^{1/3}}$
Tranverse ribs			
Prasad and Saini [86]	Transverse wire	$e/D_h = 0.020 - 0.033$ $P/e = 10 - 20$ Re = 500 – 50 000	1.78
Bhagoria et al. [130]	Transverse wedge shaped ribs	$60.17\Phi^{-1.0264} < P/e < 12.12$ $\Phi = 8^\circ - 15^\circ$ $e/D = 0.015 - 0.033$ Re = 3000 – 18000	1.38
Karwa et al. [88]	Transverse ribs	$P/e = 5.5$	1.53
	Inclined ribs	e/D_h or $e/D = 0.0143$	1.34
	V continuous ribs	$W/H = 7.75$	1.54
	V discrete ribs	$\alpha = 15^\circ$	1.83
Singh et al. [93]	Uniform/non-uniform square-wave profile transverse roughness ribs	Re = 3000 – 15 000	1.49
Sharma et al. [94]	Transverse, inclined transverse, dimple, inclined dimple, arc and sine wave	Sine wave $e/D_h = 0.271$ $P/e = 10$ Re = 15 000 $\alpha = 45^\circ$	2.05
Mahanand and Senapati [95]	Transverse inverted T-shaped ribs	$e/D_h = 0.042$ $P/e = 7.14$ Re = 15 000	1.86
Gawande et al. [96]	Combined square and equilateral triangular ribs	$e = 1.4 \text{ mm}$ $e/D = 0.042$ $P = 10 \text{ mm}$ $P/e = 7.14$ Re = 18 000	1.86
Korpale et al. [111]	Rectangular section ribs	$P/e = 17.22$ $e/D = 0.044$ $w/e = 0.5$ Re = 20 000	2.77
Kumar et al. [99]	Square ribs	$P/e = 10$ $e/D = 0.05$ Re = 17 900	1.44
Kumar et al. [153]	Rectangular ribs roughened	$P/e = 10$ $e/D = 0.04$ $e/w = 4.0$ Re = 15 000	1.89
Kumar et al. [152]	Semicircular ribs	$e = 0.067 \text{ mm}$ $P/e = 25$ $P = 100 \text{ mm}$ Re = 24 000	1.7
Arc-shaped roughness			
Bhuvad et al. [110]	Apex-up discrete arc ribs	$W = 200 \text{ mm}$ $W/H = 8$ $\alpha = 30^\circ, 45^\circ, 60^\circ$ $e/D_h = 0.045$ $P/e = 10$ $N_g = 3$ $g/e = 3$	2.01
Kumar et al. [105]	Discrete multiple-arc ribs	$D_d/L_v = 0.27 - 0.86$ $g/H_r = 0.32 - 1.72$ $H_r/H = 0.025 - 0.047$ $P_r/H = 0.58 - 3.1$ $\alpha = 35^\circ - 65^\circ$ $X/D_h = 0.47 - 1.72$ $Y/D_h = 0.46 - 0.82$ $D_j/D_h = 0.065$ $W/W_{AC} = 5$ Re = 3000 – 19 000	4.1
Ghritlahre [107]	Arc-shaped wire ribs with apex upstream	$\dot{m} = 0.007 - 0.022 \text{ kg/s}$	2.09
	Arc-shaped wire ribs with apex downstream		1.59

Table 8. Details of thermo-hydraulic performance of solar air heaters reported to date (continued).

Authors	Roughness type	Roughness parameters	$THP = \frac{Nu/Nu_s}{(f/f_s)^{1/3}}$
Kumar et al. [161]	Innovatively arranged arc (combination of alternatively arranged forward and back-ward facing arced ribs)	$e/D = 0.045$ $\alpha = 60^\circ$ $d/x = 0.65$ $g/e = 1.0$ $W/w = 3$ $P/e = 8$	3.61
Kumar et al. [112]	Discrete multiple arc-shaped ribs	$g/e = 1$ $N_g = 3$ $d/x = 0.6$ $Re = 19000$	3.85
Srivastava et al. [174]	Arc-shaped rib with gap	$P/e=10$ $e/D = 0.043$ $g/e = 1$	1.5
	V-shaped rib with gap	$d/W = 0.67$ $\alpha = 60^\circ$ $Re = 3000$	2.21
V-shaped ribs			
Singh et al. [113]	V down ribs	$P/e = 10$ e/D_h or $e/D = 0.033$ $\alpha = 61.9^\circ$	2.03
Momin et al. [114]	V-shaped continuous wire rib	$P/e = 10$ e/D_h or $e/D = 0.034$ $\alpha = 60^\circ$	1.63
Hans et al. [167]	Multiple V-shaped ribs	$P/e = 8$ e/D_h or $e/D = 0.03$ $W/w = 10$ $\alpha = 60^\circ$	3.51
Singh et al. [115]	Discrete V-shaped ribs with gap	$P/e = 8$ $e/D = 0.043$ $\alpha = 60^\circ$ $W/H = 12$	2.06
Kumar et al. [175]	Multiple V-shaped ribs with gap	$P/e = 10$ $e/D = 0.043$ $G_d/L_v = 0.69$ $g/e = 1.0$ $\alpha = 60^\circ$	3.46
Kumar et al. [117]	Multiple V-shaped ribs with gap	$P/e = 10$ e/D_h or $e/D = 0.043$ $g/e = 1$ $G_d/L_v = 0.69$ $W/w = 6$ $\alpha = 60^\circ$	3.64
Dimple or protrusion ribs			
Saini and Verma [120]	Dimple shape	$P/e = 10$ e/D_h or $e/D = 0.0379$	4.53
Kumar and Verma [145]	Sinusoidal rib	$D_h = 33.33$ mm $P = 10, 14, 18$ mm $e = 1$ mm $W/H = 5$ $q = 1000$ W/m ² $Re = 4000 - 15\,000$	2.05
Bhushan and Singh [170]	Protrusion	$S/e = 31.25$ $L/e = 31.25$ $d/D = 0.294$	2.92
Hassan et al. [123]	Dimple	$P/e = 4 - 16$ $e/D_h = 0.018 - 0.045$ $W/w = 1 - 5$ $\alpha = 30^\circ - 75^\circ$	4.73
Kumar and Prasad [119]	Concave dimple	$P/e = 4 - 15$ $e/D_h = 0.018 - 0.045$ $e/d_d = 1 - 2$	2.88
Alam and Kim [176]	Conical protrusion rib over spherical ribs	$P/e = 10$ $e/D = 0.044$	1.28
W-shaped ribs			
Kumar et al. [126]	Discritised W-shaped ribs	$P/e = 10$ e/D_h or $e/D = 0.0338$ $\alpha = 60^\circ$	1.54

Table 8. Details of thermo-hydraulic performance of solar air heaters reported to date (continued).

Authors	Roughness type	Roughness parameters	$THP = \frac{Nu/Nu_s}{(f/f_s)^{1/3}}$
Lanjewar et al. [171]	W-down ribs	$e/D_h = 0.03375$ $P/e = 10$ $e = 1.5$ mm $\alpha = 30^\circ, 45^\circ, 60^\circ, 75^\circ$	1.98
	W-up ribs	$W/H = 8$ $e^+ = 8 - 44$ $Re = 2300 - 14\,000$	1.81
Thakur and Thakur [127]	W-shaped ribs with symmetrical gaps along with staggered ribs	$P/e = 12$ $g/e = 1$ $w/e = 4.5$ $p/P = 0.65$ $N_g = 2$ $Re = 12\,000$	3.24
Thakur and Thakur [128]	Multiple-staggered rib parameters of the W'-shaped roughness	$P/e = 12$ $e/D_h = 0.0423$ $\alpha = 60^\circ$ $g/e = 1$ $w/e = 4.5$ $p/P = 0.65$ $N_g = 2$	3.02
Spherical ball roughness			
Murmu et al. [156]	Inclined spherical ball	$P/e = 15$ $e/D_h = 0.036$ $e/d_b = 1$ $\alpha = 55^\circ$	2.68
Non-uniform ribs			
Layek et al. [89]	Chamfered compound rib	$P/e = 10$ e/D_h or $e/D = 0.03$ $\Phi = 18^\circ$	1.74
Kumar et al. [100]	Forward-facing chamfered rectangular ribs	$e/D = 0.043$ $P/e = 12$ $e/w = 1.5$ $e'/e = 0.75$ $Re = 17\,000$	1.89
Bhagoria et al. [130]	Wedge shaped ribs	$P/e = 7.57$ e/D_h or $e/D = 0.033$ $\Phi = 10^\circ$	1.38
Barik et al. [131]	T-shaped rib, left arm inclined T-shaped rib, right arm inclined T-shaped rib, Y-shaped rib, concave and convex T-shaped ribs	For T-shaped rib $P = 80$ mm $Re = 18\,000$	1.7
Kumar and Goel [132]	Semi-circular, circular, triangular, square, rectangular	For forward-chamfered rectangular rib $e/D = 0.0433$ $e/w = 0.67, 2.0$ $P/e = 11$ $e'/e = 0.75$ $Re = 3700 - 18\,700$	2.66
Zina et al. [133]	Triangular ribs with semi-circular nooks	$P = 10$ $e = 2.2, 1.8, 1.4$ mm $Re = 5000$	1.93
Gawande et al. [134]	20° chamfered rib	$P/e = 7.14$	2.05
	Right-angle triangular rib	$e/D = 0.042$	2.04
	Reverse L-shaped rib	$W/H = 5$ $Re = 15\,000$	1.90
Inclined baffles			
Kumar et al. [137]	Angled perforated baffles	$W/H = 10$ $H_B/H = 0.50$ $P_B/H_B = 10$ $O_B/H_B = 0.266$ $\alpha = 35^\circ - 65^\circ$ $\beta_0 = 12\%$ $Re = 5000 - 17\,000$	1.94
Kottayat et al. [139]	Discrete multiple inclined baffle	$Re = 6000 - 14000$	2.69

Table 8. Details of thermo-hydraulic performance of solar air heaters reported to date (continued).

Authors	Roughness type	Roughness parameters	THP = $\frac{Nu/Nu_s}{(f/f_s)^{1/3}}$
Parsa et al. [140]	Staggered cuboid baffles	$a/H = 2.5$ (relative length) $b/H = 0.8$ (relative width) $c/H = 0.7$ (relative height) $s/H = 1.5$ (relative pitch) $\alpha = 22.5^\circ$ $Re = 5080$	3.28
Aharwal et al. [135]	Inclined and continuous rib with gap	$P/e = 10$ e/D_h or $e/D = 0.0377$ $\alpha = 60^\circ$ $W/H = 5.84$ $g/e = 1$ $d/W = 0.25$	2.08
Aharwal et al. [136]	Integral inclined discrete ribs	$P/e = 8$ e/D_h or $e/D = 0.037$ $\alpha = 60^\circ$ $d/W = 0.25$ $g/e = 1$	1.85
Kumar and Layek [138]	Winglet shaped ribs	$P/e = 8$ $W/w = 5$ $\alpha = 60^\circ$ $Re = 22\,000$	1.84
Other roughness geometries			
Jaurker et al. [141]	Rib groove	$P/e = 4.5 - 10.0$ $g/p = 0.3 - 0.7$ $e/D = 0.0181 - 0.0363$ $Re = 3000 - 21000$	1.76
Saini and Saini [90]	Expanded metal mesh	$l/e = 25 - 71.87$ $s/e = 15.62 - 46.87$ $e/D_h = 0.012 - 0.0390$ $Re = 1900 - 13\,000$	2.34
Karmare and Tikekar [143]	Metal grit ribs	$e/D_h = 0.035 - 0.044$ $P/e = 12.5 - 36$ $l/s = 1.72 - 1$ $Re = 4000 - 17000$	2.39
Bopche and Tandale [144]	Inverted U-shaped ribs	$e/D_h = 0.0186 - 0.03986$ $p/e = 6.667 - 57.14$ $\alpha = 90^\circ$ $W/H = 6$ $Re = 3800 - 18\,000$	1.82
Kumar and Verma [145]	Sinusoidal protrusion ribs	$P_r = 10, 14, 18$ mm (transverse) $P_L = 10, 14, 18$ mm (longitudinal) $e = 1$ mm $W/H = 5$ $D_h = 33.33$ mm $Re = 4000 - 15\,000$	2.02
Saravanan et al. [146]	Staggered C-shape finned absorber plate	$P/g = 3.8$ $h/H = 0.6$ $Re = 24\,000$	1.64
Bezbaruah et al. [147]	Staggered conical vortex generators	$P/e = 2.67$ $e/D = 0.17$ $Re = 16\,000$	2.04
Goel et al. [148]	Hemispherical cavities	$x/r_d = 11$ $y/r_d = 11$ $r_d/D = 0.039$ $Re = 2160$	3.48
Yadav and Saini [149]	Impinging jet	$D_j/D_h = 0.065 - 0.195$ $H_j/D_h = 0 - 0.433$ $Re = 3500 - 17\,500$	3.66

Table 8. Details of thermo-hydraulic performance of solar air heaters reported to date (continued).

Authors	Roughness type	Roughness parameters	THP = $\frac{Nu/Nu_s}{(f/f_s)^{1/3}}$
Kumar and Verma [121]	Sinusoidal ribs	$D_h = 33.33$ mm $P = 10, 14, 18$ mm $e = 1$ mm $W/H = 5$ $q = 1000$ W/m ² $Re = 4000 - 15\,000$	2.05
Wang et al. [150]	S-shaped ribs with gap	$p/e = 20$ $W/w = 4$ $e/D = 0.023 - 0.036$ $g/e = 1.5$ $q = 450 - 650$ W/m ² $Re = 19\,258$	3.00

Karmare and Tikekar [143] modified SAH using wire ribs grid shape roughness, considering parameters such as e/D , L/S , Re and P/e for determining Nu and f . Saini and Saini [142] used expanded metal mesh with the following parameters: Re , e/D , S/e and L/e . Karwa et al. [158] adopted w/H , P/e and e^+ parameters for machined ribs for modification of a SAH. Bhagoria et al. [130] modified SAH using wedge shaped ribs for parameters such as Re , e/D , P/e and ϕ . Kumar et al. [100] developed correlations of Nu and f for forward-facing chamfered rectangular ribs through e'/e , e/w , e/D and Re as parameters.

Kumar et al. [137] used angled perforated baffles for modification of SAH through Re and α as parameters.

Saini and Verma [119] used dimple protrusions using parameters such as Re , e/D and P/e . Bhushan and Singh [170] used dimple-shaped roughness with Re , S/e , L/e and d/D as governing parameters. Yadav et al. [106] used circular protrusions and adopted Re , P/e , e/D and α as influencing parameters for modification of SAH. Saravaran et al. [146] utilised staggered C-shape finned absorber plate with factors affecting the performance of a SAH, such as P/g , h/H and h/d_p as parameters.

Murmu et al. [156] modified the conventional SAH using spherical ball-shaped ribs over the absorber plate and successfully developed strong predictive correlations for Nu and f . They used Re , e/D , P/e , e/d and α as control variables.

No reports on the development of correlations for W-shaped ribs to date have been reported by any author.

11.2. Maximum enhancement of Nusselt number and friction factor

This section is dedicated to disseminating the maximum values of Nu and f for roughness geometries considered in the present review work.

Gawande et al. [96] investigated transverse ribs in the form of combined square and equilateral triangular ribs as roughness

elements to modify conventional SAH. They found the maximum enhancement in the Nusselt number as 2.877 and the friction factor as 3.661 compared to a smooth duct of SAH. They achieved maximum Nu and f at the optimum control parameters of $P/e = 7.14$ (fixed), $e/D = 0.042$ (fixed) and $Re = 18\,000$, resulting from combining square and equilateral triangular shapes that enhanced the heat transfer.

Kumar et al. [105] modified conventional SAH with novel roughness elements in the form of discrete multiple arc-shaped ribs in the view of enhancing heat transfer. They reported the highest increment in Nu and f of 7.61 and 6.48, respectively, in a modified SAH with the optimum parameters of $W/W_{AC} = 5.0$, $D_a/L_v = 0.67$, $g_w/H_r = 0.87$, $H_r/H = 0.047$, $P_r/H = 1.7$, $\alpha = 60^\circ$, $X/D_h = 1.72$ and $Y/D_h = 0.82$. The maximum enhancement was ensured due to gaps in the multi-arc shaped ribs, which ensured the frontal discharge of flowing fluid through both primary flow and secondary flow. This further contributes to better mixing, i.e. limited surface side growth of the boundary layer, which leads to an enhancement of heat transfer.

Kumar et al. [117] used multi V-shaped ribs with a gap as roughness elements to extend the absorber plate in the expectation of improved SAH with higher heat transfer rates. They found maximum augmentation in Nu and f of 6.74 and 6.37, respectively, at the optimum variables of $P/e = 10$, e/D_h or $e/D = 0.043$, $\alpha = 60^\circ$, $g/e = 1$, $G_d/L_v = 0.69$, $W/w = 6$ and Re of 20 000. The gap distance used in the roughness geometry provided passage for secondary flow, which enhanced the heat transfer between the fluid and the surface.

Saini and Verma [120] employed a novel artificial roughness element in the form of dimple-shaped rib over the absorber plate in a SAH. They achieved maximum enhancement in Nu and f of 7.58 and 4.68, respectively, at the optimum influencing parameters, such as P/e of 10, e/D of 0.0379 and Re of 12 000. The dimple shape of roughness elements led to the reattachment of the free shear layer after the flow separation in the fluid domain.

This enhances the heat transfer rate between the fluid and the surface by virtue of the vicinity of the reattachment region.

Thakur and Thakur [128] proposed a modified SAH featuring a novel roughness geometry based on multi-staggered W-shaped ribs. They conducted experiments by varying different operating and geometric parameters, and reported the results at the optimum conditions of $P/e = 12$, $e/D_h = 0.0423$, $\alpha = 60^\circ$, $g/e = 1$, $N_g = 2$, $p/P = 0.65$, $w/e = 4.5$ and $Re = 12\,000$. They reported an enhancement in Nu and f of 4.1 and 2.5, respectively. It was observed that W-shaped staggered ribs further energise the flow within the gaps, promoting enhanced flow development and vortex formation, which in turn significantly improves the Nusselt number.

Murmu et al. [156] experimentally investigated SAH modified using inclined spherical balls as roughness elements. They reported an enhancement in Nu and f by 3.56 and 2.34, respectively, compared to a smooth duct SAH at the optimum parameters of $P/e = 15$, $e/D_h = 0.036$, $e/d_b = 1$ and $\alpha = 55^\circ$. They concluded that due to the spherical ball shape roughness, the secondary flow stream rejoins the primary flow, which accelerates the rate of heat transfer.

Kumar and Goel [132] used a non-uniform type of rib roughness specifically in the form of forward-chamfered rectangular rib to the developed modified SAH. They achieved maximum increases in Nu and f as 3.82 and 2.68, respectively, at the optimum parameters such as the e/D ratio of 0.0433, e/w ratio of 0.67 and 2.0, P/e ratio of 11, e/e ratio of 0.75 and Re of 3700 to 18 700. The authors observed that the main reason for heat transfer increment is that the chamfered elements guide the fluid flow in the centre of SAH, which leads to better mixing of a broken laminar sublayer over the absorber plate to the mainstream flow.

Kumar et al. [137] investigated experimentally the modified SAH using angled perforated baffles as novel baffle roughness in the view of heat transfer enhancement and controlled fluid flow. They optimised the system and operating parameters to $\alpha = 55^\circ$ and $Re = 9000$. The reported maximum enhancement in Nu and f was 3.82 and 7.14, respectively.

Wang et al. [150] modified conventional SAH using S-shaped ribs with a gap as artificial roughness over the absorber plate, focusing on the heat transfer enhancement. The maximum obtained enhancement in Nu and f was, respectively, 5.42 and 5.87 times the smooth duct SAH at the optimum variables, such as Re of 19 258, P/e ratio of 20, W/w ratio of 4, e/D ratio of 0.023 to 0.036 and g/e ratio of 1.5. Due to S-shaped ribs over the absorber plate, the incoming fluid stream underwent the flow separation, causing vortices to form downstream of the flow. These vortices improved air mixing, thereby enhancing the heat transfer and increasing pressure drop.

11.3. Thermo-hydraulic performance

As regards the thermo-hydraulic performance of the solar air heater geometries based on various transverse rectangular sectioned ribs, the maximum value obtained was 2.77, as shown in the comparison presented in Fig. 70. This SAH model was developed by Korpale et al. [111], who incorporated transverse rectangular finned types of ribs to roughen the solar air heaters. The rectangular sectioned fins installed on the absorber surface

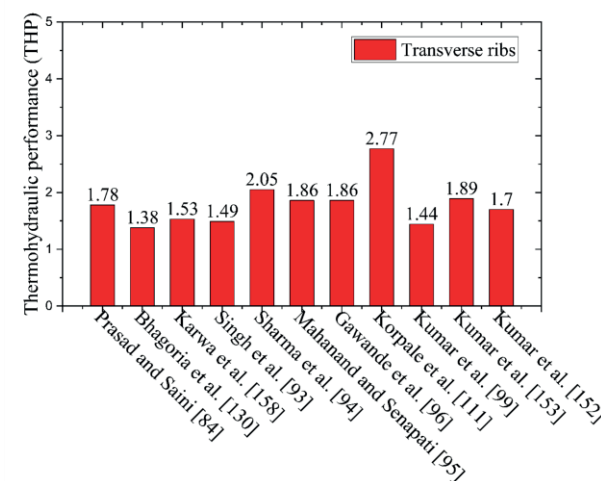


Fig. 70. Variation in THP of SAH based on various transverse ribs.

helped in creating wall-induced turbulence affecting the temperature of the air, as well as the local temperature of walls in contact with hot air. The THP value obtained from the CFD approach was 2.37, showing minor error for Nu and f calculations. The validation experiment was also performed to confirm the accuracy of optimisation and CFD results. The authors obtained a THP value of 2.55 for the same case, which was a maximum value among all other models achieved in the transverse rib SAH type.

There have been various investigations carried out on SAHs based on the arc-shaped roughness. The maximum value of THP was found to be 4.1. It was obtained by Kumar et al. [105], who developed an impingement jet SAH with discrete multiple arc-shaped ribs as roughness elements. The gaps in the multiple arc-shaped ribs permit the discharge of fluid from the secondary and mainstream. The mainstream flowing through the gap is a stable stream with a denser boundary layer containing the viscous sublayer. The occurrence of a gap results in the mixing of a secondary stream with the mainstream. This mixing facilitates and strengthens the limited boundary layer stream beside the surface, resulting in increased heat transfer. This leads to an improved heat transfer rate. However, it is accompanied by a simultaneous enhancement in Reynolds number and a relatively small increase in the friction factor for the augmented SAH. Therefore, the thermal hydraulic performance (THP) was found to be superior compared to other available models, as shown in Fig. 71.

The maximum THP value of 4.73 was obtained in the group of SAHs utilising dimple or protrusion ribs, as depicted in Fig. 72. The SAH setup, which produced the maximum THP, was built by Hassan et al. [123], who utilised multiple arc arrangements using dimple or protrusion ribs to develop an efficient SAH setup. Due to the presence of multiple arc arrangements, the highest number of flow reattachment points was formed. Two phenomena worked here to enhance the heat transfer. Firstly, the destruction of the laminar sublayer due to the flow stream and secondly, the generation, propagation and mixing of accelerating secondary flow with the primary flow. The

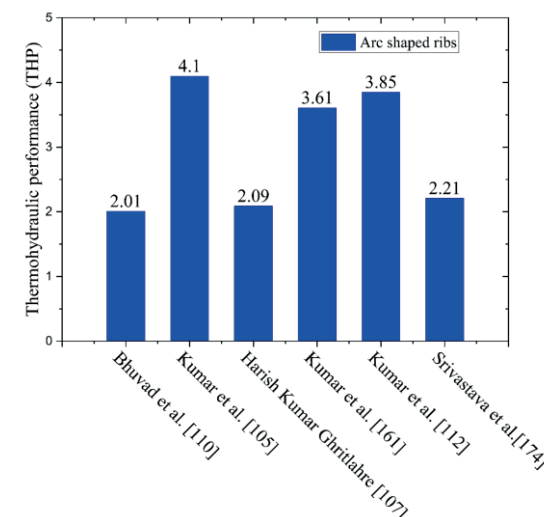


Fig. 71. Variation in THP of SAH based on various arc-shaped roughness.

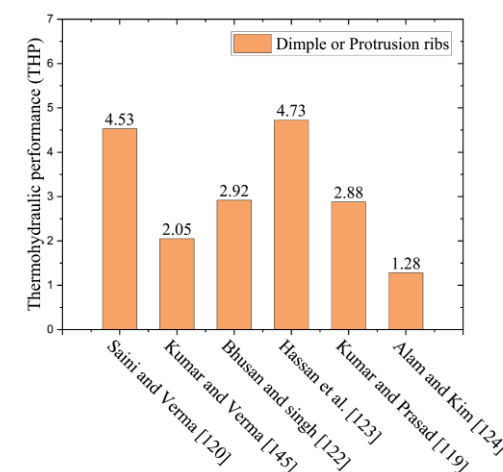


Fig. 72. Variation in THP of SAH based on various dimple or protrusion roughness.

secondary flow generated due to the dimple shape of ribs compels the less heated portion of air to get mixed with the primary highly heated air under the absorber plate, and then propagates to the trailing edge with much higher temperature. These two streams enhanced the destruction of the laminar sublayer, ultimately promoting higher heat transfer between air and the absorber.

Figure 73 shows the comparison among all reported V-shaped ribs roughened SAHs, in which Kumar et al. [117] obtained the maximum THP value of 3.64. They developed an experimental setup of SAH using multiple V-shaped ribs with gaps. The absorber plate was single-sided roughened with varying pitch that played a vital role in enhancing heat transfer, ensuring availability of lesser or greater number of reattachment points. The optimum pitch obtained was 10, at which the highest number of available reattachment points was observed, and fewer reattachment points were observed below and above this value, yielding the highest heat transfer and the lowest heat transfer, respectively.

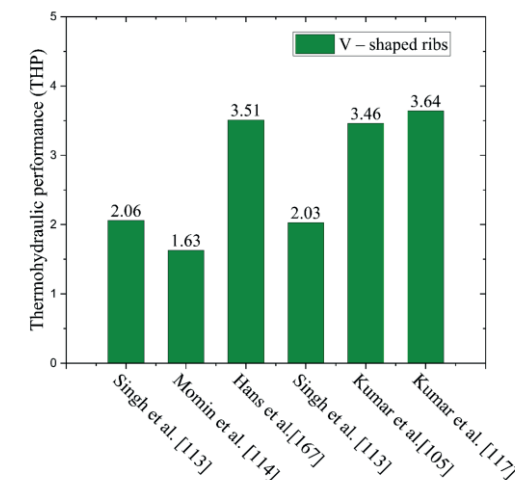


Fig. 73. Variation in THP of SAH based on various V-shaped rib roughness.

Figure 74 illustrates the THP results from various investigations on SAHs with W-shaped roughness. Thakur and Thakur [127] developed the SAH model using roughness elements in the form of W-shaped ribs with symmetrical gaps along with the staggered ribs, and found that the maximum value of THP was 3.24. The plot shows that as the Reynolds number increases, larger gaps lead to a reduction in heat transfer, possibly because more extensive gaps allow a larger part of the flow to pass without going over the ribs. Moreover, the friction factor decreases for the full arrangement values due to the weakening of the laminar sublayer. The quality of secondary stream vortices close to the shorter W-shaped rib sections of roughness design is influenced by increasing the number of gaps on every appendage of the rib geometry. As the number of gaps increases, the friction factor values decrease, which might be due to the overwhelming effect of debilitating secondary flow vortices. The Nusselt number values go on increasing due to the interaction of incoming flow from gaps to cause interruption to the flow between the two adjacent ribs. The greatest improvement in Nusselt number was

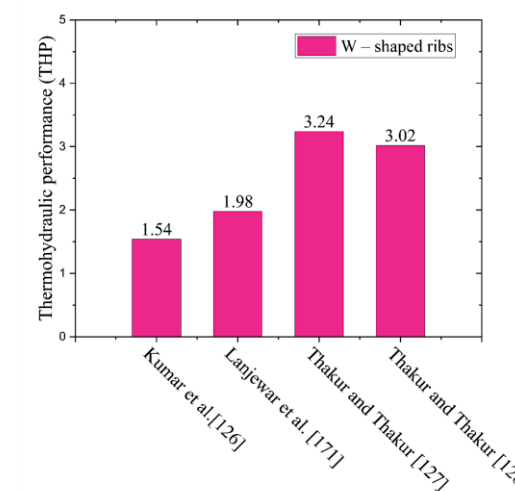


Fig. 74. Variation in THP of SAH based on various W-shaped rib roughness.

at the relative staggered pitch (p/P) equal to 0.65, which corresponds to the situation of a redevelopment of the boundary layer following the reattachment of the stream. The plot also shows a variety of friction variables with Reynolds number for relative staggered pitch. The most extreme estimate of friction was found at $p/P = 0.75$, and the least at $p/P = 0.65$. A staggered rib enables the stream to mix with the incoming flow from gaps to create more disturbances in the flow between two adjacent ribs, which leads to an increase in friction. In the case of low friction values, the cause was attributed to the flow separation and reattachment. The most extreme heat exchange credited to $w/e = 4.5$ might be the result of reattachment of the primary stream and breaking of the redeveloping boundary layer. The low value of the friction factor at $w/e = 4.5$ might be credited to the dissipation of the approaching stream from gaps, while in the case of $w/e = 6$, redirections of the primary stream happen towards the trailing edge of the multigap staggered W-shaped rib roughness.

Among all reported inclined baffles roughened SAHs, Parsa et al. [140] obtained the maximum THP factor of 3.28, as depicted in Fig. 75. They used novel staggered cuboid baffles as roughness elements for the development of the SAH setup. The cuboid baffles under the absorber plate increase the heat transfer. They also increase the frictional resistance to the flow, which increases the requirement of pumping power. The cuboid baffles are more effective at lower Reynolds numbers compared to higher Reynolds numbers due to the nature of heat transfer that decreases with an increase in Reynolds number. Various cuboid baffle arrangements were employed under that study, but the most effective were 'I', 'T' and 'L' shaped configurations, which yielded the highest THP. The rise in THP indicated higher heat transfer and nominal increment in friction factor, due to the additional secondary flow generation by baffles and thereby, better mixing and turbulence created under the absorber plate.

The maximum THP obtained amongst the SAHs modified with non-uniform ribs, as reported under this review work, was 2.66. The model, which exhibited the maximum THP, was built

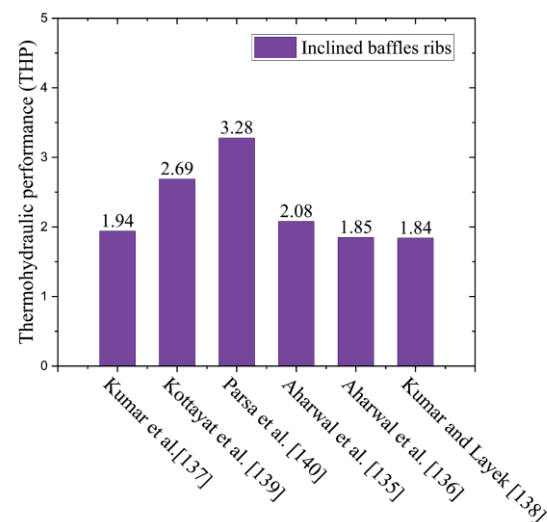


Fig. 75. Variation in THP of SAH based on various inclined baffles.

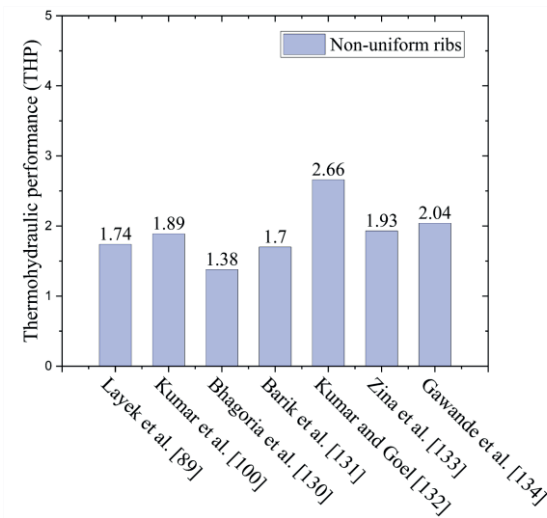


Fig. 76. Variation in THP of SAH based on various non-uniform rib roughness.

by Kumar and Goel [132], as shown in Fig. 76. The experimental setup developed by them had five types of rib roughness, such as circular, semi-circular, square, triangular and rectangular, in a triangular fluid passage as the test section. These ribs were employed to break the laminar sublayer to yield more mixing of fluid near the absorber wall, thereby advancing heat transfer. These ribs generated primary, as well as secondary flow due to their geometrical shape. The microlevel view indicated that ribs created various reattachment points between two successive ribs. The multiple reattachments of the fluid stream cause flow separation and attachment of the flow at the upstream and downstream sides. These phenomena introduce local turbulence under the absorber plate, which ensures THP as high as 2.66, as was demonstrated by the authors in this experimental study.

Within the group of other rib geometries reported here in this review work, the maximum THP was 3.66, which was obtained by Yadav and Saini [149]. They developed SAH based on impinging jet to numerically investigate heat transfer behaviour of a solar air heater. Nusselt number was increased initially with an increase in the jet height, and it started to decline beyond jet height ratio (H_j/D_h) of 0.216. It was due to crossflow of air to the upward direction through the jet height that causes back pressure with the increasing height of jet and velocity inside the jets. The maximum THP found with these ribs was because of the interference between jet was minimised with an increment in the jet height. The detailed comparison of THP values for other roughness geometries reported is presented in Fig. 77.

According to the comprehensive comparison of optimum SAH in every type of SAH considered in the present review paper (Table 9), the maximum THP was reported by Hassan et al. [123], who used dimples or protrusions as roughness elements for roughened SAH (Fig. 78). The authors introduced modified SAH using dimple ribs of multiple arc arrangements. They led to the highest heat transfer due to availability of maximum reattachments point, destruction of laminar sublayer because of the presence of arc array and two-way flow pattern, such as primary – secondary flow stream.

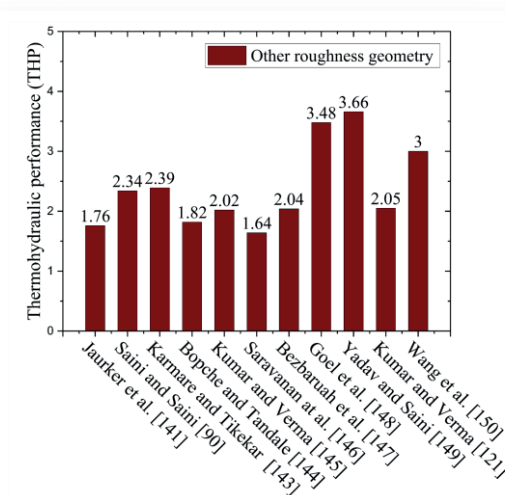


Fig. 77. Variation in THP of SAH based on various other roughness geometries.

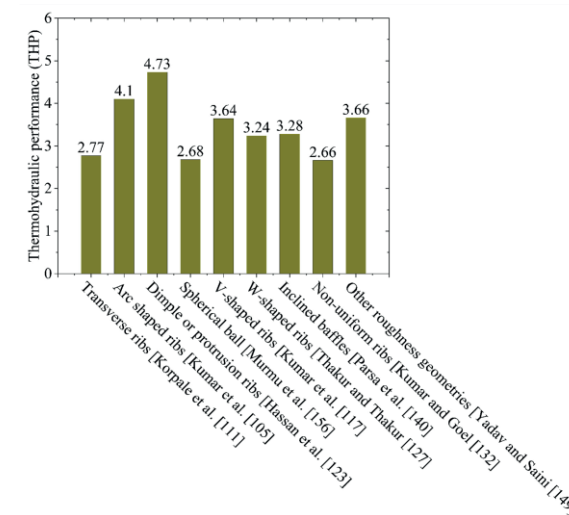


Fig. 78. Optimal THP of SAH based on various rib roughness.

12. Research gaps for future works

Based on the extensive literature review, the following observations were found for future work. These recommendations would be very beneficial for those who are working in the field of solar air heater performance improvements, especially on roughened solar air heaters:

- Very limited work involving gap and multiple gaps of roughness geometry over the absorber plate of SAH has been reported to date. There is a very significant scope to study various thermal performance enhancements in transverse wire ribs with a gap and with multiple gaps.
- Numerous studies have been conducted on plate roughness (absorber plate) to improve the thermal and thermo-hydraulic performance of solar air heaters and heat exchang-

ers. However, there is room for and a need for research on the effect of roughness geometry on flat plates with and without reflectors (on the left and right sides of the duct walls).

- There have been numerical studies conducted separately, such as CFD and ANN techniques, but no combined comparative studies containing various well-established numerical approaches or open source-based tools, such as CFD, ANN, Taguchi method, surface volume methods, or OpenFOAM, on the specific new novel roughness geometry over the flat surface of the absorber in solar air heaters have been reported yet.
- So far, only a few studies been undertaken to investigate the effect of spherical ball-based artificial roughness in SAH. Various setup configurations, such as transverse shape, W-shape, V-shape, with and without gaps, can be used using this type of artificial roughness in SAH.
- There is also the possibility of observing the effect of coating and nanocoating, since nanoparticle change the roughness over the flat plate (absorber plate). Nanocoating may be applied on many types of roughness to improve the thermal performance of SAHs.
- There have been relatively few works documented on three-sided roughness for effective SAH. There is scope and potential to expand the work done on one-side roughness (bottom plate/absorber plate) geometry to three-side roughness geometry in the SAH duct for improved thermal and thermo-hydraulic performance.
- Various studies on SAH have been conducted utilising various duct shapes, but no study has been reported on comparative studies on duct shapes of SAH for the optimum configuration to improve overall performance of SAH.
- A lot of studies have been done on the dimple shape, and dimples with gap has also been examined for effective efficiency of SAH; however, there is no report by any author on continuous dimple roughness geometry. There are other patterns that may be used, including a transverse shape, an S-shape, a C-shape, and so on.
- The angle of attack has a substantial impact on the performance evaluation of SAH. This may be adjusted by switching the absorber plate's side (upstream/downstream). Various studies have been published in this respect for the performance assessment of SAH, especially as an arc with gap, i.e. apex-down roughness geometry, but there is potential to employ apex-up with gap and multiple gaps for an optimum thermal performance of SAH.
- There have been very few works done employing a combination of different roughness geometries. Mixed roughness geometries can be applied here, such as dimple continuous and discontinuous, transverse wire ribs and arc-shaped ribs, apex up and apex down arrangement in W-shaped, C-shaped and S-shaped roughness geometries.

Table 9. Optimal thermo-hydraulic performance of solar air heaters based on different artificial roughness.

Authors	Type of artificial roughness utilised	Thermo-hydraulic performance
Hassan et al. [123]	Dimple or protrusion ribs	4.73
Kumar et al. [105]	Discrete multi arc-shaped ribs	4.10
Kumar et al. [112]	Discrete multiple arc-shaped ribs	3.85
Yadav and Saini [149]	Impinging jet	3.66
Kumar et al. [161]	Innovatively arranged arcs (combination of alternatively arranged forward and backward facing arced-ribs)	3.61
Goel et al. [148]	Hemispherical cavities	3.48
Parsa et al. [140]	Staggered cuboid baffles	3.28
Korpale et al. [111]	Rectangular section ribs	2.77
Kottayat et al. [139]	Discrete multiple inclined baffle	2.69
Kumar and Goel [132]	Semi-circular, circular, triangular, square, rectangular ribs	2.66
Karmare and Tikekar [143]	Metal grit ribs	2.39
Saini and Saini [90]	Expanded metal mesh	2.34
Srivastava et al. [174]	V-shaped ribs with gap	2.21
Ghritlahre [107]	Arc-shaped wire ribs with apex upstream	2.09
Sharma et al. [94]	Transverse, inclined transverse, dimple, inclined dimple, arc and sine wave ribs	2.05
Kumar and Verma [145]	Sinusoidal rib	2.05
Gawande et al. [134]	20° chamfered rib	2.05
Bezbaruah et al. [147]	Staggered conical vortex generators	2.04
Gawande et al. [134]	Right-angle triangular rib	2.04
Kumar and Verma [145]	Sinusoidal protrusion rib	2.02
Bhuvad, Azad and Lanjewar [110]	Apex-up discrete arc ribs	2.01
Lanjewar et al. [171]	W-shaped ribs of different orientations	1.98
Kumar et al. [137]	Angled perforated baffles	1.94
Zina et al. [133]	Triangular ribs with semi-circular nooks	1.93
Gawande et al. [134]	Reverse I-shaped rib	1.90
Kumar et al. [153]	Rectangular rib roughened	1.89
Mahanand and Senapati [95]	Transverse inverted-t shaped rib	1.86
Gawande et al. [96]	Combined square and equilateral triangular ribs as	1.86
Bopche and Tandale [144]	Inverted u shape rib	1.82
Prasad and Saini [86]	Transverse wire	1.78
Jaurker et al. [141]	Rib groove	1.76
Barik et al. [131]	T-rib, left arm inclined T-rib, right arm inclined T-rib, Y-rib, concave and convex T-ribs	1.70
Kumar et al. [152]	Semicircular rib	1.70
Saravanan at al. [146]	Staggered C-shape finned absorber plate	1.64
Ghritlahre [107]	Apex downstream	1.59
Srivastava et al. [108]	Arc-shaped ribs with gap	1.50
Singh et al. [93]	Uniform/non-uniform square-wave profile transverse roughness rib	1.49
Kumar et al. [99]	Square rib	1.44
Bhagoria et al. [130]	Transverse wedge-shaped ribs	1.38

13. Conclusions

This paper aims to analyse the diverse experimental studies conducted on ducts and solar air heaters incorporating artificial roughness rib elements, focusing on their impact on heat transfer and friction factor characteristics. Various elements, materials, and geometric parameters have been employed to assess the thermo-hydraulic performance of the ducts and solar air heaters. Drawing from a thorough literature review, the following conclusions can be derived:

- 1) An evident enhancement in heat transfer is observed in solar air heaters when the flow is subjected to surface roughening, leading to increased friction. Nevertheless, it is important to note that various researchers have reported varying increments in heat transfer and friction factor values for each specific rib geometry employed.
- 2) A comparison of the investigations conducted by different researchers, in terms of the Nusselt number and friction factor increments compared to a smooth duct, showed that the highest values, of 7.61 for the Nusselt number and 6.48 for the friction factor, were obtained utilising artificial roughness elements in the form of discrete multi-arc ribs. It also indicated that an increase in heat transfer is accompanied by a simultaneous increase in the friction factor.
- 3) Among the various rib geometries studied in experiments thus far, it has been observed that protrusions or dimple-shaped elements tend to exhibit the lowest friction factor values compared to other rib elements. Simultaneously, these protrusions or dimple-shaped elements also contribute to an increase in the Nusselt number at the optimum parameter settings. Therefore, in terms of achieving a balance between heat transfer enhancement and minimising friction, protrusions or dimple-shaped rib elements appear to be more favourable compared to other alternatives.
- 4) The experiments conducted on solar air heaters with various rib geometries and the set of corresponding parameter values identified the specific parameter values for each rib element that yield the best overall performance.
- 5) The developed correlations for the Nusselt number and friction factor based on the range of parameters considered in the experiments provide valuable insights for designers and researchers. These correlations, along with the heat transfer values, can assist beginners in this field by providing an understanding of the different geometries used and the impact of each parameter on heat transfer and flow characteristics. This knowledge enables designers and investigators to work towards identifying the optimum values for enhanced performance in their respective applications.
- 6) Dimple protrusion ribs have proven to be the most suitable rib elements in terms of thermo-hydraulic performance parameters. Extensive research and experimentation have demonstrated their effectiveness in enhancing heat transfer, while simultaneously minimising friction. The unique characteristics of dimple protrusion ribs, such

as their roughness profile and surface texture, contribute to improved heat transfer coefficients and efficient fluid flow. Thus, in terms of overall thermo-hydraulic performance, dimple protrusion ribs have emerged as a favourable choice among rib elements.

- 7) In the study conducted, it has been observed that the friction factor decreases as the Reynolds number increases for all roughness geometries considered. Among the different roughness geometries, the one of four ribbed walls resulted in the highest frictional losses. This can be attributed to their large frontal area, which contributes to increased form drag. On the other hand, circular protrusions arranged in an arc fashion exhibit the lowest values of the friction factor. This may be due to the discrete arrangement of the circular protrusions, which results in relatively lower form drag, as well as skin drag. In fact, the skin drag for circular protrusions can be even lower than that of rib roughness, leading to their favourable performance in terms of frictional losses.

Acknowledgements

The authors would like to thank Chhattisgarh Swami Vivekanand Technical University, Bhilai, Chhattisgarh, 491107, India, and Chhattisgarh Institute of Technology (CGIT) Jagdalpur, Bastar, Chhattisgarh, India, as well as the University Teaching Department of Engineering for their continuous support and encouragement in preparing this research article.

References

[1] Kumar, A., Saini, R.P., & Saini, J S. (2014). A review of thermo-hydraulic performance of artificially roughened solar air heaters. *Renewable and Sustainable Energy Reviews*, 37, 100–122. doi: 10.1016/j.rser.2014.04.063

[2] Prakash, C., & Saini, R.P. (2019). Use of artificial roughness for performance enhancement of solar air heaters—a review. *International Journal of Green Energy*, 16(7), 549–569. doi: 10.1080/15435075.2019.1598418

[3] Mohammadi, K., & Sabzpooshani, M. (2013). Comprehensive performance evaluation and parametric studies of single pass solar air heater with fins and baffles attached over the absorber plate. *Energy*, 57, 741–750. doi: 10.1016/j.energy.2013.05.016

[4] Chamoli, S., Chauhan, R., Thakur, N.S., & Saini, J.S. (2012). A review of the performance of double pass solar air heater. *Renewable and Sustainable Energy Reviews*, 16(1), 481–492. doi: 10.1016/j.rser.2011.08.012

[5] Khanlari, A., Sözen, A., Afshari, F., Tuncer, A.D., Ağbulut, Ü., & Aytaç Yılmaz, Z. (2022). Numerical and experimental analysis of parallel-pass forced convection solar air heating wall with different plenum and absorber configurations. *International Journal of Numerical Methods for Heat & Fluid Flow*, 32(3), 978–1001. doi: 10.1108/HFF-03-2021-0160

[6] Sahu, M.K., Sharma, M., Matheswaran, M.M., & Maitra, K. (2019). On the Use of Longitudinal Fins to Enhance the Performance in Rectangular Duct of Solar Air Heaters – A Review. *Journal of Solar Energy Engineering*, 141(3), 030801. doi: 10.1115/1.4042827

[7] Ghritlahre, H.K. (2019). Performance evaluation of solar air heating systems using artificial neural network. PhD thesis, NIT Jamshedpur.

- [8] Ghritlahre, H.K., & Prasad, R.K. (2018). Prediction of heat transfer of two different types of roughened solar air heater using Artificial Neural Network technique. *Thermal Science and Engineering Progress*, 8, 145–153. doi: 10.1016/j.tsep.2018.08.014
- [9] Kumar, V. (2019). Nusselt number and friction factor correlations of three sides concave dimple roughened solar air heater. *Renewable Energy*, 135, 355–377. doi: 10.1016/j.renene.2018.12.002
- [10] Kumar, R., Nadda, R., Rana, A., Chauhan, R., & Chandel, S.S. (2022). Impact of artificial roughness variation on heat transfer and friction characteristics of solar air heating system. *Alexandria Engineering Journal*, 61(1), 481–491. doi: 10.1016/j.aej.2021.06.031
- [11] Kumar, R., Nadda, R., Rana, A., Chauhan, R., & Chandel, S.S. (2020). Performance investigation of a solar thermal collector provided with air jets impingement on multi V-shaped protrusion ribs absorber plate. *Heat and Mass Transfer*, 56(3), 913–930. doi: 10.1007/s00231-019-02755-2
- [12] Kumar, K., Prajapati, D.R., & Samir, S. (2017). Heat transfer and friction factor correlations development for solar air heater duct artificially roughened with ‘S’ shape ribs. *Experimental Thermal and Fluid Science*, 82, 249–261. doi: 10.1016/j.expthermflusci.2016.11.012
- [13] Abdulmejeed, A.E.A., Tuncer, A.D., Khanlari, A., & Gungor, A. (2024). Investigation of combined parallel and triple-pass v-corrugated solar air heater: A numerical and experimental study. *Process Safety and Environmental Protection*, 185, 1385–1398. doi: 10.1016/j.psep.2024.03.107
- [14] Khanlari, A., Tuncer, A.D., Afshari, F., & Sözen, A. (2023). Utilization of recyclable aluminum cans as fins in a vertical solar air heating system: An experimental and numerical study. *Journal of Building Engineering*, 63, 105446. doi: 10.1016/j.job.2022.105446
- [15] Khanlari, A., Aytaç, İ., Tuncer, A.D., Variyenli, H.İ., & Şahin, H.N. (2024). Improving the performance of a PCM integrated solar air collector by adding porous fins over the bottom side of the absorber: A transient CFD study. *Journal of Energy Storage*, 90, 111847. doi: 10.1016/j.est.2024.111847
- [16] Sharma, S.P., Saini, J.S., & Varma, H.K. (1991). Thermal performance of packed-bed solar air heaters. *Solar Energy*, 47(2), 59–67. doi: 10.1016/0038-092X(91)90035-U
- [17] Ghritlahre, H.K., & Prasad, R.K. (2018). Investigation of thermal performance of unidirectional flow porous bed solar air heater using MLP, GRNN, and RBF models of ANN technique. *Thermal Science and Engineering Progress*, 6, 226–235. doi: 10.1016/j.tsep.2018.04.006
- [18] Varshney, L., & Saini, J.S. (1998). Heat transfer and friction factor correlations for rectangular solar air heater duct packed with wire mesh screen matrices. *Solar Energy*, 62(4), 255–262. doi: 10.1016/S0038-092X(98)00017-6
- [19] Choudhury, C., & Garg, H.P. (1991). Design analysis of corrugated and flat plate solar air heaters. *Renewable Energy*, 1(5-6), 595–607. doi: 10.1016/0960-1481(91)90003-8
- [20] Priyam, A., & Chand, P. (2016). Thermal and thermohydraulic performance of wavy finned absorber solar air heater. *Solar Energy*, 130, 250–259. doi: 10.1016/j.solener.2016.02.030
- [21] Afshari, F., Sözen, A., Khanlari, A., Tuncer, A.D., & Şirin, C. (2020). Effect of turbulator modifications on the thermal performance of cost-effective alternative solar air heater. *Renewable Energy*, 158, 297–310. doi: 10.1016/j.renene.2020.05.148
- [22] Khanlari, A., Sözen, A., Tuncer, A.D., Afshari, F., Gürbüz, E.Y., & Bilge, Y. C. (2021). Numerical and experimental analysis of longitudinal tubular solar air heaters made from plastic and metal waste materials. *Heat Transfer Research*, 52(10), 19–45. doi: 10.1615/HeatTransRes.2021038204
- [23] Varun, Saini, R.P., & Singal, S.K. (2007). A review on roughness geometry used in solar air heaters. *Solar Energy*, 81(11), 1340–1350. doi: 10.1016/j.solener.2007.01.017
- [24] Gupta, M.K., & Kaushik, S.C. (2009). Performance evaluation of solar air heater for various artificial roughness geometries based on energy, effective and exergy efficiencies. *Renewable Energy*, 34(3), 465–476. doi: 10.1016/j.renene.2008.06.001
- [25] Hans, V.S., Saini, R.P., & Saini, J.S. (2009). Performance of artificially roughened solar air heaters—A review. *Renewable and Sustainable Energy Reviews*, 13(8), 1854–1869. doi: 10.1016/j.rser.2009.01.030
- [26] Chamoli, S., Thakur, N.S., & Saini, J.S. (2012). A review of turbulence promoters used in solar thermal systems. *Renewable and Sustainable Energy Reviews*, 16(5), 3154–3175. doi: 10.1016/j.rser.2012.01.021
- [27] El-Sebaei, A.A., & Shalaby, S.M. (2012). Solar drying of agricultural products: A review. *Renewable and Sustainable Energy Reviews*, 16(1), 37–43. doi: 10.1016/j.rser.2011.07.134
- [28] Yadav, A.S., & Bhagoria, J.L. (2013). Heat transfer and fluid flow analysis of solar air heater: A review of CFD approach. *Renewable and Sustainable Energy Reviews*, 23, 60–79. doi: 10.1016/j.rser.2013.02.035
- [29] Yadav, A.S., & Thapak, M.K. (2014). Artificially roughened solar air heater: Experimental investigations. *Renewable and Sustainable Energy Reviews*, 36, 370–411. doi: 10.1016/j.rser.2014.04.077
- [30] Gawande, V.B., Dhoble, A.S., & Zodpe, D.B. (2014). Effect of roughness geometries on heat transfer enhancement in solar thermal systems – A review. *Renewable and Sustainable Energy Reviews*, 32, 347–378. doi: 10.1016/j.rser.2014.01.024
- [31] Alam, T., Saini, R.P., & Saini, J.S. (2014). Heat and flow characteristics of air heater ducts provided with turbulators – A review. *Renewable and Sustainable Energy Reviews*, 31, 289–304. doi: 10.1016/j.rser.2013.11.050
- [32] Alam, T., Saini, R.P., & Saini, J.S. (2014). Use of turbulators for heat transfer augmentation in an air duct – A review. *Renewable Energy*, 62, 689–715. doi: 10.1016/j.renene.2013.08.024
- [33] Mund, C., Rathore, S.K., & Sahoo, R.K. (2021). A review of solar air collectors about various modifications for performance enhancement. *Solar Energy*, 228, 140–167. doi: 10.1016/j.solener.2021.08.040
- [34] Patil, A.K. (2015). Heat transfer mechanism and energy efficiency of artificially roughened solar air heaters – A review. *Renewable and Sustainable Energy Reviews*, 42, 681–689. doi: 10.1016/j.rser.2014.10.063
- [35] Sharma, S.K., & Kalamkar, V.R. (2015). Thermo-hydraulic performance analysis of solar air heaters having artificial roughness—A review. *Renewable and Sustainable Energy Reviews*, 41, 413–435. doi: 10.1016/j.rser.2014.08.051
- [36] Kumar, A., & Kim, M.H. (2015). Convective heat transfer enhancement in solar air channels. *Applied Thermal Engineering*, 89, 239–261. doi: 10.1016/j.applthermaleng.2015.06.015
- [37] Lanjewar, A.M., Bhagoria, J.L., & Agrawal, M.K. (2015). Review of development of artificial roughness in solar air heater and performance evaluation of different orientations for double arc rib roughness. *Renewable and Sustainable Energy Reviews*, 43, 1214–1223. doi: 10.1016/j.rser.2014.11.081
- [38] Sheikholslami, M., Gorji-Bandpy, M., & Ganji, D.D. (2015). Review of heat transfer enhancement methods: Focus on passive methods using swirl flow devices. *Renewable and Sustainable Energy Reviews*, 49, 444–469. doi: 10.1016/j.rser.2015.04.113
- [39] Singh, S., Chander, S., & Saini, J.S. (2015). Thermo-hydraulic performance due to relative roughness pitch in V-down rib with gap in solar air heater duct – Comparison with similar rib roughness geometries. *Renewable and Sustainable Energy Reviews*, 43, 1159–1166. doi: 10.1016/j.rser.2014.11.087
- [40] Saxena, A., Varun, & El-Sebaei, A.A. (2015). A thermodynamic review of solar air heaters. *Renewable and Sustainable Energy Reviews*, 43, 863–890. doi: 10.1016/j.rser.2014.11.059
- [41] Kumar, R., Varun, & Kumar, A. (2016). Thermal and fluid dynamic characteristics of flow through triangular cross-sectional duct: A review. *Renewable and Sustainable Energy Reviews*, 61, 123–140. doi: 10.1016/j.rser.2016.03.011
- [42] Aravindh, M.A., & Sreekumar, A. (2016). Efficiency enhancement in solar air heaters by modification of absorber plate—a review. *International Journal of Green Energy*, 13(12), 1209–1223. doi: 10.1080/15435075.2016.1183207
- [43] Sharma, S.K., & Kalamkar, V.R. (2016). Computational Fluid Dynamics approach in thermo-hydraulic analysis of flow in ducts with rib roughened walls – A review. *Renewable and Sustainable Energy Reviews*, 55, 756–788. doi: 10.1016/j.rser.2015.10.160
- [44] Kumar, A., & Kim, M.H. (2016). Thermohydraulic performance of rectangular ducts with different multiple V-rib roughness shapes: A comprehensive review and comparative study. *Renewable and Sustainable Energy Reviews*, 54, 635–652. doi: 10.1016/j.rser.2015.10.030
- [45] Ravi, R.K., & Saini, R.P. (2016). A review on different techniques used for performance enhancement of double pass solar air heaters. *Renewable and Sustainable Energy Reviews*, 56, 941–952. doi: 10.1016/j.rser.2015.12.004
- [46] Gawande, V.B., Dhoble, A.S., Zodpe, D.B., & Chamoli, S. (2016). A review of CFD methodology used in literature for predicting thermo-hydraulic performance of a roughened solar air heater. *Renewable and Sustainable Energy Reviews*, 54, 550–605. doi: 10.1016/j.rser.2015.10.025
- [47] Yadav, A.S., & Thapak, M.K. (2016). Artificially roughened solar air heater: A comparative study. *International Journal of Green Energy*, 13(2), 143–172. doi: 10.1080/15435075.2014.917419
- [48] Kabeel, A.E., Hamed, M.H., Omara, Z.M., & Kandael, A.W. (2017). Solar air heaters: Design configurations, improvement methods and applications – A detailed review. *Renewable and Sustainable Energy Reviews*, 70, 1189–1206. doi: 10.1016/j.rser.2016.12.021
- [49] Alam, T., & Kim, M.H. (2017). A critical review on artificial roughness provided in rectangular solar air heater duct. *Renewable and Sustainable Energy Reviews*, 69, 387–400. doi: 10.1016/j.rser.2016.11.192
- [50] Singh, I., & Singh, S. (2018). A review of artificial roughness geometries employed in solar air heaters. *Renewable and Sustainable Energy Reviews*, 92, 405–425. doi: 10.1016/j.rser.2018.04.108
- [51] Singh Bisht, V., Patil, A.K., & Gupta, A. (2018). Review and performance evaluation of roughened solar air heaters. *Renewable and Sustainable Energy Reviews*, 81, 954–977. doi: 10.1016/j.rser.2017.08.036
- [52] Kumar, R., Kumar, A., Chauhan, R., & Maithani, R. (2018). Comparative study of effect of various blockage arrangements on thermal hydraulic performance in a roughened air passage. *Renewable and Sustainable Energy Reviews*, 81, 447–463. doi: 10.1016/j.rser.2017.08.023
- [53] Chauhan, R., Singh, T., Thakur, N.S., Kumar, N., Kumar, R., & Kumar, A. (2018). Heat transfer augmentation in solar thermal collectors using impinging air jets: A comprehensive review. *Renewable and Sustainable Energy Reviews*, 82, 3179–3190. doi: 10.1016/j.rser.2017.10.025
- [54] Elsheikh, A.H., Sharshir, S.W., Abd Elaziz, M., Kabeel, A.E., Guilan, W., & Haiou, Z. (2019). Modeling of solar energy systems using artificial neural network: A comprehensive review. *Solar Energy*, 180, 622–639. doi: 10.1016/j.solener.2019.01.037
- [55] Hu, J., & Zhang, G. (2019). Performance improvement of solar air collector based on airflow reorganization: A review. *Applied Thermal Engineering*, 155, 592–611. doi: 10.1016/j.applthermaleng.2019.04.021
- [56] Chhapparwal, G.K., Srivastava, A., & Dayal, R. (2019). Artificial repeated-rib roughness in a solar air heater – A review. *Solar Energy*, 193, 329–359. doi: 10.1016/j.solener.2019.10.011
- [57] Ghritlahre, H.K., & Sahu, P.K. (2020). A comprehensive review on energy and exergy analysis of solar air heaters. *Archives of Thermodynamics*, 41(3), 183–222. doi: 10.24425/ather.2020.134577
- [58] Vengadesan, E., & Senthil, R. (2020). A review on recent developments in thermal performance enhancement methods of flat plate solar air collector. *Renewable and Sustainable Energy Reviews*, 134, 110315. doi: 10.1016/j.rser.2020.110315
- [59] Yadav, K.D., & Prasad, R.K. (2020). Performance analysis of parallel flow flat plate solar air heater having arc shaped wire roughened absorber plate. *Renewable Energy Focus*, 32, 23–44. doi: 10.1016/j.ref.2019.10.002
- [60] Arunkumar, H.S., Karanth, K.V., & Kumar, S. (2020). Review on the design modifications of a solar air heater for improvement in the thermal performance. *Sustainable Energy Technologies and Assessments*, 39, 100685. doi: 10.1016/j.seta.2020.100685
- [61] Nidhul, K., Yadav, A.K., Anish, S., & Kumar, S. (2021). Critical review of ribbed solar air heater and performance evaluation of various V-rib configuration. *Renewable and Sustainable Energy Reviews*, 142, 110871. doi: 10.1016/j.rser.2021.110871
- [62] Chamarthi, S., & Singh, S. (2021). A comprehensive review of experimental investigation procedures and thermal performance enhancement techniques of solar air heaters. *International Journal of Energy Research*, 45(4), 5098–5164. doi: 10.1002/er.6255
- [63] Ghritlahre, H.K., Chandrakar, P., & Ahmad, A. (2021). A Comprehensive Review on Performance Prediction of Solar Air Heaters Using Artificial Neural Network. *Annals of Data Science*, 8(3), 547–570. doi: 10.1007/s40745-019-00236-1
- [64] Ghritlahre, H.K., Verma, M., Parihar, J.S., Mondloe, D.S., & Agrawal, S. (2022). A detailed review of various types of solar air heaters performance. *Solar Energy*, 237, 173–195. doi: 10.1016/j.solener.2022.03.042
- [65] Rice, D. (1867). *Quoted in Solar Energy Research by Daniels and Duffie*. US Patent No. 68 459.
- [66] Morse, E.S. (1881). *Warming and ventilating apartments by Sun's rays*. US Patent No. 246 626.
- [67] Cabot, G.L. (1938). *Solar 1: The MIT Solar House*.
- [68] Telkes, M., & Raymond, E. (1949). Storing solar heat in chemicals. *Heating and Ventilating*, 46, 79–86.
- [69] Bliss Jr, R.W. (1955). Design and performance of nation's only fully solar-heated house. *Air Conditioning, Heating, Ventilating*, 52(10), 68–73.
- [70] Löf, G.O.G. (1954). *Solar heating apparatus and method*. US Patent No. 2 680 565.
- [71] Doherty, P. (2007). *Solar air heater*. US Patent No. US2007/0199562A1.
- [72] Vachon, C. (2010). *Perforated transparent glazing for heat recovery and solar air heating*. US Patent No. US2010/0000520A1.
- [73] MacKay, K.H. (2011). *Solar air heating device*. US Patent No. US2011/0180058A1.
- [74] Ryan, T.D. (2011). *Curved transpired solar air heater and conduit*. US Patent No. US2011/0120449A1.

- [75] Wu, H. (2013). *Solar energy air heater*. Chinese Patent No. CN202973585U.
- [76] Goldstein, L., & Sparrow, E.M. (1976). Experiments on the Transfer Characteristics of a Corrugated Fin and Tube Heat Exchanger Configuration. *Journal of Heat Transfer*, 98(1), 26–34. doi: 10.1115/1.3450464
- [77] Okamoto, S., Seo, S., Nakaso, K., & Kawai, I. (1993). Turbulent shear flow and heat transfer over the repeated two-dimensional square ribs on ground plane. *ASME Journal of Fluids Engineering*, 115(4), 631–637. doi: 10.1115/1.2910191
- [78] Nikuradse, J. (1950). Laws of flow in rough pipes. *VDI-Forschungsheft*, 361.
- [79] Nunner, W. (1958). *Heat transfer and pressure drop in rough tubes*. Atomic Energy Research Establishment, AERE Lib/Trans 786.
- [80] Dipprey, D.F., & Sabersky, R.H. (1963). Heat and momentum transfer in smooth and rough tubes at various Prandtl numbers. *International Journal of Heat and Mass Transfer*, 6(5), 329–353. doi: 10.1016/0017-9310(63)90120-2
- [81] Lewis, M.J. (1975). Optimising the thermohydraulic performance of rough surfaces. *International Journal of Heat and Mass Transfer*, 18(11), 1243–1248. doi: 10.1016/0017-9310(75)90232-X
- [82] Han, J.C. (1984). Heat Transfer and Friction in Channels With Two Opposite Rib-Roughened Walls. *Journal of Heat Transfer*, 106(4), 774–781. doi: 10.1115/1.3246751
- [83] Prasad, B.N., & Saini, J.S. (1991). Optimal thermohydraulic performance of artificially roughened solar air heaters. *Solar Energy*, 47(2), 91–96. doi: 10.1016/0038-092X(91)90039-Y
- [84] Prasad, B.N., & Saini, J.S. (1988). Effect of artificial roughness on heat transfer and friction factor in a solar air heater. *Solar Energy*, 41(6), 555–560. doi: 10.1016/0038-092X(88)90058-8
- [85] Sethi, M., Varun, & Thakur, N.S. (2012). Correlations for solar air heater duct with dimpled shape roughness elements on absorber plate. *Solar Energy*, 86(9), 2852–2861. doi: 10.1016/j.solener.2012.06.024
- [86] Prasad, B.N., & Saini, J.S. (1990). Enhancement of solar air heater performance using artificial roughness. In *Proceedings of the 1st World Renewable Energy Congress* (pp. 1067–1071), 23–28 September, Reading, UK.
- [87] Gupta, D. (1993). *Investigations on fluid flow and heat transfer in solar air heaters with roughened absorbers*. PhD thesis, University of Roorkee, India.
- [88] Karwa, R. (2003). Experimental studies of augmented heat transfer and friction in asymmetrically heated rectangular ducts with ribs on the heated wall in transverse, inclined, v-continuous and v-discrete pattern. *International Communications in Heat and Mass Transfer*, 30(2), 241–250. doi: 10.1016/S0735-1933(03)00035-6
- [89] Layek, A., Saini, J.S., & Solanki, S.C. (2009). Effect of chamfering on heat transfer and friction characteristics of solar air heater having absorber plate roughened with compound turbulators. *Renewable Energy*, 34(5), 1292–1298. doi: 10.1016/j.renene.2008.09.016
- [90] Saini, R.P., & Saini, J.S. (1997). Heat transfer and friction factor correlations for artificially roughened ducts with expanded metal mesh as roughness element. *International Journal of Heat and Mass Transfer*, 40(4), 973–986. doi: 10.1016/0017-9310(96)00019-1
- [91] Verma, S., & Prasad, B. (2000). Investigation for the optimal thermohydraulic performance of artificially roughened solar air heaters. *Renewable Energy*, 20(1), 19–36. doi: 10.1016/S0960-1481(99)00081-6
- [92] Singh, I., Vardhan, S., Singh, S., & Singh, A. (2019). Experimental and CFD analysis of solar air heater duct roughened with multiple broken transverse ribs: A comparative study. *Solar Energy*, 188, 519–532. doi: 10.1016/j.solener.2019.06.022
- [93] Singh, I., Singh, S., & Vardhan, S. (2021). Heat transfer and fluid flow characteristics of solar air heater duct with non-uniform ribs. *Journal of Mechanical Science and Technology*, 35(1), 343–350. doi: 10.1007/s12206-020-1234-9
- [94] Sharma, S., Das, R.K., & Kulkarni, K. (2021). Computational and experimental assessment of solar air heater roughened with six different baffles. *Case Studies in Thermal Engineering*, 27, 101350. doi: 10.1016/j.csite.2021.101350
- [95] Mahanand, Y., & Senapati, J.R. (2020). Thermal enhancement study of a transverse inverted-T shaped ribbed solar air heater. *International Communications in Heat and Mass Transfer*, 119, 104922. doi: 10.1016/j.icheatmasstransfer.2020.104922
- [96] Gawande, V.B., Dhoble, A.S., Zodpe, D.B., & Fale, S.G. (2020). Thermal performance evaluation of solar air heater using combined square and equilateral triangular rib roughness. *Australian Journal of Mechanical Engineering*, 18(2), 234–244. doi: 10.1080/14484846.2018.1519986
- [97] Kumar, A., Akshayveer, Singh, A.P., & Singh, O.P. (2020). Efficient designs of double-pass curved solar air heaters. *Renewable Energy*, 160, 1105–1118. doi: 10.1016/j.renene.2020.06.115
- [98] Kumar, A., & Layek, A. (2019). Nusselt number and fluid flow analysis of solar air heater having transverse circular rib roughness on absorber plate using LCT and computational technique. *Thermal Science and Engineering Progress*, 14, 100398. doi: 10.1016/j.tsep.2019.100398
- [99] Kumar, R., Kumar, A., & Goel, V. (2019). Performance improvement and development of correlation for friction factor and heat transfer using computational fluid dynamics for ribbed triangular duct solar air heater. *Renewable Energy*, 131, 788–799. doi: 10.1016/j.renene.2018.07.078
- [100] Kumar, R., Goel, V., & Kumar, A. (2018). Investigation of heat transfer augmentation and friction factor in triangular duct solar air heater due to forward facing chamfered rectangular ribs: A CFD based analysis. *Renewable Energy*, 115, 824–835. doi: 10.1016/j.renene.2017.09.010
- [101] Kumar, R., Varun, & Kumar, A. (2017). Experimental and Computational Fluid Dynamics Study on Fluid Flow and Heat Transfer in Triangular Passage Solar Air Heater of Different Configurations. *Journal of Solar Energy Engineering*, 139(4), 041003. doi: 10.1115/1.4036775
- [102] Saini, S.K., & Saini, R.P. (2008). Development of correlations for Nusselt number and friction factor for solar air heater with roughened duct having arc-shaped wire as artificial roughness. *Solar Energy*, 82(12), 1118–1130. doi: 10.1016/j.solener.2008.05.010
- [103] Singh, A.P., Varun, & Siddhartha. (2014). Effect of artificial roughness on heat transfer and friction characteristics having multiple arc shaped roughness element on the absorber plate. *Solar Energy*, 105, 479–493. doi: 10.1016/j.solener.2014.04.007
- [104] Sahu, M.K., & Prasad, R.K. (2016). Exergy based performance evaluation of solar air heater with arc-shaped wire roughened absorber plate. *Renewable Energy*, 96, 233–243. doi: 10.1016/j.renene.2016.04.083
- [105] Kumar, R., Kumar, S., Nadda, R., Kumar, K., & Goel, V. (2022). Thermo-hydraulic efficiency and correlation development of an indoor designed jet impingement solar thermal collector roughened with discrete multi-arc ribs. *Renewable Energy*, 189, 1259–1277. doi: 10.1016/j.renene.2022.03.037
- [106] Yadav, S., Kaushal, M., Varun, & Siddhartha. (2013). Nusselt number and friction factor correlations for solar air heater duct having protrusions as roughness elements on absorber plate. *Experimental Thermal and Fluid Science*, 44, 34–41. doi: 10.1016/j.expthermflusci.2012.05.011
- [107] Ghritlahre, H.K. (2021). Heat transfer and friction factor characteristic investigation of roughened solar air heater using arc-shaped wire rib roughness. *International Journal of Ambient Energy*, 43(1), 1866–1877. doi: 10.1080/01430750.2021.1934115
- [108] Srivastava, A., Chhapparwal, G.K., & Sharma, R.K. (2020). Numerical and experimental investigation of different rib roughness in a solar air heater. *Thermal Science and Engineering Progress*, 19, 100576. doi: 10.1016/j.tsep.2020.100576
- [109] Gill, R.S., Hans, V.S., Saini, J.S., & Singh, S. (2017). Investigation on performance enhancement due to staggered piece in a broken arc rib roughened solar air heater duct. *Renewable Energy*, 104, 148–162. doi: 10.1016/j.renene.2016.12.002
- [110] Bhuvad, S.S., Azad, R., & Lanjewar, A. (2022). Thermal performance analysis of apex-up discrete arc ribs solar air heater-an experimental study. *Renewable Energy*, 185, 403–415. doi: 10.1016/j.renene.2021.12.037
- [111] Korpale, V.S., Deshmukh, S.P., Mathpati, C.S., & Dalvi, V.H. (2020). Numerical simulations and optimization of solar air heaters. *Applied Thermal Engineering*, 180, 115744. doi: 10.1016/j.applthermaleng.2020.115744
- [112] Kumar, R., Goel, V., Singh, P., Saxena, A., Kashyap, A.S., & Rai, A. (2019). Performance evaluation and optimization of solar assisted air heater with discrete multiple arc shaped ribs. *Journal of Energy Storage*, 26, 100978. doi: 10.1016/j.est.2019.100978
- [113] Singh, S., Chander, S., & Saini, J.S. (2011). Heat transfer and friction factor of discrete V-down rib roughened solar air heater ducts. *Journal of Renewable and Sustainable Energy*, 3(1), 013108. doi: 10.1063/1.3558865
- [114] Ebrahim Momin, A.-M., Saini, J.S., & Solanki, S.C. (2002). Heat transfer and friction in solar air heater duct with V-shaped rib roughness on absorber plate. *International Journal of Heat and Mass Transfer*, 45(16), 3383–3396. doi: 10.1016/S0017-9310(02)00046-7
- [115] Deo, N.S., Chander, S., & Saini, J.S. (2016). Performance analysis of solar air heater duct roughened with multigap V-down ribs combined with staggered ribs. *Renewable Energy*, 91, 484–500. doi: 10.1016/j.renene.2016.01.067
- [116] Singh, S., Chander, S., & Saini, J.S. (2011). Heat transfer and friction factor correlations of solar air heater ducts artificially roughened with discrete V-down ribs. *Energy*, 36(8), 5053–5064. doi: 10.1016/j.energy.2011.05.052
- [117] Kumar, A., Saini, R.P., & Saini, J.S. (2012). Experimental investigation on heat transfer and fluid flow characteristics of air flow in a rectangular duct with Multi v-shaped rib with gap roughness on the heated plate. *Solar Energy*, 86(6), 1733–1749. doi: 10.1016/j.solener.2012.03.014
- [118] Patel, S.S., & Lanjewar, A. (2021). Heat transfer enhancement using additional gap in symmetrical element of V-geometry roughened solar air heater. *Journal of Energy Storage*, 38, 102545. doi: 10.1016/j.est.2021.102545
- [119] Kumar, V., & Prasad, L. (2019). Thermal performance investigation of three sides concave dimple roughened solar air heaters. *Solar Energy*, 188, 361–379. doi: 10.1016/j.solener.2019.06.008
- [120] Saini, R.P., & Verma, J. (2008). Heat transfer and friction factor correlations for a duct having dimple-shape artificial roughness for solar air heaters. *Energy*, 33(8), 1277–1287. doi: 10.1016/j.energy.2008.02.017
- [121] Kumar, S., & Verma, S.K. (2020). Three-dimensional simulation and experimental validation of solar air heater having sinusoidal rib. *Energy Sources, Part A: Recovery, Utilization, and Environmental Effects*, 43(24), 3170–3188. doi: 10.1080/15567036.2020.1818007
- [122] Bhushan, B., & Singh, R. (2011). Nusselt number and friction factor correlations for solar air heater duct having artificially roughened absorber plate. *Solar Energy*, 85(5), 1109–1118. doi: 10.1016/j.solener.2011.03.007
- [123] Hassan, A.K., Hasan, M.M., & Khan, M.E. (2022). Heat transfer and fluid flow investigation in dimple roughened solar duct arranged in multiple arc pattern. *International Journal of Ambient Energy*, 43(1), 1866–1877. doi: 10.1080/01430750.2020.1722742
- [124] Alam, T., & Kim, M.H. (2017). Heat transfer enhancement in solar air heater duct with conical protrusion roughness ribs. *Applied Thermal Engineering*, 126, 458–469. doi: 10.1016/j.applthermaleng.2017.07.181
- [125] Lanjewar, A., Bhagoria, J.L., & Sarviya, R.M. (2011). Heat transfer and friction in solar air heater duct with W-shaped rib roughness on absorber plate. *Energy*, 36(7), 4531–4541. doi: 10.1016/j.energy.2011.03.054
- [126] Kumar, A., Bhagoria, J.L., & Sarviya, R. M. (2009). Heat transfer and friction correlations for artificially roughened solar air heater duct with discrete W-shaped ribs. *Energy Conversion and Management*, 50(8), 2106–2117. doi: 10.1016/j.enconman.2009.01.025
- [127] Thakur, S., & Thakur, N.S. (2019). Investigational analysis of roughened solar air heater channel having W-shaped ribs with symmetrical gaps along with staggered ribs. *Energy Sources, Part A: Recovery, Utilization, and Environmental Effects*, 42(7), 886–901. doi: 10.1080/15567036.2019.1675815
- [128] Thakur, S., & Thakur, N.S. (2020). Impact of multi-staggered rib parameters of the ‘W’ shaped roughness on the performance of a solar air heater channel. *Energy Sources, Part A: Recovery, Utilization, and Environmental Effects*, 43(8), 960–979. doi: 10.1080/15567036.2020.1764672
- [129] Murmu, R., Kumar, P., & Singh, H. (2018). Experimental investigation on heat transfer and friction factor for an inclined spherical ball roughened solar air heater. *Instrumentation Mesure Métrologie*, 18(1), 7–36. doi: 10.3166/i2m.17.7-36
- [130] Bhagoria, J.L., Saini, J.S., & Solanki, S.C. (2002). Heat transfer coefficient and friction factor correlations for rectangular solar air heater duct having transverse wedge shaped rib roughness on the absorber plate. *Renewable Energy*, 25(3), 341–369. doi: 10.1016/S0960-1481(01)00057-X
- [131] Barik, A.K., Mohanty, A., Senapati, J.R., & Awad, M.M. (2021). Constructal design of different ribs for thermo-fluid performance enhancement of a solar air heater (SAH). *International Journal of Thermal Sciences*, 160, 106655. doi: 10.1016/j.ijthermalsci.2020.106655
- [132] Kumar, R., & Goel, V. (2021). Unconventional solar air heater with triangular flow-passage: A CFD based comparative performance assessment of different cross-sectional rib-roughnesses. *Renewable Energy*, 172, 1267–1278. doi: 10.1016/j.renene.2021.03.068
- [133] Zina, B., Filali, A., Benamara, N., Laouedj, S., & Ahmed, H. (2020). Numerical simulation of heat transfer improvement of a new designed artificially roughened solar air heater using triangular ribs with semi-circular nooks. *Energy Sources, Part A: Recovery, Utilization, and Environmental Effects*, 43(22), 2820–2838. doi: 10.1080/15567036.2020.1825564
- [134] Gawande, V.B., Dhoble, A.S., Zodpe, D.B., & Mangrulkar, C. (2020). A comparative analysis of thermo-hydraulic performance of a roughened solar air heater using various rib shapes. *Australian Journal of Mechanical Engineering*, 18(3), 331–350. doi: 10.1080/14484846.2018.1525171

- [135] Aharwal, K.R., Gandhi, B.K., & Saini, J.S. (2008). Experimental investigation on heat-transfer enhancement due to a gap in an inclined continuous rib arrangement in a rectangular duct of solar air heater. *Renewable Energy*, 33(4), 585–596. doi: 10.1016/j.renene.2007.03.023
- [136] Aharwal, K.R., Gandhi, B.K., & Saini, J.S. (2009). Heat transfer and friction characteristics of solar air heater ducts having integral inclined discrete ribs on absorber plate. *International Journal of Heat and Mass Transfer*, 52(25–26), 5970–5977. doi: 10.1016/j.ijheatmasstransfer.2009.05.032
- [137] Kumar, R., Nadda, R., Kumar, S., Razak, A., Sharifpur, M., Aybar, H.S., Saleel, C.A., & Afzal, A. (2022). Influence of artificial roughness parametric variation on thermal performance of solar thermal collector: An experimental study, response surface analysis and ANN modelling. *Sustainable Energy Technologies and Assessments*, 52(A), 102047. doi: 10.1016/j.seta.2022.102047
- [138] Kumar, A., & Layek, A. (2020). Evaluation of the performance analysis of an improved solar air heater with Winglet shaped ribs. *Experimental Heat Transfer*, 34(4), 358–377. doi: 10.1080/08916152.2020.1838670
- [139] Nidhul, K., Yadav, A.K., Anish, S., & Arunachala, U.C. (2022). Thermo-hydraulic and exergetic performance of a cost-effective solar air heater: CFD and experimental study. *Renewable Energy*, 184, 627–641. doi: 10.1016/j.renene.2021.11.111
- [140] Parsa, H., Saffar-Avval, M., & Hajmohammadi, M.R. (2021). 3D simulation and parametric optimization of a solar air heater with novel staggered cuboid baffles. *International Journal of Mechanical Sciences*, 205, 106607. doi: 10.1016/j.ijmecsci.2021.106607
- [141] Jaurker, A.R., Saini, J.S., & Gandhi, B.K. (2006). Heat transfer and friction characteristics of rectangular solar air heater duct using rib-grooved artificial roughness. *Solar Energy*, 80(8), 895–907. <https://doi.org/10.1016/j.solener.2005.08.006>
- [142] Karmare, S.V., & Tikekar, A.N. (2009). Experimental investigation of optimum thermohydraulic performance of solar air heaters with metal rib grits roughness. *Solar Energy*, 83(1), 6–13. doi: 10.1016/j.solener.2008.05.017
- [143] Karmare, S.V., & Tikekar, A.N. (2007). Heat transfer and friction factor correlation for artificially roughened duct with metal grit ribs. *International Journal of Heat and Mass Transfer*, 50(21–22), 4342–4351. doi: 10.1016/j.ijheatmasstransfer.2007.01.065
- [144] Bopche, S.B., & Tandale, M.S. (2009). Experimental investigations on heat transfer and frictional characteristics of a turbulator roughened solar air heater duct. *International Journal of Heat and Mass Transfer*, 52(11–12), 2834–2848. doi: 10.1016/j.ijheatmasstransfer.2008.09.039
- [145] Kumar, S., & Verma, S.K. (2022). Heat transfer and fluid flow analysis of sinusoidal protrusion rib in solar air heater. *International Journal of Thermal Sciences*, 172, 107323. doi: 10.1016/j.ijthermalsci.2021.107323
- [146] Saravanan, A., Murugan, M., Sreenivasa Reddy, M., Ranjit, P.S., Elumalai, P.V., Kumar, P., & Rama Sree, S. (2021). Thermo-hydraulic performance of a solar air heater with staggered C-shape finned absorber plate. *International Journal of Thermal Sciences*, 168, 107068. doi: 10.1016/j.ijthermalsci.2021.107068
- [147] Bezbaruah, P.J., Das, R.S., & Sarkar, B.K. (2021). Experimental and numerical analysis of solar air heater accoutered with modified conical vortex generators in a staggered fashion. *Renewable Energy*, 180, 109–131. doi: 10.1016/j.renene.2021.08.046
- [148] Goel, V., Kumar, R., Bhattacharyya, S., Tyagi, V.V., & Abu-sorrah, A.M. (2021). A comprehensive parametric investigation of hemispherical cavities on thermal performance and flow-dynamics in the triangular-duct solar-assisted air-heater. *Renewable Energy*, 173, 896–912. doi: 10.1016/j.renene.2021.04.006
- [149] Yadav, S., & Saini, R.P. (2020). Numerical investigation on the performance of a solar air heater using jet impingement with absorber plate. *Solar Energy*, 208, 236–248. doi: 10.1016/j.solener.2020.07.088
- [150] Wang, D., Liu, J., Liu, Y., Wang, Y., Li, B., & Liu, J. (2020). Evaluation of the performance of an improved solar air heater with ‘S’ shaped ribs with gap. *Solar Energy*, 195, 89–101. doi: 10.1016/j.solener.2019.11.034
- [151] Gupta, D., Solanki, S.C., & Saini, J.S. (1993). Heat and fluid flow in rectangular solar air heater ducts having transverse rib roughness on absorber plates. *Solar Energy*, 51(1), 31–37. doi: 10.1016/0038-092X(93)90039-Q
- [152] Kumar, R., Kumar, A., & Varun. (2017). Computational fluid dynamics based study for analyzing heat transfer and friction factor in semi-circular rib-roughened equilateral triangular duct. *International Journal of Numerical Methods for Heat & Fluid Flow*, 27(4), 941–957. doi: 10.1108/HFF-10-2015-0438
- [153] Kumar, R., Kumar, A., & Goel, V. (2017). A parametric analysis of rectangular rib roughened triangular duct solar air heater using computational fluid dynamics. *Solar Energy*, 157, 1095–1107. doi: 10.1016/j.solener.2017.08.071
- [154] Sahu, M.K., & Prasad, R.K. (2017). Thermohydraulic performance analysis of an arc shape wire roughened solar air heater. *Renewable Energy*, 108, 598–614. doi: 10.1016/j.renene.2017.02.075
- [155] Saravanakumar, P.T., Somasundaram, D., & Matheswaran, M.M. (2019). Thermal and thermo-hydraulic analysis of arc shaped rib roughened solar air heater integrated with fins and baffles. *Solar Energy*, 180, 360–371. doi: 10.1016/j.solener.2019.01.036
- [156] Murmu, R., Kumar, P., & Singh, H.N. (2019). Effects of roughness parameters on the heat transfer and friction of an inclined spherical ball roughened flat plate solar collector. *International Journal of Ambient Energy*, 42(15), 1781–1797. doi: 10.1080/01430750.2019.1693426
- [157] Kumar, V., & Murmu, R. (2021). Experimental investigation for thermal performance of inclined spherical ball roughened solar air duct. *Renewable Energy*, 172, 1365–1392. doi: 10.1016/j.renene.2020.11.150
- [158] Karwa, R., Solanki, S.C., & Saini, J.S. (1999). Heat transfer coefficient and friction factor correlations for the transitional flow regime in rib-roughened rectangular ducts. *International Journal of Heat and Mass Transfer*, 42(9), 1597–1615. doi: 10.1016/S0017-9310(98)00252-X
- [159] Patel, Y.M., Jain, S.V., & Lakhera, V.J. (2021). Thermo-hydraulic performance analysis of a solar air heater roughened with discrete reverse NACA profile ribs. *International Journal of Thermal Sciences*, 167, 107026. doi: 10.1016/j.ijthermalsci.2021.107026
- [160] Arunkumar, H.S., Kumar, S., & Karanth, K.V. (2021). Experimental study on thermo-hydraulic performance of a solar air heater with rectangular perforated duct inserts. *Solar Energy*, 227, 179–189. doi: 10.1016/j.solener.2021.09.005
- [161] Kumar, R., Kashyap, A.S., Singh, P., Goel, V., & Kumar, K. (2020). Innovatively arranged curved-ribbed solar-assisted air heater: Performance and correlation development for heat and flow characteristics. *Journal of Solar Energy Engineering*, 142(3), 031008. doi: 10.1115/1.4045827
- [162] Hong, Y.J., & Hsieh, S.S. (1991). An experimental investigation of heat transfer characteristics for turbulent flow over staggered ribs in a square duct. *Experimental Thermal and Fluid Science*, 4(6), 714–722. doi: 10.1016/0894-1777(91)90078-6
- [163] Gupta, D., Solanki, S.C., & Saini, J.S. (1997). Thermohydraulic performance of solar air heaters with roughened absorber plates. *Solar Energy*, 61(1), 33–42. doi: 10.1016/S0038-092X(97)00005-4
- [164] Layek, A., Saini, J.S., & Solanki, S.C. (2007). Second law optimization of a solar air heater having chamfered rib-groove roughness on absorber plate. *Renewable Energy*, 32(12), 1967–1980. doi: 10.1016/j.renene.2006.11.005
- [165] Varun, Saini, R.P., & Singal, S.K. (2008). Investigation of thermal performance of solar air heater having roughness elements as a combination of inclined and transverse ribs on the absorber plate. *Renewable Energy*, 33(6), 1398–1405. doi: 10.1016/j.renene.2007.07.013
- [166] Muluwork, K.B., Saini, J.S., & Solanki, S.C. (1998). Studies on discrete rib roughened solar air heaters. In *Proceedings of National Solar Energy Convention* (pp. 75–84), 30 Nov. – 2 Dec., Roorkee, USA.
- [167] Hans, V.S., Saini, R.P., & Saini, J.S. (2010). Heat transfer and friction factor correlations for a solar air heater duct roughened artificially with multiple v-ribs. *Solar Energy*, 84(6), 898–911. doi: 10.1016/j.solener.2010.02.004
- [168] Singh, S., Chander, S., & Saini, J.S. (2012). Investigations on thermo-hydraulic performance due to flow-attack-angle in V-down rib with gap in a rectangular duct of solar air heater. *Applied Energy*, 97, 907–912. doi: 10.1016/j.apenergy.2011.11.090
- [169] Kumar, A., Saini, R.P., & Saini, J.S. (2013). Development of correlations for Nusselt number and friction factor for solar air heater with roughened duct having multi v-shaped with gap rib as artificial roughness. *Renewable Energy*, 58, 151–163. doi: 10.1016/j.renene.2013.03.013
- [170] Bhushan, B., & Singh, R. (2012). Thermal and thermohydraulic performance of roughened solar air heater having protruded absorber plate. *Solar Energy*, 86(11), 3388–3396. doi: 10.1016/j.solener.2012.09.004
- [171] Lanjewar, A., Bhagoria, J.L., & Sarviya, R.M. (2011). Experimental study of augmented heat transfer and friction in solar air heater with different orientations of W-Rib roughness. *Experimental Thermal and Fluid Science*, 35(6), 986–995. doi: 10.1016/j.expthermflusci.2011.01.019
- [172] Chandra, P.R., Alexander, C.R., & Han, J.C. (2003). Heat transfer and friction behaviors in rectangular channels with varying number of ribbed walls. *International Journal of Heat and Mass Transfer*, 46(3), 481–495. doi: 10.1016/S0017-9310(02)00297-1
- [173] Webb, R.L., Eckert, E.R.G., & Goldstein, R.J. (1971). Heat transfer and friction in tubes with repeated-rib roughness. *International Journal of Heat and Mass Transfer*, 14(4), 601–617. doi: 10.1016/0017-9310(71)90009-3
- [174] Srivastava, A., Chhaparwal, G.K., & Sharma, R.K. (2020). Numerical and experimental investigation of different rib roughness in a solar air heater. *Thermal Science and Engineering Progress*, 19, 100576. doi: 10.1016/j.tsep.2020.100576
- [175] Kumar, R., Goel, V., & Kumar, M. (2020). Effect of providing gap in multiple-arc rib-roughened solar air heater – Part 1. *Journal of Mechanical Science and Technology*, 34(6), 2619–2625. doi: 10.1007/s12206-020-0535-3
- [176] Alam, T., & Kim, M.H. (2018). A comprehensive review on single phase heat transfer enhancement techniques in heat exchanger applications. *Renewable and Sustainable Energy Reviews*, 81, 813–839. doi: 10.1016/j.rser.2017.08.060



Co-published by
Institute of Fluid-Flow Machinery
Polish Academy of Sciences
Committee on Thermodynamics and Combustion
Polish Academy of Sciences

Copyright©2026 by the Authors under licence CC BY-NC-ND 4.0

<http://www.imp.gda.pl/archives-of-thermodynamics/>



Three-dimensional modelling of combustion and emissions characteristics in direct-injection diesel engines using detailed chemistry based on the SAGE combustion model

Lamia Medjahed^a, Hamza Bousbaa^{b*}, Mohammed Benramdane^a,
Bouchra Nor El Houda Hagani^b

^aMechanical Department, Technology Faculty, University of Tlemcen, ETAP Laboratory, Algeria

^bNational Polytechnic school Maurice Audin, Mechanical Department, LTE Laboratory, BP 1523 El Menaouer, 31000-Oran, Algeria

*Corresponding author email: hamza.bousbaa@enp-oran.dz

Received: 07.10.2025; revised: 03.05.2026; accepted: 12.05.2026

Abstract

This study employed a three-dimensional CFD model with integrated chemical kinetics using CONVERGE CFD software. We used a detailed mechanism of 42 species and 167 reactions and compared its predictions with a reduced 77-species mechanism developed by Liu and Zhang (2023), both tailored for n-heptane oxidation. Validation of diesel-injection combustion was carried out by comparing the model results with experimental engine data. Numerical simulations were conducted on an MKDIR 620-145 engine at 1400 rpm and 50% load. The chosen models reproduced the mean in-cylinder pressure accurately, with deviations below 4.7%. Analysis of species and thermal fields revealed that key intermediate species emerge near 930 K, where H₂O₂ concentrations become sufficient to trigger auto-ignition and the heat release rate reaches its peak. At higher temperatures (around 1300 K), OH and CO concentrations are notably elevated. Soot production is initiated early in combustion and is dominated by diffusion processes; the peak net soot mass was 3.25×10⁻⁷ kg at 18° before the top dead centre. Throughout combustion, CO₂, H₂O and hydrocarbon concentrations increase markedly while O₂ is consumed, with CO₂ and H₂O rising rapidly during both premixed and diffusion combustion phases. NO_x formation closely follows the temperature history, approaching a saturation level near 1.1×10⁻⁴ kg under the studied conditions. Overall, the results indicate that the employed reaction mechanisms capture the temporal evolution of combustion, species formation, soot production and emission trends with good fidelity, supporting their suitability for studying fuel behaviour and emissions in diesel engine environments.

Keywords: CONVERGE CFD software; Diesel engine; Modelling; Chemistry approach; Combustion

Vol. 47(2026), No. 2, 59–68; doi: 10.24425/ather.2026.158681

Cite this manuscript as: Medjahed, L., Bousbaa, H., Benramdane, M., & Hagani, B.N.E.H. (2026). Three-dimensional modelling of combustion and emissions characteristics in direct-injection diesel engines using detailed chemistry based on the SAGE combustion model. *Archives of Thermodynamics*, 47(2), 59–68.

1. Introduction

As the source of energy, propulsion and power, internal combustion (IC) engines are an extremely important equipment. Compression ignition (CI) engines are the most powerful and capable when compared to spark ignition (SI) engines, especially for off-road power generation, marine applications, and large and medium-sized vehicles on the road [1,2]. CI engines

become more and more popular because of their exceptional performance, consistent dependability, long lifespan, increased fuel efficiency, and significantly reduced emissions of carbon dioxide (CO₂), carbon monoxide (CO) and hydrocarbons (HCs) compared to SI engines [3–5]. However, it must be acknowledged that CI engines do encounter challenges such as noisy combustion, elevated NO_x emissions, disruptive vibrations, and the release of harmful smoke when compared to SI engines [1,2].

Nomenclature

A_r – pre-exponential factor
 b_r – temperature exponent
 $\bar{C}_{p,m}$ – molar constant-pressure specific heat, J/(mol·K)
 C_μ – model constant
 D – diffusion coefficient
 e – internal energy, J
 E_r – activation energy, kJ/mol
 h_m – species enthalpy, J/mol
 $\bar{h}_m$ – molar specific enthalpy, J/(mol·K)
 I – initial turbulence intensity
 k – total kinetic energy, J
 K – conductivity, W/(m·K)
 K_{cr} – equilibrium coefficient
 le – length scale, m
 P – pressure, kPa
 Pr – Prandtl number
 q_i – rate-of-progress variable
 R_u – universal gas constant, J/(mol·K)
 S – source term
 S_m – source terms of species m
 t – time, s
 T – temperature, K
 u, U – velocity, m/s
 $[X_m]$ – molar concentration of species m
 Y_m – mass fraction of specie m

Greek symbols

ε – turbulent dissipation, m²/s³
 μ_t – turbulent viscosity, Pa s
 ρ – density, kg/m³

σ_{ij} – stress tensor, Pa
 $v'_{m,r}, v''_{m,r}$ – stoichiometric coefficients for the reactants
 $\dot{\omega}_m$ – production rate of species, 1/s

Subscripts and Superscripts

r – reaction
 m – species

Abbreviations and Acronyms

AMR – adaptive mesh refinement
 BTDC – before top dead centre
 CA – crank angle
 CAD – computer-aided design
 CFD – computational fluid dynamics
 CI – compression ignition
 CO – carbon monoxide
 CO₂ – carbon dioxide
 CNG – compressed natural gas
 CSP – computational singular perturbation
 EGR – exhaust gas recirculation
 HCCI – homogeneous charge compression ignition
 HCs – hydrocarbons
 IC – internal combustion
 IDT – ignition delay time
 KH-RT – Kelvin-Helmholtz and Rayleigh-Taylor
 LLNL – Lawrence Livermore National Laboratory's
 MB-NH – methylbutanoate/n-heptane
 PRF – principal reference fuel
 QSS – quasi-steady-state
 SCR – selective catalytic reduction
 SI – spark ignition
 TDC – top dead centre

Nevertheless, significant advancements have been made in recent times to address these environmental concerns. Various methods have been developed for the reduction of these polluting gases, including exhaust gas recirculation (EGR) [6,7], optimisation of injection systems [8,9], utilisation of AdBlue in selective catalytic reduction (SCR) techniques [10–13], the management of turbulence and chemistry interaction [14,15], adoption of dual fuel engines [16–18], the introduction of biofuels injections [19–24], and using compressed natural gas (CNG) [25]. These innovative approaches hold immense potential for mitigating the adverse effects associated with CI engines, thereby paving the way for a more refined and sustainable future in automotive technology. The advancement of computational fluid dynamics (CFD) methodology for internal combustion engine design presents a distinctive and formidable challenge compared to traditional experimental approaches, owing to the intricate nature of the physics and mechanics involved. Enhancing our comprehension in this field is imperative in order to unlock innovative solutions, streamline costs, and augment development efficiency. While notable progress has been achieved in various domains such as turbulence, spray and combustion modelling, pre- and post-processing, etc., there are still numerous additional prerequisites to be fulfilled for CFD to truly emerge as a reliable and comprehensive design tool [26,27]. The CONVERGE software [28] is widely used in engine research and is

popular among universities that train engineers and researchers. It solves complex equations related to fluid dynamics, taking into account factors like turbulence and chemical reactions. A detailed chemistry extension has also been developed to address the need for reducing emissions from internal combustion engines. This extension allows for more accurate simulation of engine exhaust gases. Three-dimensional conservation equations are successfully solved using CONVERGE CFD, both with and without chemical reactions. The code and its functioning have been thoroughly documented by Richards et al. [28]. In response to the need for reducing polluting emissions from internal combustion engines, a comprehensive chemistry extension for CONVERGE has been developed. This extension allows for the accurate representation of engine exhaust gas composition, including radicals, NO_x species and soot.

In this context, different numerical studies have been carried out. For instance, a simplified mechanism with 60 species and 237 reactions was used for the spray combustion simulation in the study [29]. Similar to this, Wan Manson and Olsen [30] used the CONVERGE CFD model to study the combustion and emissions of a gas-diesel dual fuel. They used a simplified chemical kinetic mechanism that had 393 reactions for n-heptane, methane, ethane and propane, and 86 species. In a different research, Ra and Reitz [31] used the n-heptane mechanism in conjunction with a reduced mechanism for iso-octane oxidation to

create a principal reference fuel (PRF) mechanism that included 130 reactions and 41 species. Direct injection diesel engines and homogeneous charge compression ignition (HCCI) engines were used in the trials to verify the PRF mechanism.

Wang and Frenklach [32] employed a comprehensive mechanism that combined the oxidation chemistry of skeletal n-heptane with the kinetics of aromatic formation in fuel-rich acetylene flames, comprising 117 species and 602 reactions. They validated the combustion simulations through detailed numerical kinetic analysis and by comparison with a reduced mechanism of 60 species and 237 reactions. Liu et al. [33] applied graph-based reduction techniques to a detailed MB-NH (methylbutanoate/n-heptane) model containing 661 species and 3019 reactions, producing a reduced mechanism with 145 species and 869 reactions. Combustion simulations performed in CONVERGE software using this reduced chemistry showed good agreement with experimental data for biodiesel blends, indicating the model's suitability for predicting combustion behaviour and emissions trends in compression-ignition engines. Liu and Zhang [34] (2023) developed a reduced diesel mechanism derived from Wang's 2015 mechanism (109 species, 543 reactions) [35]. They first defined an importance index using computational singular perturbation (CSP) to identify and remove insignificant reactions. By applying a threshold of 0.0065, they eliminated 140 reactions, reducing the mechanism to 403 reactions. Using a 0.005 threshold, they also identified 32 global quasi-steady-state (QSS) species. The final reduced mechanism contains 77 species and, when compared with Wang's original mechanism, shows a maximum deviation of 7.2% in ignition delay. They further validated the mechanism by simulating an HCCI engine case and found that the simulation results agree closely with the original mechanism, with errors under 10%.

Yin et al. [36] developed a simplified chemical kinetic mechanism for toluene within a gasoline surrogate (toluene, n-heptane, iso-octane). They reduced the original mechanism from 137 species and 633 reactions to 60 species and 234 reactions. The simplified mechanism accurately predicts ignition delay times and laminar flame speeds according to their results.

Hu et al. [37] proposed a four-step global model for moderate and intensive low-oxygen dilution (MILD) oxy-combustion in 2018. They validated it against two detailed mechanisms across three combustion conditions and compared its performance with five other global mechanisms. The results showed that, under MILD, oxy-fuel and MILD oxy-combustion conditions, the proposed model substantially improved predictions of peak and equilibrium concentrations of the main species. However, the global model was validated only against detailed mechanisms for some reaction conditions where no experimental data were available. While the global model reproduced trends similar to the two detailed mechanisms across the three conditions, the absolute species concentration values differed notably among the three approaches.

Baigmohammadi et al. [38] recently developed a detailed mechanism, NUIGMech1.1, to investigate ignition delay times (IDTs) of C1-C2 hydrocarbons over a range of temperatures, pressures and equivalence ratios. NUIGMech1.1 comprises 11 274 elementary reactions and 2746 species. Its accuracy was

assessed by comparison with experimental IDT data, and it was found to reproduce the measured IDTs well.

The main objective of this study is to validate the simplified reaction mechanism of a diesel fuel. This mechanism was created by simulating the actual physical properties of the fuel and using the chemical kinetics of n-heptane, consisting of 42 species and 167 reactions. We compared it with the reduced mechanism of 77 species developed by Liu and Zhang [34].

2. Mathematical formulations

Equations describing the conservation of mass, momentum and energy guide the modelling of combustion in diesel engines. The turbulence and the movement of passive scalars and organisms are described by additional equations. The mathematical formulas utilised in CONVERGE are described in this context in [28].

The following represents the compressible equation for species mass transport:

$$\frac{\partial \rho_m}{\partial t} + \frac{\partial u_j \rho_m}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\rho D \frac{\partial Y_m}{\partial x_j} \right) + S_m. \quad (1)$$

The total fluid density equation is obtained by adding the preceding equation for each species:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = S, \quad (2)$$

where $\rho_m = Y_m \rho$. In the above equations, the velocity is represented by u , density by ρ , species density by ρ_m , mass fraction of species m by Y_m , mass diffusion coefficient by D , and source terms by S and S_m .

Hence, the following is the equation of momentum transport for the compressible fluid:

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_j} + S_i. \quad (3)$$

The energy equation for the compressible fluid is given by:

$$\begin{aligned} \frac{\partial \rho e}{\partial t} + \frac{\partial u_j \rho e}{\partial x_j} = & -P \frac{\partial u_j}{\partial x_i} + \sigma_{ij} \frac{\partial u_i}{\partial x_j} + \frac{\partial}{\partial x_j} \left(K \frac{\partial T}{\partial x_j} \right) + \\ & + \frac{\partial}{\partial x_j} \left(\rho D \sum_m h_m \frac{\partial Y_m}{\partial x_j} \right) + S. \end{aligned} \quad (4)$$

In the above equations, P is the pressure, T is the temperature, σ_{ij} is the stress tensor, S_i and S are source terms, h_m is the species enthalpy, K is the conductivity, D is the diffusivity, and e is the specific internal energy.

The conductivity is substituted with the turbulent conductivity provided by:

$$K_t = K + c_p \frac{\mu_t}{Pr}. \quad (5)$$

where Pr and μ_t are the Prandtl number and turbulent viscosity, respectively.

In the initial stages of the pressure-implicit with splitting of operators (PISO) loop, CONVERGE computes the momentum and pressure to determine the velocity for the other transport equations. It is essential to verify the PISO loop convergence after each iteration of the PISO process.

3. SAGE chemical kinetics model

The following section discusses the incorporation of the SAGE model within the CONVERGE software, specifically focusing on chemistry modelling. The inclusion of the SAGE model allows for the integration of detailed chemical kinetics in combustion applications. This is accomplished by using a collection of input files in the CHEMKIN format, which is now the accepted format for describing chemical mechanisms in this context.

A mechanism for a multi-step chemical reaction may be expressed as follows [28]:

$$\sum_{m=1}^M v'_{m,r} x_m \Leftrightarrow \sum_{m=1}^M v''_{m,r} x_m, \quad \text{for } r = 1, 2, \dots, R. \quad (6)$$

For species m and reaction r , the stoichiometric coefficients for the reactants and products are denoted by $v'_{m,r}$ and $v''_{m,r}$, respectively. The chemical symbol for species m is represented by x_m . The following provides the production rate of species m :

$$\dot{\omega}_m = \sum_{r=1}^R v_{m,r} q_r, \quad \text{for } m = 1, 2, \dots, M, \quad (7)$$

$$v_{m,r} = v''_{m,r} - v'_{m,r},$$

and the rate-of-progress variable q_i for the r^{th} reaction is:

$$q_r = k_{fr} \prod_{m=1}^M [X_m]^{v'_{m,r}} - k_{rr} \prod_{m=1}^M [X_m]^{v''_{m,r}}, \quad (8)$$

where k_{fr} and k_{rr} are the forward and reverse rate coefficients for reaction r , and $[X_m]$ is the molar concentration of species m .

The Arrhenius form is used in the SAGE setup to represent the forward rate coefficient:

$$k_{fr} = A_r T^{b_r} e^{(-E_r/R_u T)}, \quad (9)$$

where R_u is the universal gas constant, E_r is the activation energy, b_r is the temperature exponent, and A_r is the pre-exponential factor.

Additionally, the reverse rate coefficient can be computed using the equilibrium coefficient K_{cr} , or it can be supplied in an equivalent mode:

$$k_{rr} = k_{fr}/K_{cr}. \quad (10)$$

The thermodynamic characteristics are used to calculate the equilibrium coefficient, K_{cr} .

The governing equations for mass and energy conservation for a particular computational cell may be solved using the information provided. These are the equations:

$$\frac{d[X_m]}{dt} = \dot{\omega}_m \quad \text{and} \quad \frac{dT}{dt} = \frac{V \frac{dP}{dt} - \sum_m (\bar{h}_m \dot{\omega}_m)}{\sum_m ([X_m] \bar{c}_{p,m})}, \quad (11)$$

where the molar specific enthalpy, molar constant-pressure specific heat, volume, temperature and pressure of species m are represented by the variables $\bar{h}_m$, $\bar{c}_{p,m}$, V , T and P , respectively. In CONVERGE, the mech.dat and therm.dat data files define the mechanisms and thermodynamic data for the SAGE model input parameters. Every computational time step in the CONVERGE program solves the aforementioned equations, and the species are updated appropriately. Please note that the temperature acquired is not utilised to update the cell temperature in CONVERGE, but solely to update the rate coefficients in the

SAGE system of rate equations. After applying the determined species concentrations to converge the thorough chemistry computation, the cell temperature is updated.

4. Engine description and computational domain

The model of the combustion chamber was initially created using SOLIDWORKS, a computer-aided design (CAD) software, before being exported to the preprocessing software CONVERGE CFD for geometry verification and clean-up. Figure 1 illustrates the combustion chamber model generated with Tecplot Graphic. The engine used in our work is a heavy-duty, 6-cylinder, 4-stroke diesel engine with a piston cylinder bow, model number: MK DIR 620-145 whose parameters are gathered in Table 1. It is equipped with a high-pressure injector pump system and a turbocharging system. The simulations run at 1400 rpm, 50% load, and the injection begins at 8° before TDC.

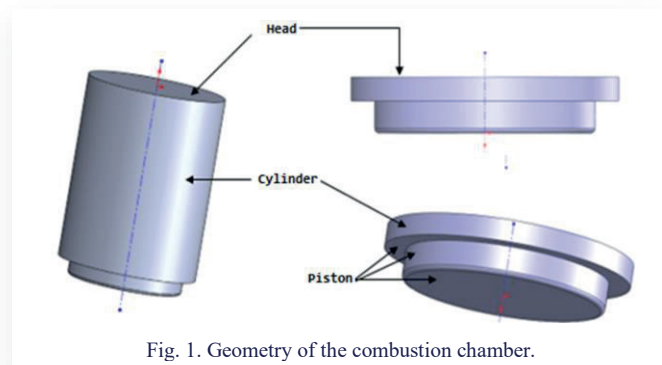


Fig. 1. Geometry of the combustion chamber.

Specifications for the engine combustion chamber and injectors are listed in Table 1. Table 2 enumerates the critical injection requirements, and Fig. 2 depicts the injection profile.

Table 1. Engine specifications.

Engine type	MKDIR 620-145
Chamber type	Vertical valve
Number of cylinders	06
Bore	12 cm
Stroke	14.5 cm
Connecting rod length	22.7 cm
Squish	0.17 cm
Compression ratio	17
Engine speed	1400 rpm
Maximum torque	158 N m at 1200 rpm
Turbocharger	Schwitzer type S2BV
Injection pump	Bosch type P.7100
EGR	0%
Swirl	3.11

Table 2 Injection specification.

Fuel type	n-heptane
Initial fuel temperature	344 K
Injection duration	21°
Fuel Injection rate	0.09 g/cycle
Injection start	8 c.a. degree BTDC
Nozzle diameter	2.59e-01 mm
Nozzles number	2

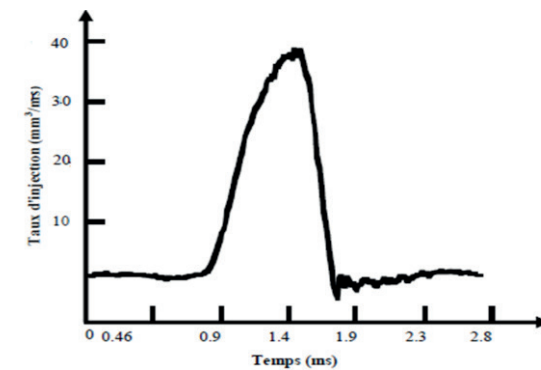


Fig. 2. Injection profile.

In the present work, the detailed boundary conditions (temperature of cylinder, head and piston), and initial conditions (pressure and temperature in the chamber) used in the software are:

- the boundary conditions:
 - $T_{cy} = 433\text{ K}$,
 - $T_{head} = 523\text{ K}$,
 - $T_p = 523\text{ K}$,
- the initial conditions:
 - $P_{ch} = 1.765\text{ MPa}$,
 - $T_{ch} = 355\text{ K}$.

The process of generating a complex geometry grid in different CFD codes can be quite time-consuming for users, which makes projects unfeasible due to financial or time constraints. The code CONVERGE CFD allows the calculation of three-dimensional, compressible, chemically-reacting fluid flows in complex geometries with moving boundaries, and it is specifically tailored for piston engine analysis. In this study, we used two techniques: basic meshing and the adaptive mesh refinement (AMR) technique to manage and refine the grid for temperature, emissions, and velocity.

The effects of the base grid size in all directions (4 mm, 2.5 mm, 1.8 mm and 1.5 mm) on the combustion are shown in Table 3. AMR was adopted based on the temperature, velocity and near nozzle region cell size of 0.2 mm. According to the mesh sensitivity study, in each of the three dimensions, the base grid size was maintained at 1.8 mm with an error in cylinder pressure of no more than 4.7%.

Table 3. Mesh details.

Base grid size [mm]	AMR [mm]	Error [%]
4	0.2	7.8
2.5	0.2	5.6
1.8	0.2	4.7
1.5	0.2	4.4

A 1.8 mm base grid was applied throughout the computational domain. Owing to piston motion, the total number of computational cells varied over the engine cycle, reaching approximately 589 133 cells at the bottom dead centre (BDC, 180 CA) and decreasing to about 10 154 cells at the top dead centre (TDC, 0 CA). Figure 3 presents the comparison of cylinder pressure

from the experiment and simulation, whereas the mesh of the engine domain is illustrated in Fig. 4.

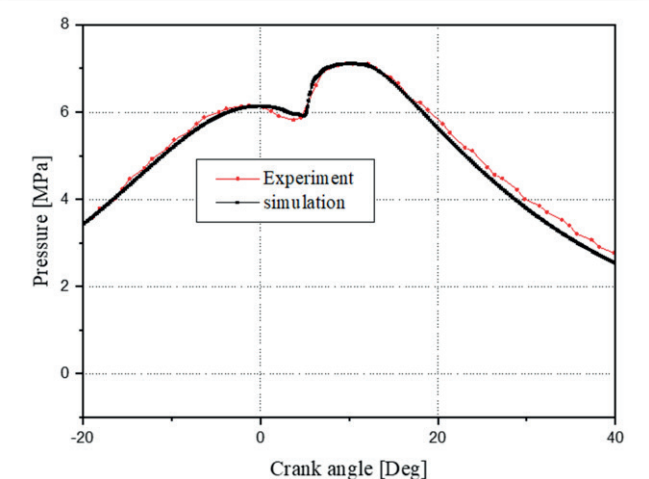


Fig. 3. Experimental and simulated evolution of cylinder pressure.

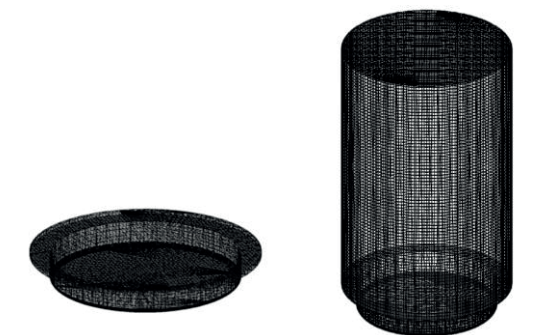


Fig. 4. Computational domain of the engine.

5. Reduced chemical kinetic reaction mechanism

N-heptane is often chosen as a fuel because it has a cetane number of around 56, which is comparable to that of conventional diesel fuel. Heptyl radicals, C_7H_{15} , formed from n-heptane through H-abstraction from fuel molecules, play a significant role in the oxidation process. It is crucial to accurately account for the number of carbon and hydrogen atoms in the final combustion products when calculating the overall heat release of oxidation reactions. Therefore, it is necessary for the simplified reaction mechanism to align with the overall chemical equilibrium state.

In this study, a reduced reaction mechanism comprising 167 chemical reactions and 42 species was considered. For a more detailed mechanism, we recommend consulting the website of the Lawrence Livermore National Laboratory (LLNL), specifically the archived mechanisms available through the U.S. Department of Energy National Nuclear Security Administration (LLNL website). Table 4 provides an excerpt of the n-heptane reaction mechanism. Additional information on reaction mechanisms can be found on the same website. Moreover, alternative literature sources offer other n-heptane mechanisms, such as a comprehensive mechanism with 544 species and 2446 reactions (LLNL), a detailed mechanism with 117 species and

602 reactions, and a reduced mechanism with 60 species and 237 reactions [35].

Table 4. Single column table example.

	Reaction	A_r	b_r	E_r
R1	$C_7H_{16} + H = C_7H_{15} + H_2$	5.600e+07	2.0	7667.0
R2	$C_7H_{16} + H = C_7H_{15} + H_2$	4.380e+07	2.0	4750.0
R3	$C_7H_{16} + OH = C_7H_{15} + H_2O$	8.610e+09	1.1	1815.0
...	...	...	...	...
R6	$C_7H_{16} + HO_2 = C_7H_{15} + H_2O_2$	1.650e+13	0.0	16950.0
R7	$C_7H_{16} + O_2 = C_7H_{15} + HO_2$	2.500e+13	0.0	48810.0
...	...	...	...	...
R9	$C_7H_{15} + O_2 = C_7H_{15}O_2$	2.000e+12	0.0	0.0
R10	$C_7H_{15} + O_2 = C_7H_{15}O_2$	2.000e+12	0.0	0.0
R11	$C_7H_{15}O_2 = C_7H_{14}O_2H$	6.000e+11	0.0	20380.0
R12	$C_7H_{14}O_2H + O_2 = C_7H_{14}O_2HO_2$	4.600e+11	0.0	0.0
R13–R158	...	...	...	...
R159	$C_2H_3 + H = C_2H_2 + H_2$	4.000e+13	0.0	0.0
R160	$C_2H_3 + O_2 = CH_2O + HCO$	4.000e+12	0.0	-250.0
R161	$C_2H_3 + OH = C_2H_2 + H_2O$	3.000e+13	0.0	0.0
R162	$C_2H_3 + CH_2 = C_2H_2 + CH_3$	3.000e+13	0.0	0.0
...	...	...	...	...
R165	$C_2H_3 + O = C_2H_2 + OH$	1.000e+13	0.0	0.0
R166	$C_2H_2 + OH = CH_3 + CO$	4.830e-04	4.0	-2000.0
R167	$C_2H_3 = C_2H_2 + H$	4.600e+40	-8.8	46200.0

6. Validation of numerical results

The CONVERGE CFD code was utilised to simultaneously predict the combustion process and emissions evolution using specific sub-models: SAGE for combustion, KH-RT (Kelvin-Helmholtz Rayleigh-Taylor hybrid model) for spray, RNG $k-\epsilon$ for turbulence, and the Hiroyasu and Zeldovich models for soot and NO_x emissions, respectively (Richards et al. [28]).

For this study, the re-normalization group (RNG) Reynolds averaged Navier-Stokes (RANS) approach was selected. The total kinetic energy is expressed as $k = 3/2 \cdot (U)^2$, where U represents the velocity and I is the initial turbulence intensity (5%). Turbulent dissipation is given by $\epsilon = C_\mu^{3/4} k^{3/2} l e^{-1}$, where ϵ is the turbulent dissipation, C_μ is a model constant, and le is the length scale. The CFD models utilised in the various physical phenomena employ model constants recommended by CONVERGE (Richards et al. [28]).

6.1. Combustion and emissions characteristics

In-cylinder pressure is the primary parameter influencing combustion processes. To validate our chosen reaction mechanism, we compared our cylinder pressure results (using a mechanism of 42 species and 167 reactions) with those of Liu and Zhang [34], who developed a reduced 77-species mechanism for diesel derived from Wang's mechanism (109 species and 543 reactions). Figure 5 illustrates a strong agreement between the numerical and experimental results, which leads to the assumption that results predicted by computational analysis are considerably similar to the actual result by experimental testing. The motoring

pressures for both cases rise in a similar manner. However, at TDC, the simulated pressure drops slightly, then increases markedly at 5° after top dead centre (ATDC) due to fuel injection. After 11° TDC, the pressure begins to decline. In contrast, Fig. 6 exhibits a low error margin of 7.2%, confirming the superior accuracy of the present model. The predicted heat release rate is similarly presented in Figure 5. Two distinct phases appear: an initial, rapid heat release from premixed combustion, which peaks at about 350 J/CA around 6° ATDC and can produce high temperatures, followed by a gradual decline into the slower, mixing-controlled diffusion combustion phase.

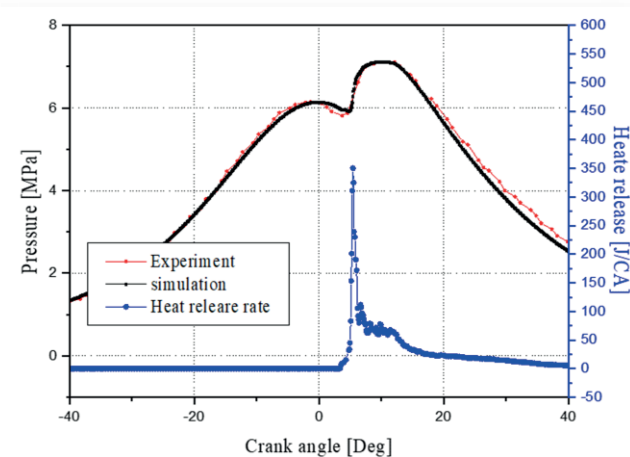


Fig. 5. The cylinder pressure (experimental vs. calculated) and heat release rate evolution.

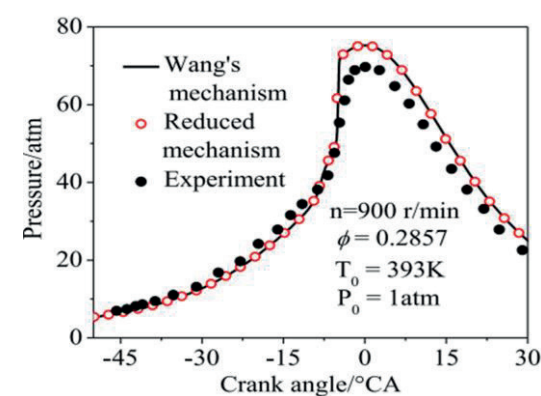


Fig. 6. The cylinder pressure reported by Liu and Zhang (2023) [34].

Figure 7 illustrates the changes in the average cylinder temperature and concentrations of CO, OH and H₂O₂. It is important to note that these species occur at temperatures around 930 K. Likewise, this temperature marks the point at which the concentration of H₂O₂ is sufficient to initiate auto-ignition, coinciding with the maximum heat release rate. Regarding the concentrations of OH and CO, it is clear that high levels of these species occur at the maximum temperature (around 1300K). Figure 8 shows the temporal evolution of n-heptane mass and soot emissions. Soot formation begins simultaneously with soot oxidation immediately after the start of injection (SOI) at 5° ATDC. Dur-

ing the initial stage, the soot formation rate increases, causing the net soot mass to rise until it reaches a maximum of 3.25×10^{-7} kg at 18° ATDC. Thereafter, the soot formation rate decreases, and the net soot mass falls to a final value of 9×10^{-8} kg at the end of the simulation.

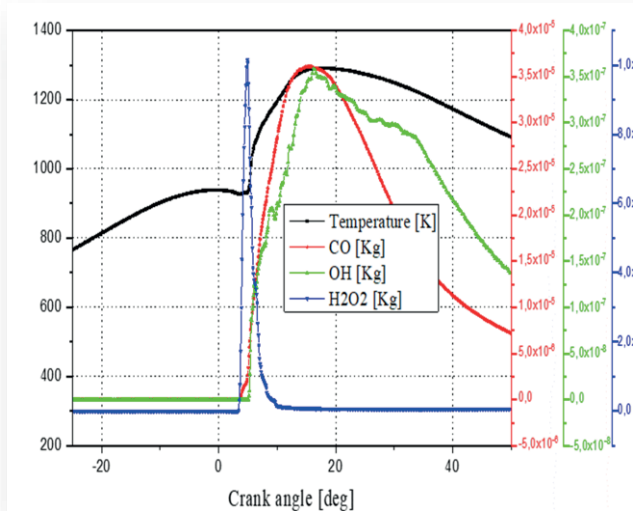
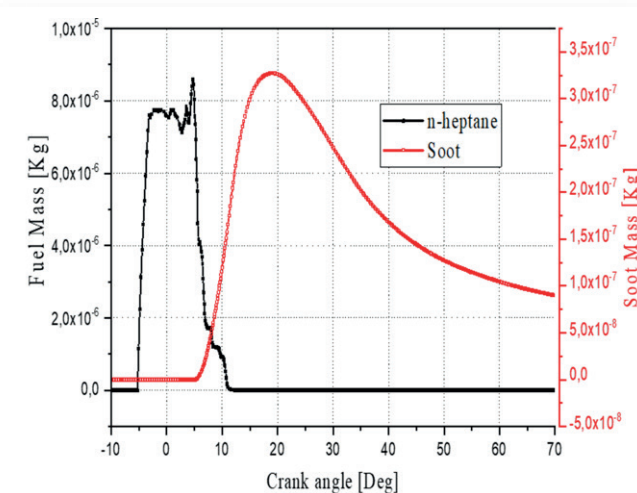
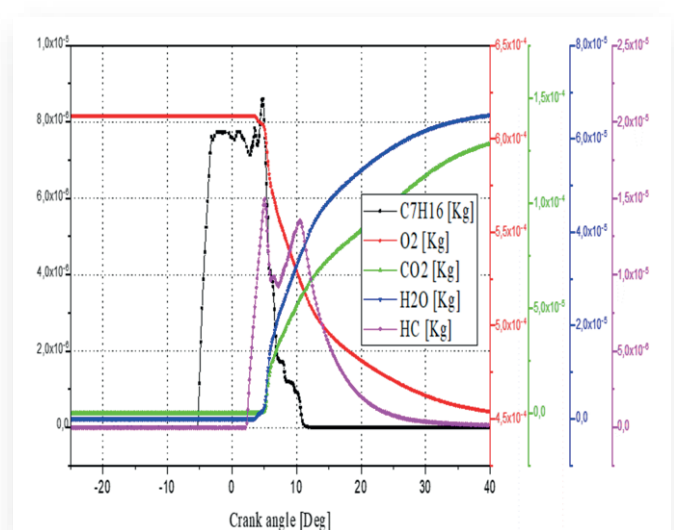
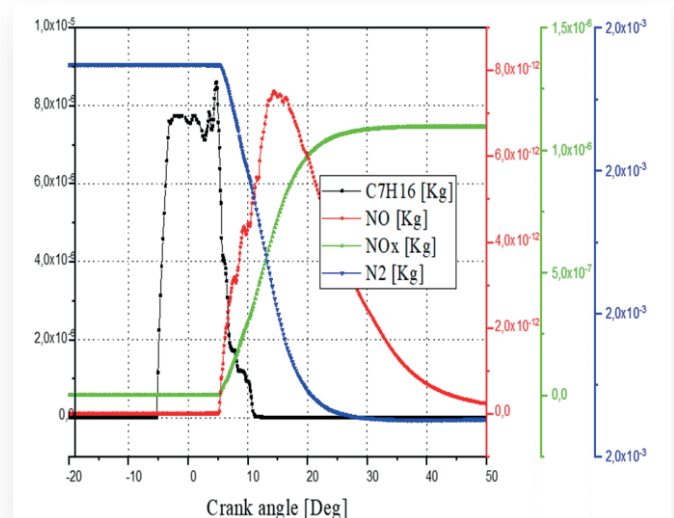
Fig. 7. Evolution of in-cylinder temperature and concentrations of CO, OH and H₂O₂.

Fig. 8. Evolution of in-cylinder fuel mass and soot emission.

In Fig. 9, the variations in fuel mass, O₂, CO₂, H₂O and HC concentrations are displayed. It is evident that the simulation accurately predicts the concentrations of these species in the combustion products. There is a significant increase in species CO₂, H₂O and HC concentrations during the combustion phase, a contrary to a decrease in O₂. It is clear that the CO₂ and H₂O concentrations rise rapidly during the premixed phase and continue to increase, albeit at a slower rate, during the diffusion phase.

Figure 10 depicts the changes in N₂, NO and NO_x concentrations. It is clearly observed that the concentrations of NO and NO_x increase during the rapid premixed phase of combustion and follow the temperature evolution, reach saturation for NO_x (around 1.1×10^{-4} kg), and decrease for NO during the diffusion phase.

Fig. 9. Evolution of in-cylinder fuel mass and concentrations of O₂, CO₂, H₂O and HC.Fig. 10. Evolution of in-cylinder fuel mass and concentrations of NO, N₂ and NO_x.

6.2. Scalar fields

Here, we present the combustion chamber's contours, explain the injection behaviour throughout the cycle, and offer a close-up of the inside to completely depict the results of the numerical simulation. Various crank angles were used to display the temperature distribution and spray progression.

Spray evolution

Figure 11 displays the contours of the fuel spray, indicating that the core of the cold jet contains an excessive amount of fuel. It clearly demonstrates the progression of the diesel spray.

Temperature evolution

The CONVERGE code has successfully predicted the temperature contours in diesel engines at different positions of the crank angle. This can be observed in Fig. 12, where a selection of different crank angles has been chosen to showcase the progression

of the combustion process. By utilising scalar fields, the evolution of the flame is visually represented through the contours of the temperature, and the rich region behaviour is presented by red and green contours. These contours demonstrate that the core of the cold jet contains an excessive amount of fuel, while the actual flame occurs in the space between the jet and the surrounding air (known as a diffusion flame). It is clearly shown that the rich region is identified mostly at the piston bowl wall and near the cylinder head. The temperature contours are observed by slicing through the axis of the spray. Notably, the red contours highlight the presence of a non-premixed flame. It is worth mentioning that the maximum temperature reached is approximately 2500 K. For this reason, it is expected that the formation of NO_x is high in these regions.

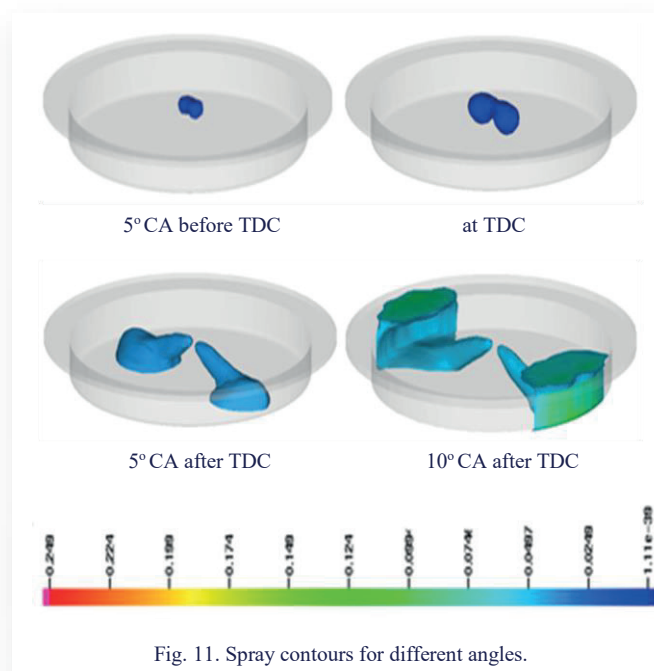


Fig. 11. Spray contours for different angles.

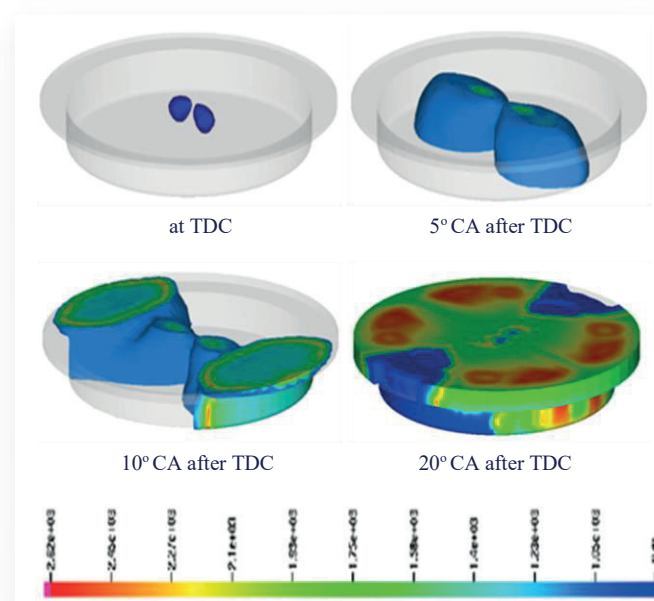


Fig. 12. Temperature distribution for different angles.

7. Conclusions

The combustion and emissions characteristics of a direct injection diesel engine fuelled with n-heptane were investigated using numerical simulations with CONVERGE CFD. The combustion process of n-heptane was analysed using a reduced reaction mechanism comprising 167 chemical reactions and 42 species, and the results were compared with the reduced mechanism developed by Liu and Zhang (2023), which contains 77 species. The main conclusions are summarised as follows:

- According to the results obtained, the combustion model used gives a good estimate of the average pressure in the cylinder – 4.7%, and the distribution of all the species of the reactive mixture can be perfectly described during the combustion phase.
- The heat release rate at its maximum is 350 J/CA at 6 BTDC.
- The concentrations of chemical species such as CO, NO, OH, N_2 , HC and soot, unlike those of H_2O , CO_2 and NO_x , seem adjusted by dosage and decrease rapidly during the slow diffusion phase of combustion.
- These species appear at temperatures of about 930 K, where the concentration of H_2O_2 is sufficient to initiate autoignition, and high levels of OH and CO are observed at a temperature of 1300 K.
- The evolution of the H_2O_2 concentrations seems to be related to that of the peaks of the in-cylinder pressure and average temperature in the cylinder during the premixed combustion phase.
- The simulation results indicate that soot is produced at the beginning of the combustion process by diffusion (around 18° BTDC, with a maximum value of 3.25×10^{-7} kg).
- The evolution of NO_x follows the evolution of temperature, reaching saturation at about 1.1×10^{-4} kg.
- During the combustion phase, we observe an increase in CO_2 , H_2O and HC species; on the other hand, O_2 decreases.
- The simulation results indicate that the diesel fuel model, which accounts for the n-heptane reaction mechanism and physical properties, possesses sufficient accuracy for implementation in more advanced computations.
- The model validated in this research will be utilised to develop fuel and biofuel injection strategies aimed at reducing emissions.
- The logic progression of this research will involve the development of a CFD model for engines that includes soot prediction capabilities, as well as engine experiments using n-heptane blended with alcohols (ethanol, methanol and MTBE) under mixing conditions. This will test the capabilities of the CFD model in the complex environment inside an engine and enhance our understanding of the pollution formation tendencies in heavy-duty engines.

Acknowledgements

The authors would like to thank Ecole Nationale Polytechnique d'Oran, LTE Laboratory, Department of Mechanical Engineering, ENP MA-Oran, Algeria. The authors would also like to

thank the CONVERGE CFD for their support throughout the research work.

References

- [1] Heywood, J.B. (1983). *Internal Combustion Engine Fundamentals*. New York, McGraw-Hill Inc.
- [2] Ramos, J.I. (1989). *Internal Combustion Engine Modeling*. Hemisphere Publishing Corporation.
- [3] Baumgarten, C. (2006). *Mixture Formation in Internal Combustion Engines*. Springer-Verlag Berlin Heidelberg, New York.
- [4] De Risi, A., Donato, T., & Laforgia, D. (2020). Theoretical investigation on the influence of physical parameters on soot and NO_x engine emissions. *Proceedings of the ASME 2001 Internal Combustion Engine Division Spring Technical Conference*. 2001ICES2001-120, 53–64, April 29–May 2, Philadelphia, Pennsylvania, USA. doi: 10.1115/ICES2001-120
- [5] Bousbaa, H., Liaqid, A., Sary, A., & Tazerout, M. (2013). Numerical Investigations on the use of Waste Animal Fats as fuel on DI Diesel engine. *Journal of Petroleum Technology and Alternative Fuels*, 4(7), 131–142. doi: 10.5897/JPTAF2013.0096
- [6] Zu, X-H., Yang, C-L., Wang, H-C., & Wang, Y-Y. (2019). Experimental study on diesel engine exhaust gas recirculation performance and optimum exhaust gas recirculation rate determination method. *Royal Society Open Science*, 6(6), 181907. doi: 10.1098/rsos.181907
- [7] De Serio, D., De Oliveira, A., & Sodr , J.R. (2017). Effects of EGR rate on performance and emissions of a diesel power generator fueled by B7. *Journal of Brazilian Society of Mechanical Sciences and Engineering*, 39, 1919–1927. doi: 10.1007/s40430-017-0777-x
- [8] Fayad, M.A. (2019). Effect of fuel injection strategy on combustion performance and NO_x /smoke trade-off under a range of operating conditions for a heavy-duty DI diesel engine, *SN Applied Sciences*, 1, 1088. doi: 10.1007/s42452-019-1083-2
- [9] Medjahed, L., & Bousbaa, H. (2021). Effect of injection parameters on combustion and emissions characteristics of a Turbo-charged DI diesel engine. A CFD approach using CONVERGE. *The First International Conference on Energy Transition and Security ICETS'21*, 22–24 May, Adrar, Algeria.
- [10] Timpanaro, A. (2019). *Reduction of NO_x Emissions in a Single Cylinder Diesel Engine Using SNCR with In-Cylinder Injection of Aqueous Urea Using SNCR with In-Cylinder Injection of Aqueous Urea*. UNF Graduate Theses, University of North Florida.
- [11] Demir, U., Kozan, A., &  zer, S. (2022). Experimental investigation of the effect of urea addition to fuel on engine performance and emissions in diesel engines. *Fuel*, 311, 122578. doi: 10.1016/j.fuel.2021.122578
- [12] Spoof-Tuomi, K., Niemi, S., Ovaska, T., Nilsson, O., & Heikkil , S. (2021). Selective Catalytic Reduction on Filter Performance Testing on Non-road Diesel Engine. *SAE Technical Paper*, 2021-01-5054. doi: 10.4271/2021-01-5054
- [13] Gurusamy, P.K., Gandhi, U., Mangalanathan, U., & Lr, K.M. (2017). Measurement of diesel exhaust fluid concentration in urea-SCR after-treatment system. *International Journal of Precision Engineering and Manufacturing*, 18(8), 1085–1092. doi: 10.1007/s12541-017-0127-z
- [14] Zhou, Q., Lucchini, T., D'Errico, G., & Hardy, G. (2019). Validation of Diesel Combustion Models with Turbulence Chemistry Interaction and Detailed Kinetics, *SAE Technical Paper*, 2019-24-0088. doi: 10.4271/2019-24-0088
- [15] Mohan, R.V., & Haworth, D.C. (2015). Turbulence-chemistry interactions in a heavy-duty compression-ignition engine. *Science Direct*, 35(3), 3053–3060. doi: 10.1016/j.proci.2014.06.098
- [16] Chen, Z., Zhang, T., Wang, X., Chen, H., Geng, L., & Zhang, T. (2021). A comparative study of combustion performance and emissions of dual-fuel engines fueled with natural gas/methanol and natural gas/gasoline. *Energy*, 237, 121586. doi: 10.1016/j.energy.2021.121586
- [17] Qin, Z., Yang, Z., & Jia, C. (2020). Experimental study on combustion characteristics of diesel–hydrogen dual-fuel engine. *Journal of Thermal Analysis and Calorimetry*, 142, 1483–1491. doi: 10.1007/s10973-019-09147-y
- [18] Han, J., Somers, L.M.T, Cracknell, R., Joedicke, A., Wardle, R., & Mohan, V.R.R. (2020). Experimental investigation of ethanol/diesel dual-fuel combustion in a heavy-duty diesel engine. *Fuel*, 275, 117867. doi: 10.1016/j.fuel.2020.117867
- [19] Bousbaa, H., Tarabet, L., Khatir, N., & Liaqid, A. (2020). Numerical Study on a Diesel Engine fueled by Eucalyptus biofuel using CONVERGE CFD Software. *Journal of Sciences and Technology*, 2(1), 106–119.
- [20] Bousbaa, H., Tarabet, L., Naima, K., Liaqid, A., & Tazerout, M. (2019). Eucalyptus Biofuel Study as Alternative for Diesel Engine, *International Journal of Renewable Energy Technology*, 10(3), 247–281. doi: 10.1504/IJRET.2019.10022907
- [21] Naima, K., Bousbaa, H., Tarabet, L., Menni, Y., & Lorenzini G. (2022). A Comparative Assessment of Combustion Behavior and Emissions Characteristics of DI Diesel Engine Fueled with Waste Plastic Oil and Eucalyptus Biofuel for Sustainable Development Applications. *International Journal of Low-Carbon Technologies*, 17, 1399–1405. doi: 10.1093/ijlct/ctac114
- [22] Elkelay, M. Bastawissi, H.A.E., El Shenawy, E.A., Shams, M.M., Panchal, H., Sadasivuni, K.K., & Choudhary, A.K. (2021). Influence of lean premixed ratio of PCCI-DI engine fueled by diesel/biodiesel blends on combustion, performance, and emission attributes; a comparison study. *Energy Conversion and Management: X*, 10, 100066. doi: 10.1016/j.ecmx.2020.100066
- [23] Singh, A.K., Pali, H.S., & Khan, M.M. (2024). Experimental Investigations of CI Engine Performance using Ternary Blends of n-Butanol/Biodiesel/Diesel and n-Octanol/Biodiesel/Diesel. *Archives of Thermodynamics*, 45(1), 109–118. doi: 10.24425/ather.2024.150443
- [24] Kumar, D., Kumar, N., & Chaudhary, R. (2024). Use of butanol, pentanol and diesel in a compression ignition engine: A review. *Archives of Thermodynamics*, 45(4), 27–35. doi: 10.24425/ather.2024.151994
- [25] Neeraj, K., Arora, B.B., & Maji, S. (2023). Environmental impact evaluation of diesel engine fuelled with CNG. *Archives of Environmental Protection*, 49(1), 79–84. doi: 10.24425/aep.2023.144740
- [26] Bousbaa, H., & Bouhadja, Y. (2019). Numerical study of the combustion process and emissions pollution in a spark ignition engine. *Communication Science and Technology*, 21, 41–58.
- [27] Bousbaa, H., Benayad, Z., & Naima, K. (2022). Numerical Analysis of Fuels Type Effect on Combustion and Emissions of Turbo-Charged Direct Injection Engine. *Energy and Thermofluids Engineering*, 2, 31–38. doi: 10.38208/ete.v2.531
- [28] Richards, K.J., Senecal, P.K., & Pomraning, E. (2022). *CONVERGE 2.1.0, Theory Manual A Three-Dimensional Computational Fluid Dynamics Program for Transient or Steady-State Flow with Complex Geometries*, Convergent Science, Inc.
- [29] Rente, T., Golovitchev, V.I., & Denbratt, I. (2001). Numerical Study of n-Heptane Spray Auto-Ignition at Different Levels of Pre-Ignition Turbulence. *The Fifth International Symposium on Diagnostics and Modeling of Combustion in Internal Combustion Engines (COMODIA 2001)*, pp. 86–91, July 1–4, Nagoya, Japan.

- [30] Wan Mansor, W.N., & Olsen, D.B. (2016), Computational Modeling of Diesel and Dual Fuel Combustion Using Converge CFD Software, *ARPJN Journal of Engineering and Applied Sciences*, 11(23), 13697–13707.
- [31] Ra, Y., & Reitz, R.D. (2008). A reduced chemical kinetic model for IC engine combustion simulations with primary reference fuels. *Combustion and Flame*, 155(4), 713–738. doi: 10.1016/j.combustflame.2008.05.002
- [32] Wang, H., & Frenklach, M. (1997), A detailed kinetic modeling study of aromatics formation in laminar premixed acetylene and ethylene flames. *Combustion and Flame*, 110(1–2), 173–221. doi: 10.1016/S0010-2180(97)00068-0
- [33] Liu, W., Sivaramakrishnan, R., Davis, M.J., Som, S., Longman, D.E., & Lu, T.F. (2013). Development of a reduced biodiesel surrogate model for compression ignition engine modeling. *Proceedings of the Combustion Institute*, 34(1), 401–409. doi: 10.1016/j.proci.2012.05.090
- [34] Liu, C., & Zhang, D. (2023). A reduced chemical kinetic mechanism of diesel fuel for HCCI engines. *Journal of Measurements in Engineering*, 11(1). doi: 10.21595/jme.2023.23132
- [35] Wang, H., Yao, M., Yue, Z., Jia, M., & Reitz, R.D. (2015). A reduced toluene reference fuel chemical kinetic mechanism for combustion and polycyclic-aromatic hydrocarbon predictions. *Combustion and Flame*, 162(6), 2390–2404. doi: 10.1016/j.combustflame.2015.02.005
- [36] Yin, P., Zhang, C., Zhou, A., Li, S., & Li, Y. (2020). A Reduced Chemical Kinetic Mechanism for Toluene, Reference Fuels Based On DRGEP and QSSA Methods. *Technical Gazette (Tehnički vjesnik)*, 27(5), 1571–1578. doi: 10.17559/TV-20190913062824
- [37] Hu, F., Li, P., Guo, J., Liu, Z., Wang, L., Mi, J., Dally, B., & Zheng, C. (2018). Global reaction mechanisms for MILD oxy-combustion of methane. *Energy*, 147, 839–857. doi: 10.1016/j.energy.2018.01.089
- [38] Baigmohammadi, M., Patel, V., Martinez, S., Panigrahy, S. Ramalingam, A.; Burke, U., Somers, K.P., Heufer, K.A.; Pekalski, A., & Curran, H.J (2020), A Comprehensive Experimental and Simulation Study of Ignition Delay Time Characteristics of Single Fuel C₁-C₂ Hydrocarbons over a Wide Range of Temperatures, Pressures, Equivalence Ratios, and Dilutions. *Energy & Fuels*, 34(3), 3755–3771. doi: 10.1021/acs.energyfuels.9b04139



Co-published by
Institute of Fluid-Flow Machinery
Polish Academy of Sciences
Committee on Thermodynamics and Combustion
Polish Academy of Sciences

Copyright©2026 by the Authors under licence CC BY-NC-ND 4.0

<http://www.imp.gda.pl/archives-of-thermodynamics/>



Delineating the influence of solvent fraction and temperature-dependent heat source/sink on magneto-convective flow: A response surface methodology and an artificial neural network approach

S.K. Abhilasha^{a*}, Manoranjan Parhi^b, Harjot Singh Gill^c, Nagaraj Patil^d, M. Umshaiah^e

^aDepartment of Studies in Mathematics, Davangere University, Davangere 577007, Karnataka, India

^bDepartment of Computer Science and Engineering, Siksha 'O' Anusandhan (Deemed to be University), Bhubaneswar, Odisha 751030, India

^cDepartment of Mechanical Engineering, Chandigarh University, Mohali 140307, Punjab, India

^dDepartment of Mechanical Engineering, School of Engineering and Technology, JAIN (Deemed to be University), Bangalore 562112, Karnataka, India

^eDepartment of Mathematics, PES Institute of Technology & Management, Shivamogga 577204, Karnataka, India

*Corresponding author email: abhilashaskmaths@gmail.com

Received: 26.08.2025; revised: 02.03.2026; accepted: 04.03.2026

Abstract

The present study investigates the influence of a non-uniform heat sink/source on the unsteady flow of Boger liquid via a slowly rotating stretching disk subjected to suction and convective boundary conditions. This study is significant because it helps optimise cooling and heating processes in chemical, pharmaceutical and energy systems. The research also highlights methods for managing heat and mass transport in porous and industrial materials. Overall, it helps to improve performance, stability and energy efficiency in engineering and applied science. Dimensionless ordinary differential equations are obtained by transforming the governing partial differential equations using similarity variables. The resultant non-dimensional ordinary differential equations are solved numerically utilising the finite difference method. Furthermore, the fluid flow, mass and heat transfer and are evaluated using an artificial neural network approach. Additionally, the response surface methodology is used to assess the heat transmission rate statistically. The influence of different parameters on the concentration, velocity and thermal profiles is exemplified graphically. Increasing the suction parameter, relaxation time ratio and magnetic parameter reduces the velocity profile. The thermal profile enhances as the space and temperature-dependent heat sink/source parameters increase.

Keywords: Boger fluid; Non-uniform heat source/sink; Activation energy; Convective boundary conditions; Slow rotating disk

Vol. 47(2026), No. 2, 69–85; doi: 10.24425/ather.2026.158675

Cite this manuscript as: Abhilasha, S.K, Parhi, M., Gill, H.S., Patil, N., & Umshaiah, M. (2026). Delineating the influence of solvent fraction and temperature-dependent heat source/sink on magneto-convective flow: A response surface methodology and an artificial neural network approach. *Archives of Thermodynamics*. 47(2), 69–85.

1. Introduction

Heat transfer (HT) is a crucial and extensively used discipline within engineering. Additionally, a lot of research has been done on increasing the rate of HT, which is important for many applications, including storing radioactive nuclear waste, cooling electronic components, catalysts, heat exchangers, and more. Enhancing the rate of HT can be achieved by using several liquid types, particularly nanoliquid and hybrid nanoliquid, which

are created by combining one or more types of highly thermally conductive nanoparticles. The function of liquids in the process of HT has been discussed in several significant scientific publications. Recently, Chamkha and Pop [1] examined the HT characteristics of a laminar boundary layer flow of a viscous liquid past a vertical plate. Kattimani et al. [2] used a Levenberg-Marquardt artificial neural network (ANN) to investigate HT behaviour at the stagnation-point motion of a fluid past an off-centred disk. Kumar et al. [3] elucidated the HT characteristics of hybrid

Nomenclature

A – constant, $A = -w_0/2\sqrt{\frac{\nu_f\Omega_0}{1-\alpha t}}$
 A^*, B^* – space and temperature-dependent heat source/sink parameters
 B_0 – magnetic field, $\sqrt{\frac{\text{kg}\Omega}{s}}\text{m}^{-1}$
 Bi – Biot number, $Bi = \frac{h_f}{k_f}\sqrt{\frac{\nu_f(1-\alpha t)}{\Omega_0}}$
 c – stretching rate, s^{-1}
 C – concentration
 C_p – specific heat, $\text{J kg}^{-1}\text{K}^{-1}$
 D_B – diffusion coefficient, m^2s^{-1}
 E – dimensionless activation energy, $E = E_a/KT_\infty$
 E_a – activation energy, $\text{kg m}^2\text{s}^{-2}$
 (F, G, H) – velocity profiles
 h_f – heat transfer coefficient, $\text{W m}^{-2}\text{K}^{-1}$
 k_f – thermal conductivity, $\text{W m}^{-1}\text{K}^{-1}$
 k_r^2 – reaction rate, s^{-1}
 K – Boltzmann constant, $\text{kg m}^2\text{s}^{-2}\text{K}^{-1}$
 M – magnetic parameter, $M = \sigma_f B_0^2/\rho_f\Omega_0$
 p – pressure, $\text{kg m}^{-1}\text{s}^{-2}$
 P – dimensionless pressure
 Pr – Prandtl number, $Pr = (\mu C_p)_f/k_f$
 q''' – non-uniform heat source/sink coefficient, W m^{-3}
 r, z – coordinates, m
 Re – Reynolds number,
 Sc – Schmidt number, $Sc = \nu_f/D_B$,
 t – time, s
 T – temperature, K
 T_∞ – ambient temperature, K
 T_w – surface temperature, K
 (u, v, w) – velocity components, m s^{-1}
 w_0 – suction velocity, m s^{-1}

nanoliquid motion through a stretchy cylinder. Hussain et al. [4] studied HT variations of the Casson ternary nanoliquid motion using ANN. Eegunjobi et al. [5] investigated HT features of an electrically conducting liquid flow via an expanding surface.

Boger liquids are the elastic liquids with a constant viscosity irrespective of shear rate, which allows them to differentiate between elastic and viscous effects in viscoelastic flows. Their flexibility enables progress in materials science, agriculture, gas and oil. These liquids are also very important for consumer products like cosmetics and food, where controlling viscosity is essential. These liquids are created by mixing a small quantity of polymer with a high-viscosity Newtonian liquid. This makes a substance that has properties of both a liquid and a solid. In order to overcome the shear thinning caused by the polymer additions, highly viscous solvents are used to form Boger liquids. When exploring elastic characteristics without the restriction of shear thinning, these fluids are essential. Norouzi et al. [6] elucidated the pressure drop of a Boger liquid motion at the inertia regime. Banakar et al. [7] examined the motion of Boger liquid over a rotating rough disk with a solid-liquid interfacial layer impact. Veeranna et al. [8] deliberated the Boger micropolar liquid flow across a semi-circular body. The experimental compar-

Greek symbols

α – angular velocity, s^{-1}
 α^* – dimensionless quantity, $\alpha^* = \alpha/\Omega_0$
 β_1 – solvent fraction parameter
 β_2 – ratio of relaxation time
 δ – temperature-difference parameter, $\delta = (T_w - T_\infty)/T_\infty$
 γ_1 – stretching ratio parameter, $\gamma_1 = c/\Omega_0$
 $\theta(\eta)$ – thermal profile
 μ_f – dynamic viscosity, $\text{kg m}^{-1}\text{s}^{-1}$
 ν_f – kinematic viscosity, m^2s^{-1}
 ρ_f – density, kg m^{-3}
 σ_f – electrical conductivity, $\Omega^{-1}\text{m}^{-1}$
 σ^* – chemical reaction rate parameter, $\sigma^* = k_r^2(1 - \alpha t)/\Omega_0$
 $\phi(\eta)$ – concentration profile
 Ω_0 – positive constant, s^{-1}

Subscripts and Superscripts

∞ – ambient
 f – fluid
 w – wall

Abbreviations and Acronyms

AE – activation energy
ANN – artificial neural network
C-BC – convective boundary condition
DE – differential equation
FDM – finite difference method
HT – heat transfer
MF – magnetic field
MHD – magnetohydrodynamics
N-HSS – non-uniform heat source/sink
ODE – ordinary differential equation
PDE – partial differential equation
RSM – response surface methodology

ison of the interfacial wave behaviour of the Boger liquid stream in microchannels was explained by Hue et al. [9].

Under the influence of an external magnetic field (MF), investigators have conducted both experimental and theoretical studies on the thermomagnetic circulation of magnetic liquids that contain various geometries. Understanding the connection between an MF and the characteristics of a liquid flow has been a major focus in the study of electrically conducting fluid motion for many years. Further, magnetohydrodynamics (MHD) deliberates the behaviour of electrically conductive liquids in the presence of MFs. When subjected to an external MF, it exhibits distinctive behaviours such as magneto-viscous, magneto-optical and magneto-thermal impacts. Srihari et al. [10] scrutinised the Casson liquid flow in a vertical microchannel with the MF effect. Krishna et al. [11] deliberated the motion of a second-grade liquid with the effect of MF past a permeable surface. Preethi et al. [12] inspected the outcome of MF and thermal radiation on the flow of nanoliquid via an extended sheet. Takhar et al. [13] addressed the unsteady HT in the presence of MF on a semi-infinite plate. Takhar et al. [14] studied the HT behaviour on the unsteady MHD liquid motion influenced by MF over a stretchy surface. Jagadha et al. [15] exemplified the impact of MF on the MHD Casson liquid flow along an extending surface.

Non-uniform heat source/sinks (N-HSSs) are unevenly distributed, with changing amounts added or removed from a liquid across a specific area. Since these situations can change the flow properties of the liquid system and HT, they are important to many industrial applications. Temperature control is essential in the manufacture of chemicals and polymers since configurations can significantly impact the convective HT rates. Non-uniform heat sources placed into fluid flow can cause deviations in the flow's pressure, thermal and velocity profiles. Kumar et al. [16] elucidated the N-HSS influence on the time-varying nanoliquid motion past a permeable slider. Banakar et al. [17] inspected the outcome of N-HSS impacts on the bioconvective liquid flow between two parallel disks. Thumma et al. [18] investigated the influence of N-HSS on the Maxwell nanoliquid flow over a stretchy sheet. Banakar et al. [19] examined the impact of N-HSS on the tetra hybrid nanliquid motion past a rotating rough disk with the Levenberg-Marquardt ANN. Algehyne et al. [20] discovered the HT features of the MHD flow of micropolar liquid with the influence of N-HSS across a stretchable surface.

In 1889, Svante Arrhenius developed the idea of activation energy (AE), referring to the smallest energy required for potential reactants in a chemical system to undergo a reaction. A reaction takes place when atoms move rapidly due to this AE. This concept is vital in chemistry and is prevalent in many chemically reactive systems, such as oil reservoirs, food processing, geothermal systems and chemical engineering. Mass transfer is inevitable in the mixture whenever a concentration difference exists between two species during a chemical reaction. As a result, the concentration will naturally move from the species with a greater concentration to one with a lesser concentration. Salahuddin et al. [21] exemplified the AE consequence on the Casson liquid flow past an exponential surface. Hayat et al. [22] deliberated the significance of AE on third-grade hydromagnetic nanofluid flow past an elongating sheet. Asjad et al. [23] probed the Williamson liquid motion under the effect of AE and bioconvection. Samantha and Mondal [24] inspected the consequence of AE on the nanoliquid flow via a stretching disk. Srilatha et al. [25] delved into the effect of AE on the chemically reactive liquid motion past a stretching disk.

Defining convective boundary conditions (C-BCs) is important for understanding how heat moves between a fluid and a solid surface. This is especially relevant in studies of cooling systems and heat exchangers, where temperature differences are significant near the solid boundaries. These types of boundary conditions are commonly applied in nuclear power plants, gas turbines and thermal energy storage. C-BCs are typically used to analyse the temperature in order to produce a linear heat exchange for some algebraic values. Akbar et al. [26] deliberated the MHD liquid flow with the radiation influence via an elongating sheet using C-BC. By employing C-BC, Malik et al. [27] exemplified the outcome of C-BC on the Sisko liquid motion over an elongating surface. Shaw et al. [28] assessed the effects of C-BC and thermal convection on the Casson nanoliquid over a horizontal plate. The C-BC outcome on the Maxwell nanoliquid flow via a stretchy surface was addressed by Wang et al. [29]. The incompressible flow of nanoliquid in the occur-

rence of C-BC and variable thermal conductivity impacts across a surface was examined by Abbas and Khan [30].

Examining decelerating spinning disks is necessary for understanding a range of industrial applications where liquid mixing is essential, such as lubrication systems and stirred tanks. These disks are also used in brakes, flywheels and gas turbines, all of which are often employed in sectors such as analytical chemistry. A slow-rotating disk generally refers to a circular object that spins at a minimal rate of angular velocity. The interaction between the disk's rotation and the viscous fluid can create intricate flow patterns, influencing efficiency and performance. Rafiq et al. [31] probed the HT modelling with variable fluid characteristics on the unsteady revolving fluid motion across a rotating decelerated porous disk. They measured the degradation rate of the decelerated disk's angular velocity. Using a decelerating rotating disk, Ahmed et al. [32] assessed the thermal energy transfer in thin-film Buongiorno nanofluid flow with thermophoresis and Brownian motion. Ejaz and Mustafa [33] scrutinised the unsteady flow and compared different viscosity models across a slow-revolving disk with various physical properties. Lin et al. [34] observed the flow of nanoliquid via a spinning disk with surface temperature. Ali et al. [35] scrutinised the flow of the Sutterby fluid past a revolving disk with convection.

To the best of the author's knowledge, there has been no prior research on the unsteady Boger liquid motion past an expandable slowly rotating disk under the impact of N-HSS, AE and C-BC. This gap in previous studies forms the core research gap. In view of this, the current work scrutinised the consequences of AE and suction on the time-dependent motion of the Boger liquid over a slowly rotating stretched disk in the presence of N-HSS, chemical reactions, C-BCs and MF. The technical uses of slow-rotating disks involve flow in centrifugal filtering processes, food and chemical processing sectors, rotating machinery, and constructing multi-pore distributors. The combination of slow-rotating stretching disks with the AE impact provides a successful structure for examining the behaviour of reacting fluids under simultaneous mechanical and thermal influences. Further, analysing MF effects has substantial implications for chemistry, physics and engineering. The interface of electrically conductive fluids with MFs affects various industrial apparatus, such as bearings, pumps, MHD generators and boundary layer control systems. Similarity variables are used to convert the governing partial differential equations (PDEs) into dimensionless ordinary differential equations (ODEs). Additionally, the finite difference method (FDM) technique is utilised to numerically solve the derived ODEs, and the ANN approach is used to evaluate the fluid's mass, heat and flow transport profiles. Additionally, the response surface methodology (RSM) is utilised to address the HT rate of the liquid flow statically. Visual representations of the impact of various non-dimensional parameters on the concentration, temperature and velocity profiles are provided.

2. Mathematical formulation

Consider the unsteady flow of Boger fluid via a stretchy disk that rotates slowly, subjected to an MF $B(t)$. Let the velocity

components be (u, v, w) , and t denotes time, while p is the pressure. Figure 1 illustrates the flow geometry of the current model.

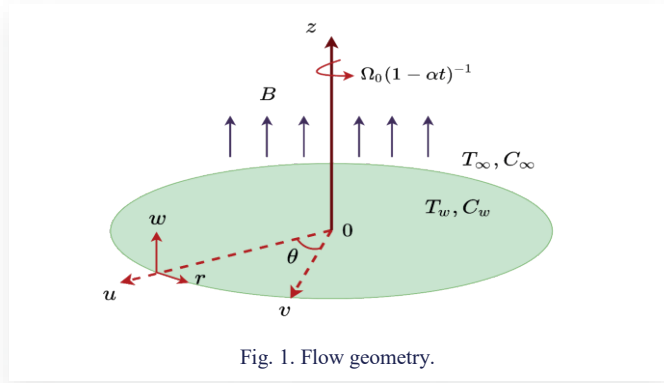


Fig. 1. Flow geometry.

The current modelling is predicated on the following assumptions:

- The main reason for the stream's instability is anticipated to be the disk's angular velocity.
- The disk's slowness is attributable to its angular velocity, which exhibits an opposite relationship with time.
- The effect induced by MF, $B(t) = \frac{B_0}{\sqrt{1-\alpha t}}$, is perpendicular to the surface of the disk and influences the fluid motion throughout the field.
- To analyse heat transmission behaviour, both C-BCs and N-HSS are considered.
- The consequences of AE and chemical reactions on the transfer of mass are taken into account.

Considering the assumptions above, the governing equations of the current model are expressed below (see [36–41]). Equations (1)–(6) represent the continuity, conservation of momentum, energy and concentration for the present flow problem.

$$\frac{\partial}{\partial r}(ur) + \frac{\partial}{\partial z}(wr) = 0, \quad (1)$$

$$u \frac{\partial u}{\partial r} + \frac{\partial p}{\partial r} \frac{1}{\rho_f} + \frac{\partial u}{\partial t} - \frac{v^2}{r} + w \frac{\partial u}{\partial z} = \nu_f \left(\frac{1+\beta_1}{1+\beta_2} \right) \left[\frac{\partial^2 u}{\partial r^2} + \frac{\partial^2 u}{\partial z^2} + \frac{\partial}{\partial r} \left(\frac{u}{r} \right) \right] - \frac{\sigma_f u B(t)^2}{\rho_f}, \quad (2)$$

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial r} + \frac{wv}{r} + w \frac{\partial v}{\partial z} = \nu_f \left(\frac{1+\beta_1}{1+\beta_2} \right) \left[\frac{\partial^2 v}{\partial r^2} + \frac{\partial}{\partial r} \left(\frac{v}{r} \right) + \frac{\partial^2 v}{\partial z^2} \right] - \frac{\sigma_f v B(t)^2}{\rho_f}, \quad (3)$$

$$u \frac{\partial w}{\partial r} + \frac{\partial w}{\partial t} + w \frac{\partial w}{\partial z} = - \frac{\partial p}{\partial z} \frac{1}{\rho_f} + \nu_f \left(\frac{1+\beta_1}{1+\beta_2} \right) \left[\frac{\partial^2 w}{\partial z^2} + \frac{\partial^2 w}{\partial r^2} + r^{-1} \frac{\partial w}{\partial r} \right], \quad (4)$$

$$\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial r} + w \frac{\partial T}{\partial z} = \frac{k_f}{(\rho C_p)_f} \left[r^{-1} \frac{\partial T}{\partial r} + \frac{\partial^2 T}{\partial z^2} + \frac{\partial^2 T}{\partial r^2} \right] + \frac{q'''}{(\rho C_p)_f}, \quad (5)$$

$$w \frac{\partial C}{\partial z} + u \frac{\partial C}{\partial r} + \frac{\partial C}{\partial t} = D_B \left[\frac{\partial^2 C}{\partial r^2} + \frac{\partial^2 C}{\partial z^2} + \frac{1}{r} \frac{\partial C}{\partial r} \right] - k_r^2 \left(\frac{T}{T_\infty} \right)^n \exp \left(\frac{-E_a}{KT} \right) (C - C_\infty). \quad (6)$$

Equation (1) is the continuity equation, where u, v and w denote the velocity components, while r and z represent the coordinates. Equations (2)–(4) explain the momentum transport in the radial, tangential and axial directions. The time derivative terms refer to the presence of unsteady flow behaviour, and the convective terms are the consequences of transporting momentum because of fluid motion. Equation (2) represents the radial momentum equation. The term $\partial u / \partial t$ denotes the unsteady radial velocity, while $u \partial u / \partial r$ and $w \partial u / \partial z$ represent convective acceleration. The term $1/\rho_f \partial p / \partial r$ corresponds to the pressure force, and $-v^2/r$ represents the centrifugal effect due to the tangential velocity v . The terms containing ν_f describe viscous diffusion. Equation (3) is the tangential momentum equation. The term $\partial v / \partial t$ represents the unsteady tangential velocity, while $u \partial v / \partial r$ and $w \partial v / \partial z$ denote convection. The term uv/r represents the curvature effect. The viscous terms describe momentum diffusion, and the magnetic term represents resistive force. Equation (4) represents the axial momentum equation. The term $\partial w / \partial t$ denotes unsteady axial velocity, while $u \partial w / \partial r$ and $w \partial w / \partial z$ represent convection. The term $1/\rho_f \partial p / \partial z$ represents pressure force. The viscous terms describe momentum diffusion. Equation (5) represents the energy equation. The term $\partial T / \partial t$ denotes unsteady temperature, while $u \partial T / \partial r$ and $w \partial T / \partial z$ represent heat convection. The diffu-

sion terms describe heat conduction, and $q'''/(\rho C_p)_f$ represents a non-uniform heat source/sink term. Equation (6) represents the concentration equation. The term $\partial C / \partial t$ denotes unsteady concentration, while $u \partial C / \partial r$ and $w \partial C / \partial z$ represent species convection. The diffusion terms describe molecular diffusion, and the last term represents the chemical reaction with activation energy.

The related boundary conditions are depicted as follows (see [42]):

- at $z = 0$

$$\begin{aligned} v &= \frac{r\Omega_0}{(1-\alpha t)}, & u &= \frac{rc}{(1-\alpha t)}, & w &= w_0, \\ -k_f \frac{\partial T}{\partial z} &= (T_w - T)h_f, & C &= C_w, \end{aligned} \quad (7a)$$

- at $z \rightarrow \infty$

$$\begin{aligned} u &\rightarrow 0, & p &\rightarrow 0, & v &\rightarrow 0, \\ T &\rightarrow T_\infty, & C &\rightarrow C_\infty. \end{aligned} \quad (7b)$$

Here, w_0 is the suction velocity and $r\Omega_0/(1-\alpha t)$ angular velocity with angular velocity variation α , and h_f is the HT coefficient.

The N-HSS q''' is given as follows:

$$q''' = \frac{k_f \Omega_0}{\nu_f (1-\alpha t)} [(T - T_\infty)B^* + A^*(T_w - T_\infty)F'(\eta)]. \quad (8)$$

The suitable similarity variables are introduced as follows:

$$\left. \begin{aligned} u &= F(\eta) \frac{r\Omega_0}{(1-\alpha t)}, & v &= G(\eta) \frac{r\Omega_0}{(1-\alpha t)}, \\ w &= -2 \sqrt{\frac{\nu_f \Omega_0}{(1-\alpha t)}} H(\eta), & p &= P(\eta) \left(\frac{-\rho_f \nu_f \Omega_0}{(1-\alpha t)} \right), \\ T &= T_\infty + (T_w - T_\infty)\theta(\eta), & \eta &= z \sqrt{\frac{\Omega_0}{\nu_f (1-\alpha t)}}, \\ C &= C_\infty + (C_w - C_\infty)\phi(\eta). \end{aligned} \right\} \quad (9)$$

Utilising Eqs. (9), the governing Eqs. (1)–(6) are reduced and obtained as follows:

$$H' - F = 0, \quad (10)$$

$$\left(\frac{1+\beta_1}{1+\beta_2} \right) F'' + 2HF' - MF - F^2 + G^2 - \alpha^* \left[\frac{\eta}{2} F' + F \right] = 0, \quad (11)$$

$$\left(\frac{1+\beta_1}{1+\beta_2} \right) G'' - \alpha^* \left[G + \frac{\eta}{2} G' \right] + 2HG' - 2FG - MG = 0, \quad (12)$$

$$2 \left(\frac{1+\beta_1}{1+\beta_2} \right) F' + 4FH - P' - \alpha^* [H + \eta H'] = 0, \quad (13)$$

$$\frac{1}{\text{Pr}} \theta'' + 2H\theta' - \frac{1}{2} \alpha^* \eta \theta' + \frac{1}{\text{Pr}} [A^* F' + B^* \theta] = 0, \quad (14)$$

$$\frac{1}{\text{Sc}} \phi'' - \frac{1}{2} \alpha^* \eta \phi' + 2H\phi' - \sigma^* (1 + \delta \theta)^n \exp \left[-\frac{E}{(1+\delta \theta)} \right] \phi = 0. \quad (15)$$

The following are the modified boundary conditions:

- at $\eta = 0$

$$\begin{aligned} F(\eta) &= \gamma_1, & H(\eta) &= A, & G(\eta) &= 1, \\ \theta'(\eta) &= -\text{Bi}(1 - \theta(\eta)), & \phi(\eta) &= 1, \end{aligned} \quad (16)$$

- as $\eta \rightarrow \infty$

$$\begin{aligned} F(\eta) &= 0, & G(\eta) &= 0, & P(\eta) &= 0, \\ \theta(\eta) &= 0, & \phi(\eta) &= 0, \end{aligned}$$

where:

$$\begin{aligned} M &= \frac{\sigma_f B_0^2}{\rho_f \Omega_0}, & \text{Bi} &= \frac{h_f}{k_f} \sqrt{\frac{\nu_f (1-\alpha t)}{\Omega_0}}, & \text{Pr} &= \frac{(\mu C_p)_f}{k_f}, \\ \alpha^* &= \frac{\alpha}{\Omega_0}, & \text{Sc} &= \frac{\nu_f}{D_B}, & E &= \frac{E_a}{KT_\infty}, & \gamma_1 &= \frac{c}{\Omega_0}, \\ \delta &= \frac{T_w - T_\infty}{T_\infty}, & \sigma^* &= \frac{k_r^2 (1-\alpha t)}{\Omega_0}, & A &= \frac{-w_0}{2 \sqrt{\frac{\nu_f \Omega_0}{1-\alpha t}}}. \end{aligned}$$

where M is the magnetic parameter, Bi is the Biot number, Pr is the Prandtl number, α^* is the dimensionless quantity, Sc is the Schmidt number, E is the dimensionless activation energy, γ_1 is the stretching ratio parameter, δ is the temperature difference parameter, σ^* is the chemical reaction rate parameter, and A is a constant. Also, $F(\eta)$, $G(\eta)$, and $H(\eta)$, denote the dimensionless velocity profiles, while $\theta(\eta)$ and $\phi(\eta)$ represent the dimensionless temperature and concentration profiles, respectively.

The engineering coefficients are given as (see [43]):

$$\begin{aligned} \frac{\text{Cf}}{\text{Re}^{1/2}} &= \sqrt{(G'(0))^2 + (F'(0))^2}, \\ \frac{\text{Nu}}{\text{Re}^{1/2}} &= -\theta'(0), \\ \frac{\text{Sh}}{\text{Re}^{1/2}} &= -\chi'(0). \end{aligned} \quad (17)$$

3. Methodology

FDM delineates the stability differential equations (DEs) pertinent to the discussed problem in connection with the finite element method that simulates the physical system. To accomplish this approximation, the derivatives contained in the DEs are substituted with approximations of the derivatives expressed in the DEs. The representation generated is a set of simultaneous algebraic equations. Because it directly relates to DE, FDM is a type of derivative solver. FDM can be utilised to numerically solve the system of Eqs. (10–15) and Eq. (16) for different values of the constraints. To accomplish it, consider the derivatives of ϕ, H, θ, F and G with respect to η , and imagine the solutions of ϕ, H, θ, G, P and F are $\chi_i, h_i, \theta_i, \tau_i, p_i$ and ς_i , respectively, and at the points (η_i) , with $i = 0, \dots, n$

$$\begin{aligned} G' &\simeq \frac{\tau_{i+1} - \tau_{i-1}}{2\Delta}, & F' &\simeq \frac{\varsigma_{i+1} - \varsigma_{i-1}}{2\Delta}, & \theta' &\simeq \frac{\theta_{i+1} - \theta_{i-1}}{2\Delta}, \\ H' &\simeq \frac{h_{i+1} - h_{i-1}}{2\Delta}, & \phi' &\simeq \frac{\chi_{i+1} - \chi_{i-1}}{2\Delta}, \\ G'' &\simeq \frac{\tau_{i-1} - 2\tau_i + \tau_{i+1}}{\Delta^2}, & F'' &\simeq \frac{\varsigma_{i-1} - 2\varsigma_i + \varsigma_{i+1}}{\Delta^2}, \\ \theta'' &\simeq \frac{\theta_{i-1} - 2\theta_i + \theta_{i+1}}{\Delta^2}, & H'' &\simeq \frac{h_{i-1} - 2h_i + h_{i+1}}{\Delta^2}, \\ \phi'' &\simeq \frac{\chi_{i-1} - 2\chi_i + \chi_{i+1}}{\Delta^2}, & P' &\simeq \frac{p_{i+1} - p_{i-1}}{2\Delta}. \end{aligned}$$

Using the terms mentioned here in place of the simplified equations and boundary conditions, one can obtain:

$$\frac{-h_{i-1} + h_{i+1}}{2\Delta} - \varsigma_i = 0, \quad (18)$$

$$\frac{\varsigma_{i-1} - 2\varsigma_i + \varsigma_{i+1}}{\Delta^2} \left(\frac{1+\beta_1}{1+\beta_2} \right) - (\varsigma_i)^2 + (\tau_i)^2 - M\varsigma_i + 2h_i \frac{\varsigma_{i+1} - \varsigma_{i-1}}{2\Delta} - \left[\varsigma_i + \left(\frac{i\Delta}{2} \right) \frac{\varsigma_{i+1} - \varsigma_{i-1}}{2\Delta} \right] \alpha^* = 0, \quad (19)$$

$$\frac{\tau_{i+1} + \tau_{i-1} - 2\tau_i}{\Delta^2} \left(\frac{1+\beta_1}{1+\beta_2} \right) + 2h_i \frac{\tau_{i+1} - \tau_{i-1}}{2\Delta} - M\tau_i - \alpha^* \left[\tau_i + \left(\frac{i\Delta}{2} \right) \frac{\tau_{i+1} - \tau_{i-1}}{2\Delta} \right] - 2\varsigma_i \tau_i = 0, \quad (20)$$

$$2 \left(\frac{1+\beta_1}{1+\beta_2} \right) \frac{\varsigma_{i+1} - \varsigma_{i-1}}{2\Delta} + 4\varsigma_i h_i - \frac{p_{i+1} - p_{i-1}}{2\Delta} - \alpha^* \left[h_i + i\Delta \frac{h_{i+1} - h_{i-1}}{2\Delta} \right] = 0, \quad (21)$$

$$\left(\frac{1}{\text{Pr}} \right) \frac{\theta_{i+1} + \theta_{i-1} - 2\theta_i}{\Delta^2} + 2h_i \left(\frac{-\theta_{i-1} + \theta_{i+1}}{2\Delta} \right) - \frac{1}{2} \alpha^* i\Delta \frac{\theta_{i+1} - \theta_{i-1}}{2\Delta} + \frac{1}{\text{Pr}} \left[A^* \frac{\varsigma_{i+1} - \varsigma_{i-1}}{2\Delta} + B^* \theta_i \right] = 0, \quad (22)$$

$$\left(\frac{1}{Sc}\right)\frac{\chi_{i-1}-2\chi_i+\chi_{i+1}}{\Delta^2} + 2h_i\frac{\chi_{i+1}-\chi_{i-1}}{2\Delta} - \frac{1}{2}\alpha^*i\Delta\left(\frac{-\chi_{i-1}+\chi_{i+1}}{2\Delta}\right) - \sigma^*(1+\delta\theta_i)^n \exp\left[-\frac{E}{(1+\delta\theta_i)}\right]\chi_i = 0. \quad (23)$$

The following are the simplified boundary conditions:

$$\zeta_0 = \gamma_1, \quad \tau_0 = 1, \quad h_0 = A, \quad \chi_0 = 1, \quad \frac{\theta_1 - \theta_{-1}}{2\Delta} = -Bi(1 - \theta_0),$$

$$\theta_n = 0, \quad \zeta_n = 0, \quad \tau_n = 0, \quad p_n = 0, \quad \chi_n = 0. \quad (24)$$

The FDM reduces the potential energy function by simulating the solution to the controlling DEs. These equations give the requirements for carrying out the operation. To minimise the potential energy functional, the FDM gives a reasonably close answer. Table 1 presents the assessment of $-\theta'(0)$, $-G'(0)$ and $F'(0)$ results with an established study for a reduced case. Figure 2 depicts the flow chart of the numerical procedure.

Table 1. The assessment of $-\theta'(0)$, $-G'(0)$ and $F'(0)$ results with an established study, for some reduced cases.

	Turkylmazoglu [44]	Present results
$-\theta'(0)$	0.93387794	0.93388563
$-G'(0)$	0.61592201	0.61591889
$F'(0)$	0.51023262	0.51024235

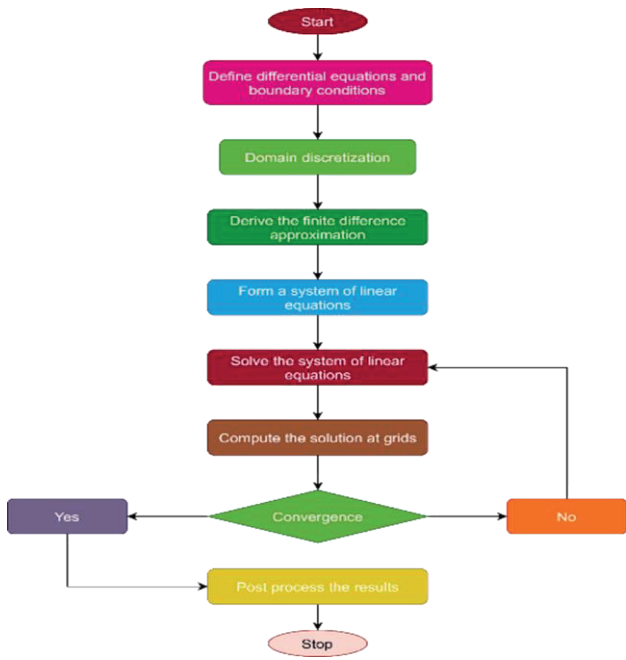


Fig. 2. Flowchart of the numerical procedure.

4. Artificial neural networks

ANNs are an essential instrument for modelling and predicting complex systems. ANNs are particularly beneficial in this scenario since they can identify the essential patterns and relationships within the data, allowing more precise predictions of fluid dynamics and thermal behaviour. By being trained on a set of data, the network could uncover the complex connections between the output variables and the input parameters. ANNs are capable of handling large datasets and identifying complex patterns that traditional mathematical modelling approaches would overlook. ANNs are also proficient in processing massive amounts of data. The ANN structure for the profiles $-\theta'(0)$,

$-G'(0)$ and $F'(0)$ are seen in the figures. The correctness of the supplied data is ensured through comparison analysis.

ANNs can be expressed mathematically as:

$$A = \sum_{j=0}^N Q_i \xi_i + C_0. \quad (25)$$

Here, N is the number of inputs, ξ_i is the input value, and C_0 and Q_i represent the bias and weight.

The regression (R^2) and mean square error (MSE) equations are quantified as follows:

$$R^2 = 1 - \left(\frac{\sum_{i=1}^j (-\delta_{ANN(i)} + \delta_{NUM(i)})^2}{\sum_{i=1}^j (\delta_{NUM(i)})^2} \right), \quad (26)$$

$$MSE = \frac{1}{j} \sum_{i=1}^j (-\delta_{ANN(i)} + \delta_{NUM(i)}). \quad (27)$$

5. Response surface methodology

RSM is a statistical method that predicts and maximises the connections between different input components and outcome features. The effect of the Biot number, A^* and B^* on the Nusselt number is examined using RSM for thermal transfer analysis. To evaluate the correctness of the research, analysis of variance (ANOVA) employs statistical measures, including p-values, F-values, error rates and lack-of-fit tests. If the F-value is greater than 1 and the p-value is less than 0.05, the accuracy is confirmed at the 95% confidence level. This study confirms that the regression model accurately predicts the Nusselt number by accounting for all relevant inputs. The model's correctness is confirmed by residual and distribution analyses, as well as the correlation between adjusted and predicted values. This method is effective at evaluating massive non-linear interactions, and offers significant techniques for maximising or limiting responses. The response is modelled using the necessary parameters using the traditional multivariate design, which is explained below:

$$Nu = Y_1 + Y_2 A + Y_3 B + Y_4 C + Y_5 A \cdot A + Y_6 B \cdot B + Y_7 C \cdot C + Y_8 A \cdot B + Y_9 A \cdot C + Y_{10} B \cdot C \quad (28)$$

Here, Y_i ($i = 1, 2, \dots, 10$) signifies the factors of the quadratic model resulting from RSM, which is used to assess the parameters and their interactive effects on HT.

The provided regression model contains interaction, linear and quadratic effects. The central composite design experimental framework includes centralised runs as well as axial and factorial runs. To match the square-shaped region of interest, the axis extends about one unit from the centre. Table 2 provides specifics about the effective factor range for imitating Nu. Furthermore, the regression model that corresponds to the important factorial parameters is expressed in actual units as:

Table 2. Factors coded values and their range.

Parameter	Range	Level		
		-1	0	1
A^*	-0.1 to 0.2	-0.1	0.05	0.2
B^*	-0.1 to 0.2	-0.1	0.05	0.2
Bi	0.35 to 10	0.35	5.175	10

$$Nu = 0.1527 - 0.117 \cdot B^* + 0.352 \cdot A^* + 0.82 \cdot A^* \cdot A^* + 0.1478 \cdot Bi - 0.26 \cdot B^* \cdot B^* - 0.00955 \cdot Bi \cdot Bi + 0.3030 \cdot A^* \cdot Bi + 0.768 \cdot A^* \cdot B^* - 0.0430 \cdot B^* \cdot Bi. \quad (29)$$

5.1. Model validation

The reliability and robustness of the model are evaluated using statistical techniques such as lack-of-fit tests and ANOVA. To assess the model's ability to capture changes in the response variable, specific indicators are needed. To guarantee exceptional precision in the results, a 5% significance level or 95% confidence level is frequently utilised. Together with a model that links the response to the factors and the Nusselt number, the ANOVA results display the p-values for each of the three components. At the 95% level of significance, a parameter is considered statistically significant if its p-value is less than 0.05. Except for $A^* \cdot A^*$, $B^* \cdot B^*$, $A^* \cdot B^*$ and $A^* \cdot Bi$, all terms have p-values below 0.05, so the insignificant term has been eliminated. As a result, the updated model is shown as follows:

$$Nu = 0.1527 + 0.352 \cdot A^* + 0.1478 \cdot Bi - 0.00955 \cdot Bi \cdot Bi + 0.3030 \cdot A^* \cdot Bi. \quad (30)$$

The modified values of R^2 (97.17%) and estimated values of R^2 (98.51%) were found, along with a synopsis of the scrutiny. ANOVA was used to assess the model's dependability, and Table 3 provides specifics of the findings.

Table 3. Factor-coded values and their range.

Source	DF	Adj SS	Adj MS	F-value	P-value
B^*	1	0.02406	0.024058	6.41	0.030
Square	3	0.23158	0.077194	20.57	0.000
A^*	1	0.93709	0.937089	249.71	0.000
Model	9	2.48152	0.275724	73.47	0.000
Bi	1	0.89384	0.893843	238.19	0.000
Linear	3	1.85499	0.618330	164.77	0.000
$B^* \cdot B^*$	1	0.00009	0.000091	0.02	0.879
Bi·Bi	1	0.13601	0.136006	36.24	0.000
$A^* \cdot A^*$	1	0.00094	0.000936	0.25	0.628
2-way interaction	3	0.39495	0.131649	35.08	0.000
$A^* \cdot B^*$	1	0.00239	0.002391	0.64	0.443
$B^* \cdot Bi$	1	0.00775	0.007754	2.07	0.181
$A^* \cdot Bi$	1	0.38480	0.384800	102.54	0.000
Error	10	0.03753	0.003753	-	-
Pure error	5	0.00000	0.000000	-	-
Lack-of-fit	5	0.03753	0.007505	-	-
Total	19	2.51904	-	-	-

6. Results and discussion

This research analyses the motion of Boger liquid around a slowly revolving stretching disk with the effect of an MF. The consequence of N-HSS, C-BCs and AE on the motion of the liquid is also considered. The governing PDEs are converted to ODEs with the aid of similarity variables. The resulting ODEs are solved by using the FDM scheme. Additionally, ANN is used to examine the mass, heat and liquid flow transport properties, and the obtained ANN graphs for $F(\eta)$, $\theta(\eta)$ and $\phi(\eta)$ are

depicted with the appropriate results. Additionally, the Nusselt number was evaluated for different parameters using RSM. The graphical representation reveals many dimensionless factors influencing the different profiles.

The consequences of β_1 on $F(\eta)$ and $G(\eta)$ are exposed in Fig. 3 and Fig. 4. An increased solvent fraction signifies an improvement in $G(\eta)$ and $F(\eta)$ for growing values corresponding to a greater solvent ratio relative to the fluid particles. The augmented solvent volume leads to a drop in the liquid's flow resistance and viscosity. Reduced fluid flow friction within the system results in lower viscosity and higher velocities. The reduced viscosity enhances the liquid's response to flow-driving forces, facilitating the deformation and movement of fluid layers with greater ease. Consequently, as the fluid undergoes a reduction in resistance and internal friction, both velocity components augment, which is due to the reduction in friction and resistance. The effect is most evident in Boger fluids, which exhibit a constant viscosity irrespective of the shear rate.

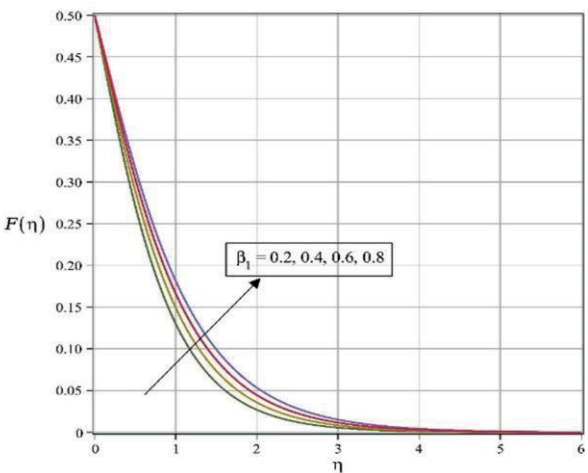


Fig. 3. The effect of β_1 on $F(\eta)$.

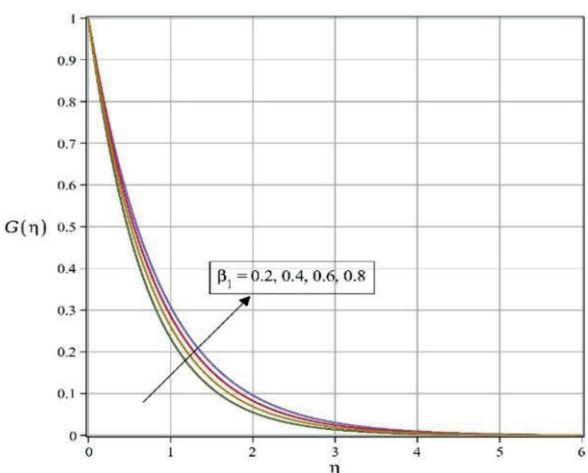


Fig. 4. The effect of β_1 on $G(\eta)$.

Figures 5 and 6 illustrate the outcome of M on $F(\eta)$ and $G(\eta)$. It is observed that both velocity profiles drop significantly as the MF strength increases. The M is related to the Lorentz force.

Physically, increasing the M value lowers the velocity by increasing the Lorentz force, causing the liquid's motion to slow. The resistive force in speed and amplitude causes the fluid's velocity to decrease. MF effectively limits the flow because it opposes the flow's driving forces, which lowers $F(\eta)$ and $G(\eta)$.

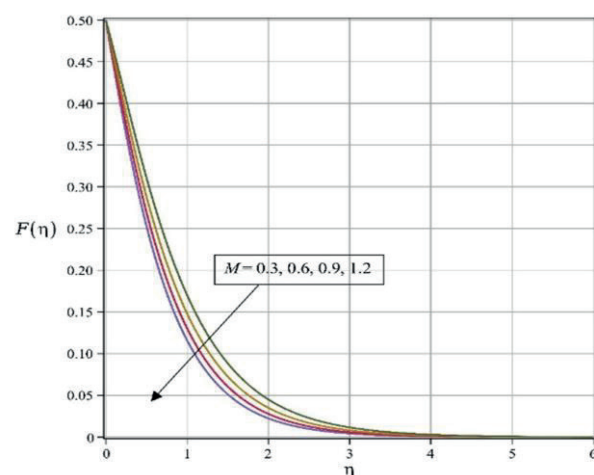


Fig. 5. The effect of M on $F(\eta)$.

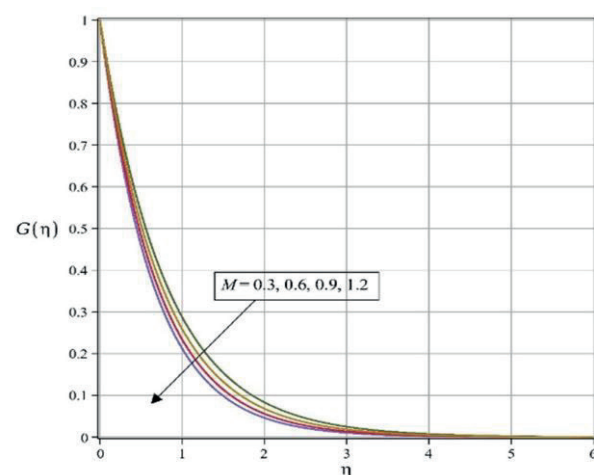


Fig. 6. The effect of M on $G(\eta)$.

The impact of β_2 on $F(\eta)$ and $G(\eta)$ is revealed in Fig. 7 and Fig. 8, respectively. $G(\eta)$ and $F(\eta)$ drop for increased values of β_2 . An increase in β_2 indicates an extended liquid relaxation duration compared to the flow's time scale. The liquid's slow reaction to distortion, attributable to its extended relaxation period, clearly demonstrates its viscoelastic characteristics. Consequently, the fluid exhibits greater resistance to distortion and motion. However, its ability to adapt to variations in dynamic flow is limited. In both velocity profiles, the liquid is less susceptible to shear forces due to the resistance. By prolonging the relaxation time, the fluid's velocity components are effectively reduced, leading to decreased flow rates. As the liquid's viscoelastic resistance rises, both velocity profiles decrease.

Figures 9 and 10 demonstrate the impact of A on $F(\eta)$ and $G(\eta)$. The intensification in the values of A causes reductions of both $G(\eta)$ and $F(\eta)$. When they get close to the surface, the velocities drastically decrease. An increase in A implies stronger

fluid withdrawal from the flow region. This retards the momentum of the fluid and makes the velocity boundary layer thinner, which results in a decreased value of $F(\eta)$ and $G(\eta)$.

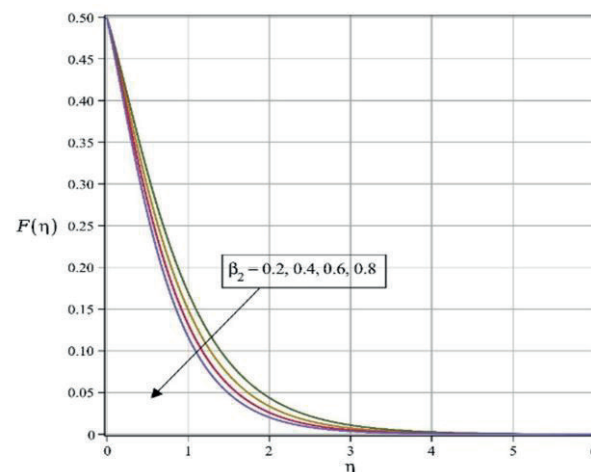


Fig. 7. The effect of β_2 on $F(\eta)$.

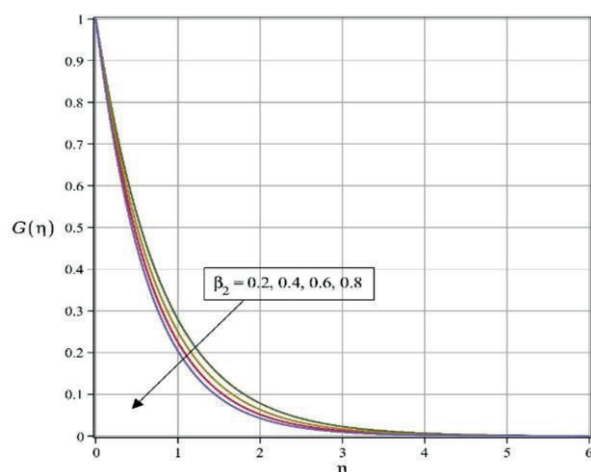


Fig. 8. The effect of β_2 on $G(\eta)$.

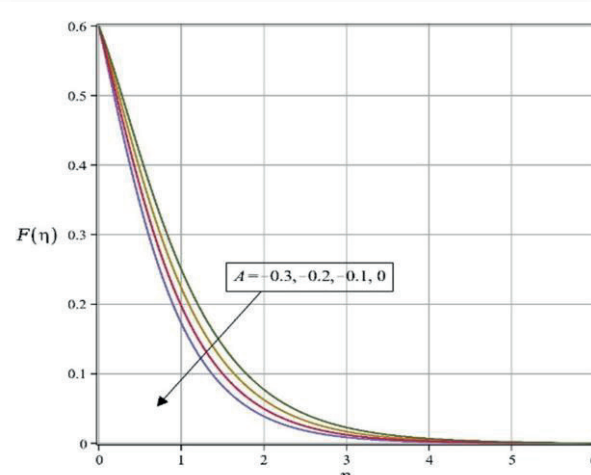


Fig. 9. The effect of A on $F(\eta)$.

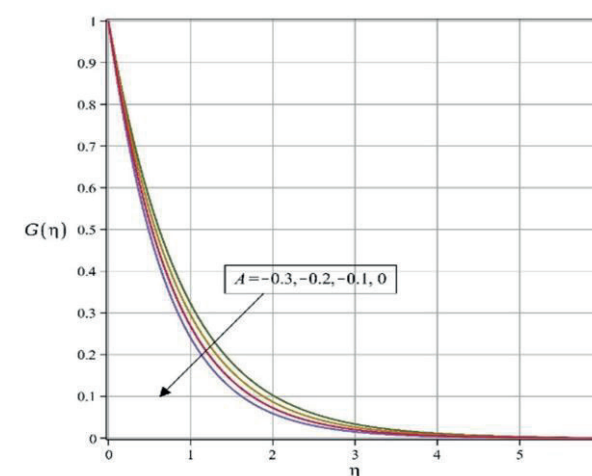


Fig. 10. The effect of A on $G(\eta)$.

Figures 11 and 12 demonstrate the significance of A^* and B^* for $\theta(\eta)$, respectively. $\theta(\eta)$ is growing due to the elevated values of N-HSS parameters (A^* , B^*).

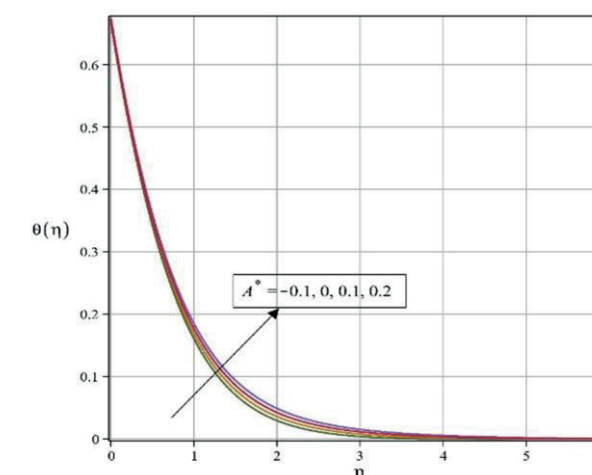


Fig. 11 The effect of A^* on $\theta(\eta)$.

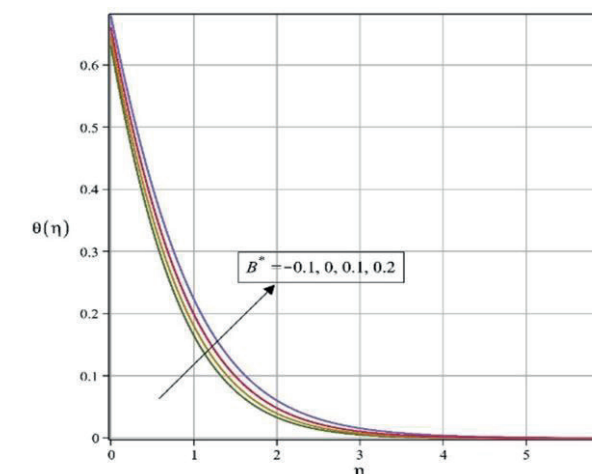


Fig. 12. The effect of B^* on $\theta(\eta)$.

The positive values of A^* and B^* can be read physically as heat sources that function as heat producers, resulting in the re-

lease of heat energy into the stream and an increase in $\theta(\eta)$. The negative values of B^* and A^* indicate the heat sink, which functions as a heat absorber, with energy being absorbed for these negative values. Figure 13 demonstrates the effect of Bi on $\theta(\eta)$. The thermal profile advances with the increase in Bi values. The relationship between temperature and temperature boundary layer thickness correlates strongly with Bi. The heat transport coefficient is crucial for Bi, its value escalates with a rise in the heat transmission coefficient. Consequently, elevated heat transport coefficient values lead to the enhanced temperature and boundary layer thickness.

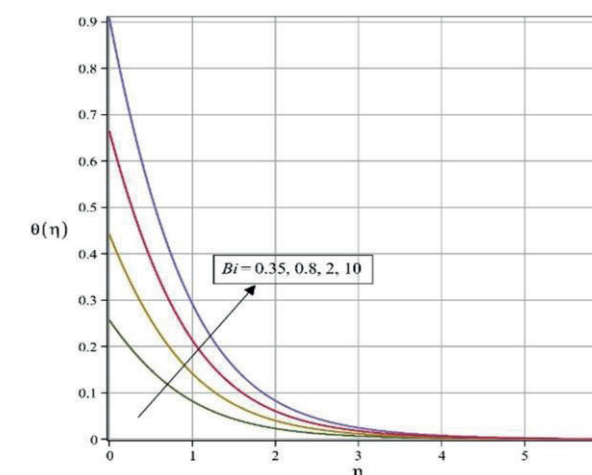


Fig. 13. The effect of Bi on $\theta(\eta)$.

Figure 14 demonstrates the significance of σ^* for $\phi(\eta)$. The increasing σ^* values decrease the concentration profile. As the chemical reaction rises, it quickly increases the concentration of reactants near the reacting surface. As those reactant particles get depleted, the difference in concentration that pushes them to move from one place to another becomes smaller. Since this concentration difference drives mass transfer, its reduction slows down the overall transport of material. Overall, stronger reactions exhaust the reactants too fast, leaving less to be carried forward, which ultimately weakens mass transfer. The effect of E on the $\phi(\eta)$ is exhibited in Fig. 15.

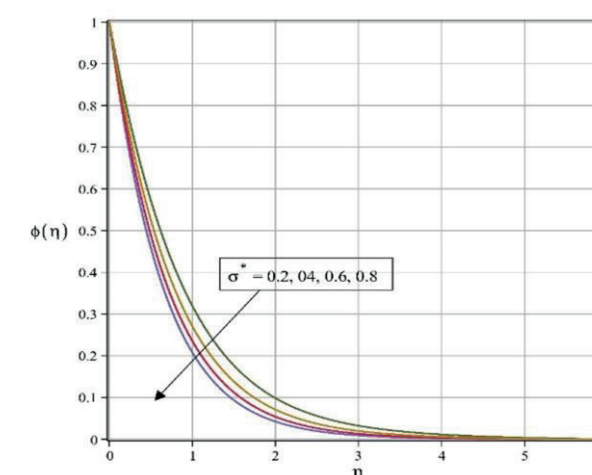


Fig. 14. The effect of σ^* on $\phi(\eta)$.

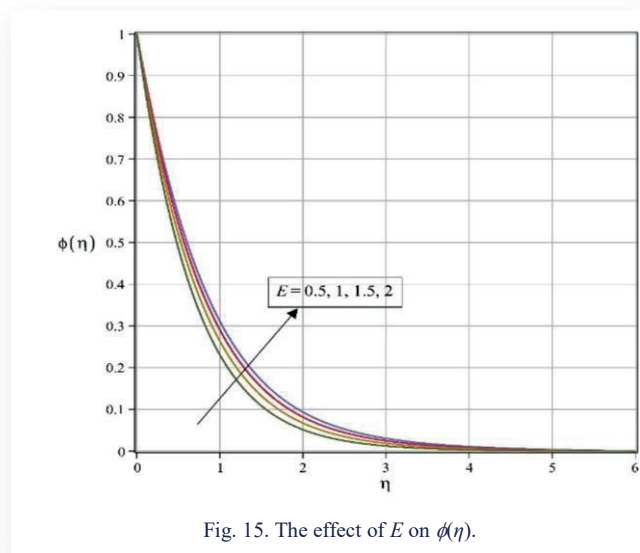


Fig. 15. The effect of E on $\phi(\eta)$.

The mass transfer is advanced by the increase in E . A low AE has been shown to decrease the reaction rate, which in turn causes chemical reaction mechanisms to decay. The improved Arrhenius process declines with increasing E . Thus, this accelerates the generative chemical process, raising the nanoparticle concentration. Figure 16 represents the effect of Sc on $\phi(\eta)$. The increased value of Sc declines the $\phi(\eta)$ profile. A higher Sc means that the fluid's momentum spreads much faster than its mass does. When the mass diffusivity becomes low, the species cannot disperse easily through the liquid. As a result, the concentration gradient remains steep near the surface, and the overall concentration level decreases throughout the boundary layer. Therefore, an increase in Sc causes a noticeable drop in $\phi(\eta)$.

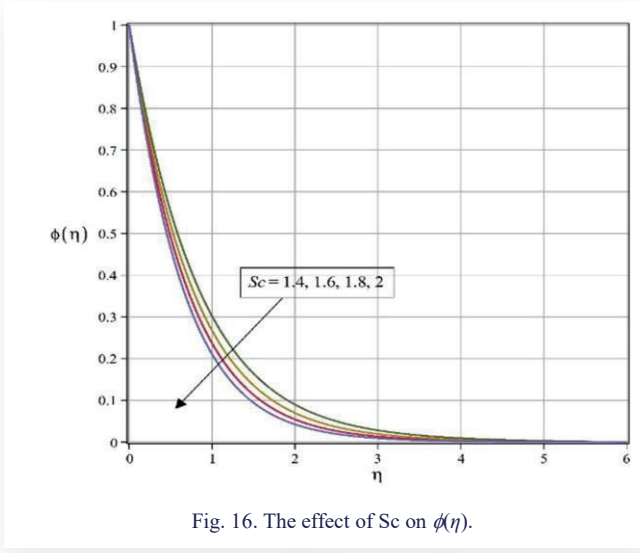


Fig. 16. The effect of Sc on $\phi(\eta)$.

Figures 17, 18, and 19 show the impact of β_2 , γ_1 and M on the isotherms. A larger β_2 and γ_1 value often causes more affected bending of the isotherms, which modifies the stream pattern of the disk. This effect may enhance or impair the liquid's mixing and transport properties, depending on the application. Additionally, complex flow behaviours that are essential for optimising processes like magnetic resonance imaging and magnetic drug targeting may arise from the interaction of the MF

parameter. The increased effective diffusivity lessens changes in the concentration gradient. The increased depletion of reactants as Sc increases, owing to reduced mass diffusivity, results in a fall in $\phi(\eta)$. Tables 4 and 5 depict values of C_f and Nu for various values of different parameters.

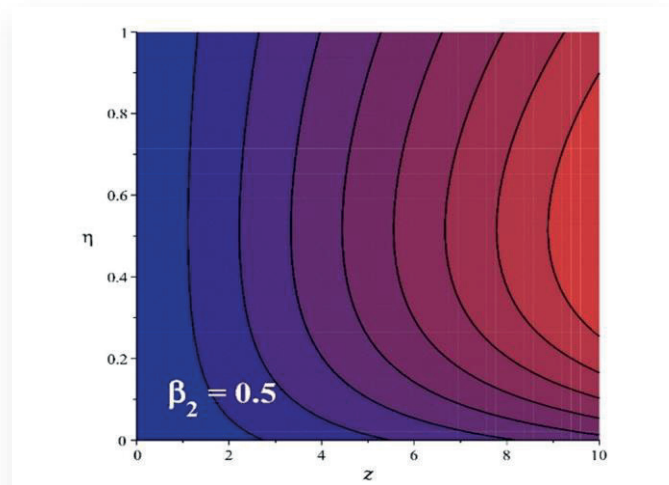


Fig. 17. The effect of β_2 on isotherms.

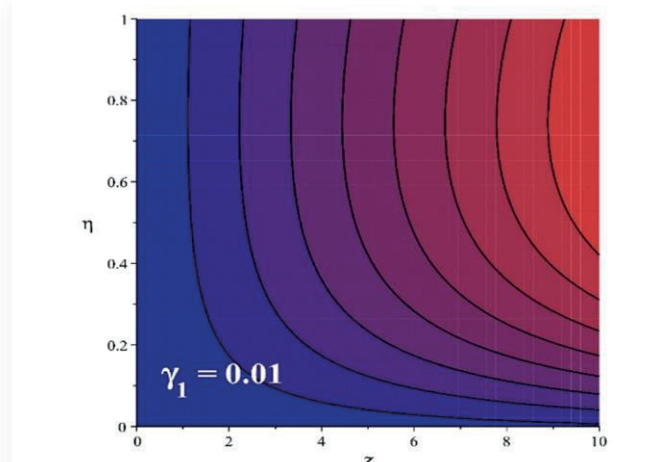


Fig. 18. The effect of γ_1 on isotherms.

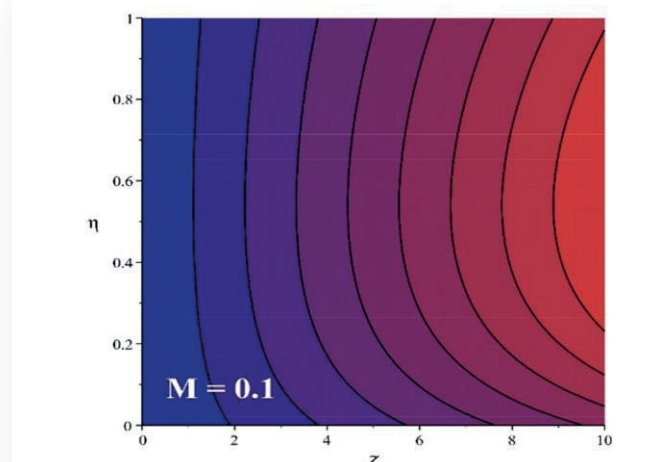


Fig. 19. The effect of M on isotherms.

Table 4. Values of C_f for several values of different parameters.

β_1	β_2	M	α^*	A	γ_1	C_f
0.1						1.042287
0.2	0.1	0.1	0.1	0.2	0.1	0.990247
0.3						0.944963
	0.2					1.097463
0.1	0.4	0.1	0.1	0.2	0.1	1.20364
	0.6					1.305283
		0.1				1.092923
0.2	0.3	0.2	0.1	0.2	0.1	1.133078
		0.3				1.17445
			0.1			1.184868
0.2	0.2	0.2	0.2	0.3	0.1	1.214202
			0.3			1.243693
				0.1		1.053558
0.1	0.1	0.1	0.1	0.2	0.2	1.151923
				0.3		1.262031
					0.1	1.331356
0.1	0.1	0.1	0.2	0.4	0.2	1.412899
					0.3	1.508619

Table 5. Values of Nu for several values of different parameters.

M	α^*	A^*	B^*	Bi	Nu
0.1					0.744257
0.2	0.1	0.1	0.1	1	0.743091
0.3					0.742016
	0.1				0.745093
0.1	0.2	0.2	0.2	1	0.738799
					0.731253
		0.1			0.740948
0.1	0.2	0.2	0.3	1	0.739542
					0.738136
			0.1		0.739161
0.3	0.1	0.3	0.2	1	0.741271
			0.3		0.743311
					0.096555
0.1	0.1	0.1	0.1	0.1	0.186934
					0.271711

ANNs are an extremely flexible modelling tool that can be used to solve a wide range of classification or prediction issues. They can comprehend the accurate relationship between output and data. Within an ANN architecture, output and input neurons are connected by various layers of neurons, including hidden layers. Initial training and evaluation of ANNs are based on empirical data to control the optimal topology and weights. ANNs use the back-propagation approach to adjust their weights during training to decrease the differences between predicted and actual outcomes. The input, output and hidden layers of neurons comprise ANNs. The data is divided as 80% goes into the training database, 10% into the validation database, and 10% into the testing database.

Figure 20 depicts the ANN design, consisting of an input, hidden, and output layer.

Figures 21(a), 22(a) and 23(a) show MSE for $F(\eta)$, $\theta(\eta)$ and $\phi(\eta)$ profiles in the rotating disk configuration, accounting for the testing, validation and training phases. The best efficiency is obtained with MSE values of 1.0118E-14, 1.4356E-10 and

3.0595E-11 for 133, 833 and 908 epochs, respectively. Since MSE calculates the difference between the ideal outcomes and the intended solutions, the lowest MSE values indicate precise and effective outcomes.

Figures 21(b), 22(b) and 23(b) show the training state for $F(\eta)$, $\theta(\eta)$ and $\phi(\eta)$. As seen in Figs. 21(b), 22(b) and 23(b), the gradient parameter is 9.8743E-08 for 133 epochs, 9.9822E-08 for 833 epochs and 9.9726E-08 for 908 epochs of $F(\eta)$, $\theta(\eta)$ and $\phi(\eta)$ profiles, correspondingly. The Mu parameter of ANN is 1E-14 for 133 epochs, 1E-08 for 833 epochs and 1E-09 for 908 epochs of $F(\eta)$, $\theta(\eta)$ and $\phi(\eta)$ profiles. The validation checks are zero for all profiles. The results of the procedure demonstrated that the built ANN actively converged and was accurate.

The histograms of error for the $F(\eta)$, $\theta(\eta)$ and $\phi(\eta)$ profiles are shown in Figs. 21(c), 22(c), and 23(c). The zero error line for $F(\eta)$, $\theta(\eta)$ and $\phi(\eta)$ is indicated by a coloured line. It is seen that the zero-error line is approximately at $-5.3E-09$ for $F(\eta)$, $3.52E-07$ for $\theta(\eta)$ and $-4.8E-07$ for $\phi(\eta)$.

Figures 21(d), 22(d) and 23(d) present extensive information on regression cases for all three of the suggested model's situations. By demonstrating the linearity between the intended and actual outputs, the graphs reveal the relationship between all datasets, including training, validation and testing data. A regression value of $R = 1$ signifies an optimal fit, and the regression diagrams illustrate the degree of correspondence between the data and the curve or line for the $F(\eta)$, $\theta(\eta)$ and $\phi(\eta)$ profiles, as represented in the related figures.

The fit curve for $F(\eta)$, $\theta(\eta)$ and $\phi(\eta)$ is illustrated in Figs. 21(e), 22(e) and 23(e). Errors identified by analysing the difference between the ANN's predicted and actual outcomes are shown in Figs. 21(f), 22(f) and 23(f). The accuracy of the ANN model's implementation is reflected in the absolute error.

Figs. 24(a), 24(b) and 24(c) illustrate the comparison between ANN values and FDM results for $F(\eta)$, $\theta(\eta)$, and $\phi(\eta)$. The comparison shows a strong correlation between the values derived from the ANN method and those from the FDM results.

The regression plots for Nu are detailed in Fig. 25. The detailed analysis of the surface plots and contours is visualised in Fig. 26.

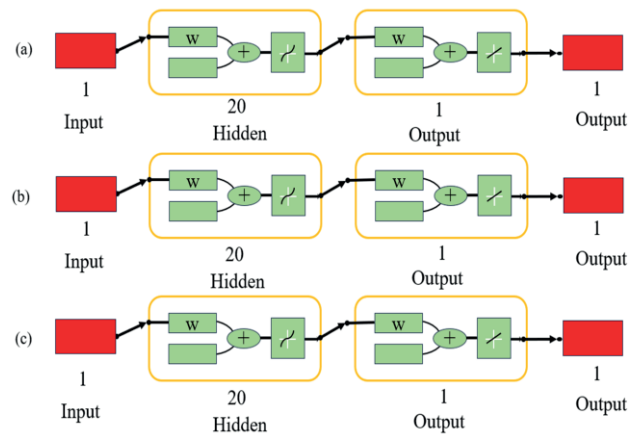


Fig. 20. ANN design.

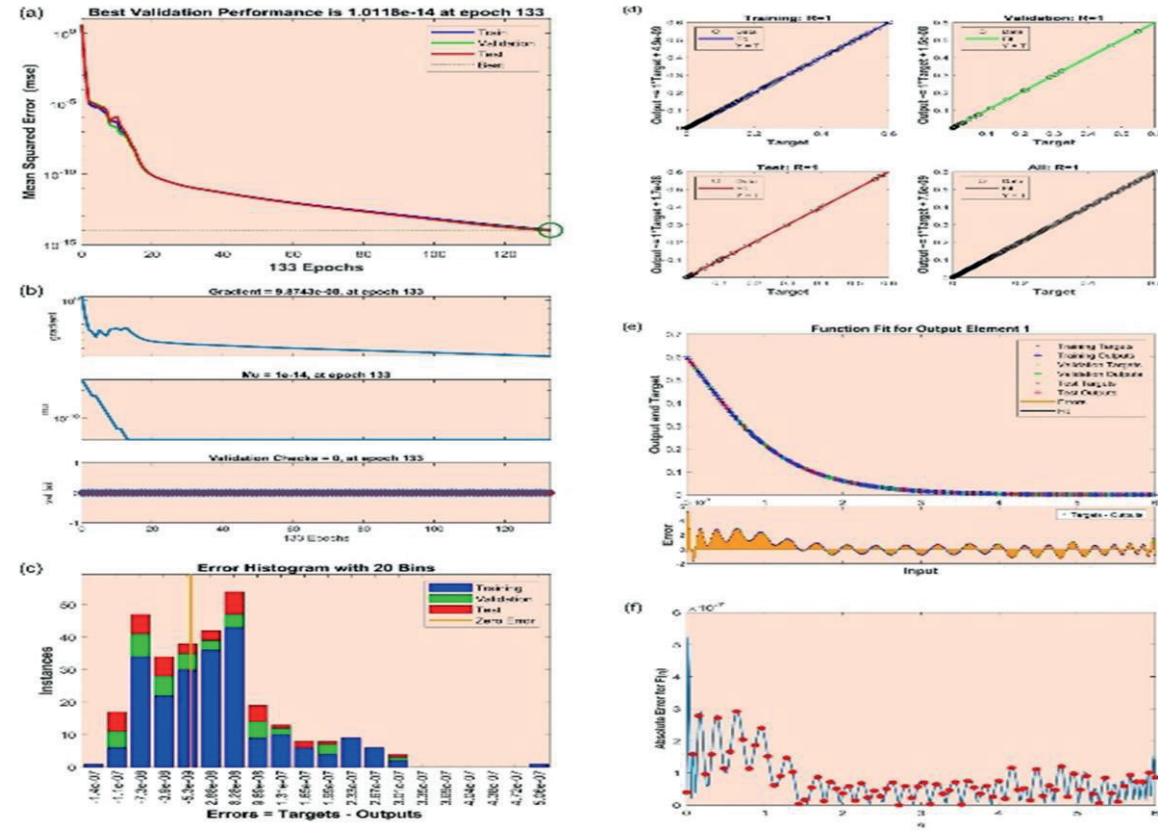


Fig. 21. Visual representation of ANN outcome of (a) performance, (b) training, (c) histogram of error, (d) regression analysis, (e) fit curve and (f) absolute error for $F(\eta)$.

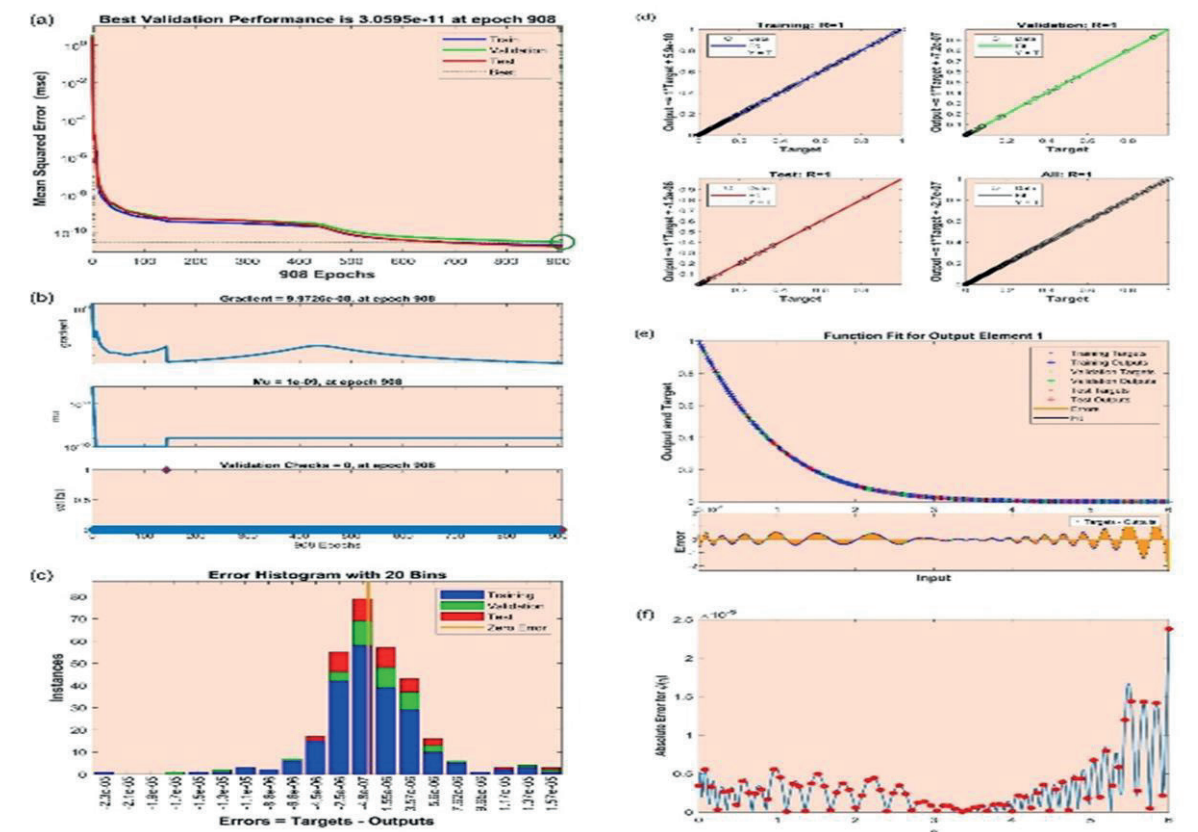


Fig. 23. Visual representation of ANN outcome of (a) performance, (b) training, (c) histogram of error, (d) regression analysis, (e) fit curve and (f) absolute error for $\phi(\eta)$.

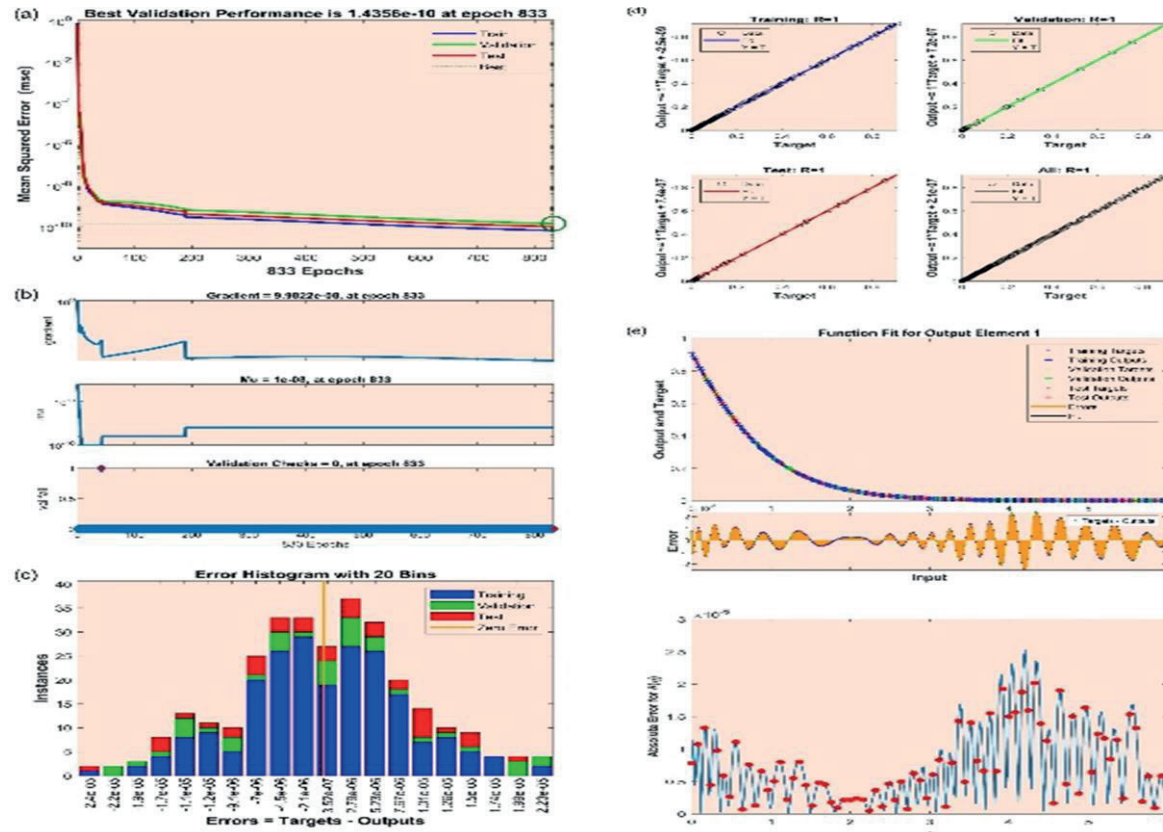


Fig. 22. Visual representation of ANN outcome of (a) performance, (b) training, (c) histogram of error, (d) regression analysis, (e) fit curve and (f) absolute error for $\theta(\eta)$.

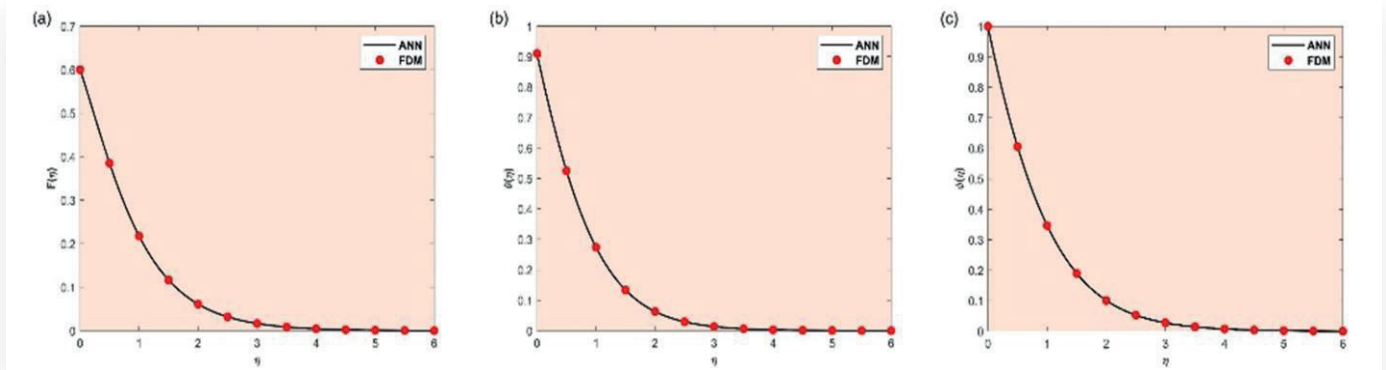


Fig. 24. Comparison of ANN results and FDM outcomes for (a) $F(\eta)$, (b) $\theta(\eta)$ and (c) $\phi(\eta)$.

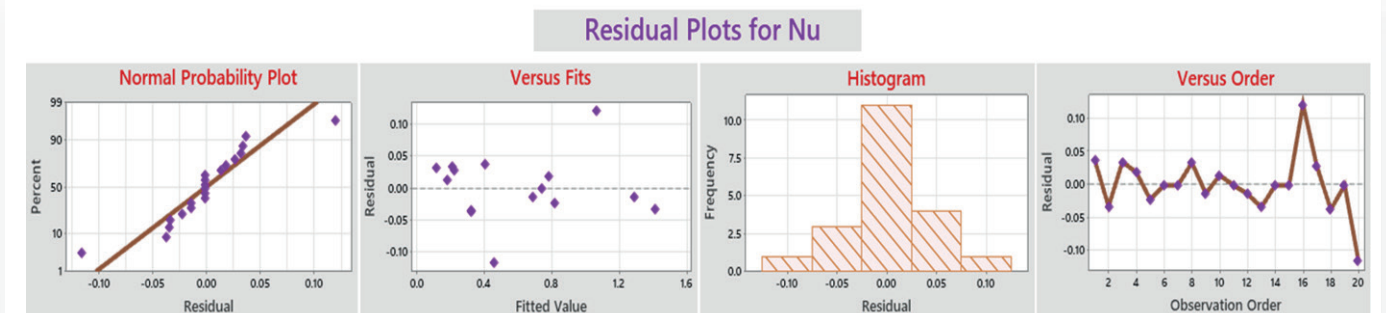


Fig. 25. Regression plots for Nu.

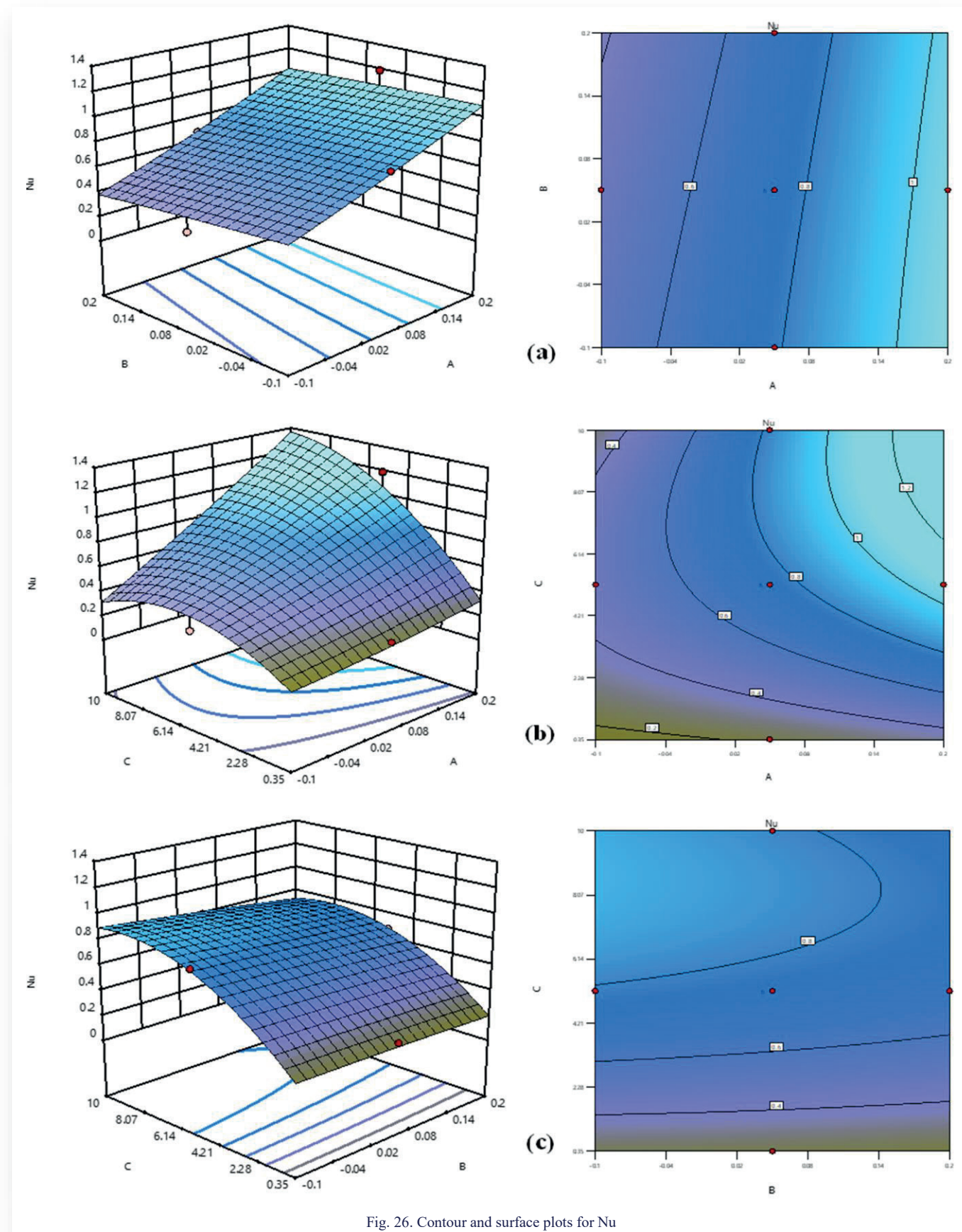


Fig. 26. Contour and surface plots for Nu

7. Conclusions

The current analysis examines the effects of AE, chemical reactions and MF on the time-dependent flow of a Boger liquid over

a slowly rotating stretching disk. The analysis also accounts for the impact of suction, C-BCs and N-HSS on the liquid stream to assess the thermal behaviour. It is anticipated that the primary source of the flow's instability would be the disk's angular ve-

locity. The decreased speed of a disk is explained by the fact that its angular velocity is inversely proportional to time. The governing PDEs are transformed into dimensionless ODEs by using similarity variables. The reduced ODEs are solved using the FDM technique. ANN is used in the study to assess the mass transport, heat transport and fluid flow characteristics. Additionally, RSM is employed to evaluate the HT rate of the fluid flow. Graphical representations show how diverse variables affect the velocity, concentration, and temperature profiles. The following are the main findings of the investigation:

- ❖ $F(\eta)$ and $G(\eta)$ decline as the β_2 rises. Reduced flow rates are the outcome of increasing the relaxation time, excellently reducing the velocity components of the fluid.
- ❖ As the magnetic parameter values increase, $F(\eta)$ and $G(\eta)$ decline. The resistive force causes the liquid velocity components to decrease in magnitude.
- ❖ An increase in β_1 enhances $F(\eta)$ and $G(\eta)$. The fluid's increased reactivity to the flow's driving forces makes the lower viscosity easier for fluid layers to move and deform.
- ❖ $\theta(\eta)$ intensifies as the A^* and B^* rise. The positive values of A^* and B^* could be interpreted physically as heat sources that generate heat, leading to the emission of thermal energy into the flow and an elevation in $\theta(\eta)$.
- ❖ As the values of Bi increase, $\theta(\eta)$ rises. The heat transport coefficient is essential to Bi, since its value increases with an elevation in the heat transportation efficiency.
- ❖ The increase in values of σ^* reduces the concentration profile. Intense chemical reactions have detrimental physical effects that diminish the reactant species, resulting in a mass transfer drop.
- ❖ The $\phi(\eta)$ upsurges as the values of E increase. Reduced AE has been shown to diminish the reaction rate, leading to chemical reaction degradation.
- ❖ The graphical representation illustrates the comparison between ANN values and FDM outputs. The comparison reveals that the results derived from the FDM and ANN approaches are in close agreement, with no significant difference between them.
- ❖ The optimisation of heat transmission is significantly influenced by factors such as the Biot number, space and temperature-dependent heat source/sink parameters.
- ❖ The current analysis with a confidence level of 95% identifies important influential components, revealing an enhanced technique for improving heat transmission.

References

- [1] Chamkha, A.J., & Pop, I. (2004). Effect of thermophoresis particle deposition in free convection boundary layer from a vertical flat plate embedded in a porous medium. *International Communications in Heat and Mass Transfer*, 31(3), 421–430. doi: 10.1016/j.icheatmasstransfer.2004.02.012
- [2] Kattimani, P., Prasad, K.V., Anwar, T., Jena, S.P., Shankhyan, A., & Naveen Kumar, R. (2025). Application of Levenberg-Marquardt artificial neural network to study nanoparticle aggregation phenomena in stagnation point flow towards an off-centered rotating disk. *International Journal of Thermofluids*, 27, 101276. doi: 10.1016/j.ijft.2025.101276
- [3] Varun Kumar, R.S., Chandan, K., Venkadeshwaran, K., Muhammad, T., Punith Gowda, R.J., & Prasannakumara, B.C. (2025). Influence of magnetic dipole on the hybrid nanofluid flow with chemical reactions due to the generated and absorbed heat. *Pramana*, 99(2), 65. doi: 10.1007/s12043-025-02917-4
- [4] Hussain, S.M., Arshad, M., Qureshi, M.A., Guedri, K., Prasad, K.V., Karthik, K., Varun Kumar, R., & Bayram, M. (2025). Entropy analysis of Casson ternary hybrid nanofluid using convective cross diffusion model via spinning sphere based artificial neural network. *Proceedings of the Institution of Mechanical Engineers, Part N: Journal of Nanomaterials, Nanoengineering and Nanosystems*, 23977914251345873. doi: 10.1177/23977914251345873
- [5] Egunjobi, A.S., Makinde, O.D., & Jangili, S. (2017). Unsteady MHD Chemically Reacting and Radiating Mixed Convection Slip Flow Past a Stretching Surface in a Porous Medium. *Defect and Diffusion Forum*, 377, 200–210. doi: 10.4028/www.scientific.net/DDF.377.200
- [6] Norouzi, M., Emamian, A., & Davoodi, M. (2019). An analytical and experimental study on dynamics of a circulating Boger drop translating through Newtonian fluids at inertia regime. *Journal of Non-Newtonian Fluid Mechanics*, 267, 1–13. doi: 10.1016/j.jnnfm.2019.03.005
- [7] Banakar, V.R., Patil, N., Kalia, R., Shankar, B., & Karthik, K. (2025). The role of nanoparticle diameter and solid-fluid interfacial layer behavior under low oscillating magnetic fields: A response surface methodology and ANOVA approach. *Proceedings of the Institution of Mechanical Engineers, Part N: Journal of Nanomaterials, Nanoengineering and Nanosystems*, 23977914251341443. doi: 10.1177/23977914251341443
- [8] Veeranna, Y., Kattimani, P., Kulshreshtha, A., Punith Gowda, R.J., & Sarris, I.E. (2025). Significance of solvent percentage and external pollutant source concentration in Boger-micropolar fluid flow across a semi-circular body. *International Journal of Thermofluids*, 29, 101399. doi: 10.1016/j.ijft.2025.101399
- [9] Hue, S.H., Chagot, L., & Angeli, P. (2024). Studies of interfacial wave properties during displacement with pure viscoelastic fluids in microchannels. *Physics of Fluids*, 36(1), 013109. doi: 10.1063/5.0186036
- [10] Sreehari, P., Gopal, D., Kishan, N., Venkadeshwaran, K., Singh, S., & Karthik, K. (2025). A numerical study on the radiative flow of magneto-Casson fluid via a vertical microchannel with entropy generation. *Journal of the Korean Physical Society*, 86(5), 372–382. doi: 10.1007/s40042-025-01289-7
- [11] Veera Krishna, M., Ameer Ahamad, N., & Chamkha, A.J. (2021). Hall and ion slip impacts on unsteady MHD convective rotating flow of heat generating/absorbing second grade fluid. *Alexandria Engineering Journal*, 60(1), 845–858. doi: 10.1016/j.aej.2020.10.013
- [12] Preethi, K., Hanumagowda, B.N., Pavithra, K.M., Varma, S.V. K., Mann, V.S., Singh, R., & Kattimani, P. (2025). Thermal scrutinization of MHD Darcy–Forchheimer flow of hybrid nanofluid over a stretching sheet with Richardson number and quadratic thermal radiation: Hyperbolic tangent model. *Multiscale and Multidisciplinary Modeling, Experiments and Design*, 8(3), 171. doi: 10.1007/s41939-025-00745-y
- [13] Takhar, H.S., Chamkha, A.J., & Nath, G. (1999). Unsteady flow and heat transfer on a semi-infinite flat plate with an aligned magnetic field. *International Journal of Engineering Science*, 37(13), 1723–1736. doi: 10.1016/S0020-7225(98)00144-X
- [14] Takhar, H.S., Chamkha, A.J., & Nath, G. (2001). Unsteady three-dimensional MHD-boundary-layer flow due to the impulsive motion of a stretching surface. *Acta Mechanica*, 146(1), 59–71. doi: 10.1007/BF01178795
- [15] Jagadha, S., Gopal, D., Patil, N., Singh, S., Kishan, N., & Abhilasha, S.K. (2025). Effect of higher order chemical reaction and

- Joule heating on rotational and permeable radiative MHD flow of Casson manganese ferrite fluid with heat generation/absorption. *Multiscale and Multidisciplinary Modeling, Experiments and Design*, 8(3), 164. doi: 10.1007/s41939-025-00756-9
- [16] Vinay Kumar, V., Sharma, R.P., & Naveen Kumar, R. (2025). The role of time-varying nanofluid flow and particle aggregation on the kinetics of homogeneous and heterogeneous reactions in position-controlled squeezing porous slider. *Proceedings of the Institution of Mechanical Engineers, Part N: Journal of Nanomaterials, Nanoengineering and Nanosystems*, 23977914251338162. doi: 10.1177/23977914251338162
- [17] Banakar, V.R., Kumar, T., Ramachandran, T., Karthikeyan, A., & Karthik, K. (2025). Activity of Motile Microorganisms in Radiative Heat Transfer Analysis of Chemically Reactive Flow Between Stretching Upper and Rotating Lower Disks. *Heat Transfer*, 54(7), 4390–4406. doi: 10.1002/htj.70001
- [18] Thumma, T., Mishra, S.R., Abbas, M.A., Bhatti, M.M., & Abdelsalam, S.I. (2022). Three-dimensional nanofluid stirring with non-uniform heat source/sink through an elongated sheet. *Applied Mathematics and Computation*, 421, 126927. doi: 10.1016/j.amc.2022.126927
- [19] Banakar, V.R., Sollapur, S.B., Kedia, A., Singh, V., & Sarris, I.E. (2025). Levenberg–Marquardt artificial neural network approach to analyze tetra hybrid nanofluid flow over a rough rotating disk under low/high oscillating magnetic fields. *Modern Physics Letters B*. doi: 10.1142/S0217984925502641
- [20] Algehyne, E.A., Alriheli, H.F., Saeed, A., Dawar, A., & Kumam, P. (2022). Cattaneo–Christov heat flux theory toward the magnetohydrodynamic micropolar hybrid nanofluid flow past a stretching/shrinking sheet with non-uniform heat source/sink and thermal radiation. *ZAMM - Journal of Applied Mathematics and Mechanics / Zeitschrift Für Angewandte Mathematik Und Mechanik*, 102(10), e202200168. doi: 10.1002/zamm.202200168
- [21] Salahuddin, T., Arshad, M., Siddique, N., Alqahtani, A.S., & Malik, M.Y. (2020). Thermophysical properties and internal energy change in Casson fluid flow along with activation energy. *Ain Shams Engineering Journal*, 11(4), 1355–1365. doi: 10.1016/j.asej.2020.02.011
- [22] Hayat, T., Riaz, R., Aziz, A., & Alsaedi, A. (2020). Influence of Arrhenius activation energy in MHD flow of third grade nanofluid over a nonlinear stretching surface with convective heat and mass conditions. *Physica A: Statistical Mechanics and Its Applications*, 549, 124006. doi: 10.1016/j.physa.2019.124006
- [23] Asjad, M.I., Zahid, M., Inc, M., Baleanu, D., & Almohsen, B. (2022). Impact of activation energy and MHD on Williamson fluid flow in the presence of bioconvection. *Alexandria Engineering Journal*, 61(11), 8715–8727. doi: 10.1016/j.aej.2022.02.013
- [24] Samanta, A., & Mondal, H. (2023). Numerical simulation of bioconvictional nanofluidic flow in the presence of activation energy past a stretching cylinder subject to swimming micro-organisms. *Pramana*, 97(4), 182. doi: 10.1007/s12043-023-02644-8
- [25] Srilatha, P., Karthik, K., Prasad, K.V., Abdulrahman, A., Kumar, R.S.V., Gowda, R.J.P., & Kumar, R.N. (2024). Dynamics of Fourier’s and Fick’s laws on the convectively heated oscillatory sheet under Arrhenius kinetics: The finite-difference technique. *Journal of Computational Science*, 82, 102428. doi: 10.1016/j.jocs.2024.102428
- [26] Akbar, N.S., Nadeem, S., Ul Haq, R., & Khan, Z.H. (2013). Radiation effects on MHD stagnation point flow of nano fluid towards a stretching surface with convective boundary condition. *Chinese Journal of Aeronautics*, 26(6), 1389–1397. doi: 10.1016/j.cja.2013.10.008
- [27] Malik, R., Khan, M., Munir, A., & Khan, W.A. (2014). Flow and Heat Transfer in Sisko Fluid with Convective Boundary Condition. *PLOS One*, 9(10), e107989. doi: 10.1371/journal.pone.0107989
- [28] Shaw, S., Mahanta, G., & Sibanda, P. (2016). Non-linear thermal convection in a Casson fluid flow over a horizontal plate with convective boundary condition. *Alexandria Engineering Journal*, 55(2), 1295–1304. doi: 10.1016/j.aej.2016.04.020
- [29] Wang, F., Ahmad, S., Al Mdallal, Q., Alammari, M., Khan, M.N., & Rehman, A. (2022). Natural bio-convective flow of Maxwell nanofluid over an exponentially stretching surface with slip effect and convective boundary condition. *Scientific Reports*, 12(1), 2220. doi: 10.1038/s41598-022-04948-y
- [30] Abbas, M., & Khan, N. (2023). Numerical study for bioconvection in Marangoni convective flow of Cross nanofluid with convective boundary conditions. *Advances in Mechanical Engineering*, 15(11). doi: 10.1177/16878132231207625
- [31] Rafiq, T., Mustafa, M., & Farooq, M.A. (2020). Modeling heat transfer in fluid flow near a decelerating rotating disk with variable fluid properties. *International Communications in Heat and Mass Transfer*, 116, 104673. doi: 10.1016/j.icheatmasstransfer.2020.104673
- [32] Ahmad, L., Ahmed, J., & Ahmed, A. (2022). Thermal energy transport in thin film flow of nanofluid over a decelerating rotating disk. *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering*, 236(2), 491–499. doi: 10.1177/09544089211050717
- [33] Ejaz, I., & Mustafa, M. (2022). A comparative study of different viscosity models for unsteady flow over a decelerating rotating disk with variable physical properties. *International Communications in Heat and Mass Transfer*, 135, 106155. doi: 10.1016/j.icheatmasstransfer.2022.106155
- [34] Lin, Y., Cao, L., Tan, Z., & Tan, W. (2024). Magneto-hydrodynamics Marangoni boundary-layer copper/water nanofluid flow driven by surface temperature gradient over a rotating disk. *Physica Scripta*, 99(9), 095305. doi: 10.1088/1402-4896/ad6bcd
- [35] Kamal, A.F.M., Faizan, M., & Zafar, S.S. (2024). Mixed convective magneto flow of nanofluid for the Sutterby fluid containing micro-sized selfpropelled microorganisms with chemical reaction: Keller box analysis. *Physica Scripta*, 99(9), 095202. doi: 10.1088/1402-4896/ad6642
- [36] Rahman, M., Turkyilmazoglu, M., Bilal, M., & Sharif, F. (2023). On heat transfer with unsteady MHD nanofluid von Karman flow with uniform suction. *Pramana*, 97(4), 146. doi: 10.1007/s12043-023-02618-w
- [37] Kattimani, P., Sollapur, S.B., Kedia, A., Singh, K., & Sarris, I.E. (2025). The Role of High-Oscillating Magnetic Field Dynamics in Nanofluid Flow. *Nano*, 2550101. doi: 10.1142/S1793292025501012
- [38] Asma, M., Othman, W.A.M., Muhammad, T., Mallawi, F., & Wong, B.R. (2019). Numerical Study for Magnetohydrodynamic Flow of Nanofluid Due to a Rotating Disk with Binary Chemical Reaction and Arrhenius Activation Energy. *Symmetry*, 11(10), 1282. doi: 10.3390/sym11101282
- [39] Mallikarjuna, A.N., Banakar, V.R., Abdulrahman, A., & Kumar, R.S.V. (2025). Study on thermophoretic deposition of particles and ratio of effective heat capacity: A collocation approach. *International Communications in Heat and Mass Transfer*, 169, 109841. doi:10.1016/j.icheatmasstransfer.2025.109841
- [40] Jyothi, K., Sudarsana Reddy, P., & Suryanarayana Reddy, M. (2018). Influence of magnetic field and thermal radiation on convective flow of SWCNTs-water and MWCNTs-water nanofluid between rotating stretchable disks with convective boundary conditions. *Powder Technology*, 331, 326–337. doi: 10.1016/j.powtec.2018.03.020
- [41] Narayana, P.V.S., Akshit, S.M., Ghor, J.P., & Venkateswarlu, B. (2017). Thermal Radiation Effects on an Unsteady MHD Nanofluid Flow Over a Stretching Sheet with Non-Uniform Heat Source/Sink. *Journal of Nanofluids*, 6(5), 899–907. doi: 10.1166/jon.2017.1374
- [42] Acharya, N. (2021). Spectral simulation to investigate the effects of nanoparticle diameter and nanolayer on the ferrofluid flow over a slippery rotating disk in the presence of low oscillating magnetic field. *Heat Transfer*, 50(6), 5951–5981. doi: 10.1002/htj.22157
- [43] Varun Kumar, R.S., Kumar, R.N., Ben Ahmed, S., Madhu, J., Verma, A., & Punith Gowda, R.J. (2024). Unsteady flow of a ternary nanofluid over a slow-rotating disk subject to uniform suction and backpropagated neural network. *Numerical Heat Transfer, Part B: Fundamentals*, 85(12), 1705–1725. doi: 10.1080/10407790.2023.2269610
- [44] Turkyilmazoglu, M. (2014). Nanofluid flow and heat transfer due to a rotating disk. *Computers and Fluids*, 94, 139–146. doi: 10.1016/j.compfluid.2014.02.009



Co-published by
Institute of Fluid-Flow Machinery
Polish Academy of Sciences
Committee on Thermodynamics and Combustion
Polish Academy of Sciences

Copyright©2026 by the Authors under licence CC BY-NC-ND 4.0

<http://www.imp.gda.pl/archives-of-thermodynamics/>



Influence of spacing of waterwall tubes on their operating parameters

Marek Majdak

Cracow University of Technology, Faculty of Environmental Engineering and Energy, Department of Energy, Warszawska 24, 31-155 Cracow, Poland
Author email: marek.majdak@pk.edu.pl

Received: 01.10.2025; revised: 20.03.2026; accepted: 22.04.2026

Abstract

The article presents the results of thermal and flow analysis of the working conditions of neighbouring waterwall tubes, loaded with heat streams of different values for two different widths of tube spacing. The numerical model used for the analysis allows us to calculate the temperature distribution at the tubes and in the fluid flowing through them at each time step, depending on the thermo-physical parameters of the fluid and the material from which the tubes and fins were made. By using the algorithm, it is possible to precisely determine the three-dimensional temperature distribution for tubes, allowing us to determine the places where the most divergent temperatures occur and in which thermal stresses of the highest value may occur. The analysis of several adjacent tubes will allow for determining the effect of temperature differences in the tubes and fins connecting them, and to collect data that may be used for the determination of stress distribution in the tubes and fins. In the presented case 3, smooth tubes connected by fins were used, and the heat flux on the centre tube was by 50% higher than on the outer ones.

Keywords: Waterwall tube; Supercritical steam boiler; Temperature distribution; Thermal stresses

Vol. 47(2026), No. 2, 87–97; doi: 10.24425/ather.2026.158676

Cite this manuscript as: Majdak, M. (2026). Influence of spacing of waterwall tubes on their operating parameters. *Archives of Thermodynamics*, 47(2), 87–97.

1. Introduction

In subcritical boilers, the heat flux distribution on the width of the combustion chamber wall is close to uniform. Therefore, the temperature of the walls of neighbouring waterwall tubes is similar. In supercritical boilers, the heat flux distribution on the width of the combustion chamber wall is uneven – on the centre of the wall, its value is about 50% higher than close to the combustion chamber corners [1].

The most common method of reduction of thermal stresses resulting from uneven heating of neighbouring waterwall tubes is differentiation of mass flux of fluid flowing inside tubes – the mass flow in tubes near the centre of the combustion chamber wall is higher than in tubes close to the corners of the combustion chamber. Another method analysed by designers of com-

bustion chambers is to vary the spacing of the waterwall tubes. In some publications, the idea of reducing the spacing near the centre of the combustion chamber wall to increase the number of waterwall tubes on the unit width of the wall is presented.

2. Literature review

Many publications may be found in the literature regarding the heat and mass transfer in subcritical and supercritical boilers. Authors of [2] proposed one of the first computer-based monitoring systems for boilers with natural water circulation. A computer system allows the calculation of, among others, the boiler efficiency, combustion gas mass flow rate, and heat load at the walls of the combustion chamber. The results of the calculations were validated with the measurement data.

Nomenclature

A – tube cross-section surface area, m^2
 c – specific heat per mass unit, $kJ/(kg \cdot K)$
 $d\tau$ – time step, s
 e – energy per fluid mass, J/kg
 g – gravitational acceleration, m/s^2
 i – enthalpy, kJ/kg
 $\dot{m}$ – mass flow
 p – pressure, kPa
 q – heat flux, W/m^2
 r – radius, m
 s – tube spacing, m
 t – fluid temperature, $^{\circ}C$
 u – unit internal energy, J/kg
 U – tube perimeter, m
 V – fluid volume, m^3

Greek symbols

α – heat transfer coefficient, $W/(m^2 \cdot K)$

Δr – tube wall thickness, m
 ΔV – finite volume, m^3
 Δx – fin width, m
 Δz – length of control volume, m
 $\Delta \varphi$ – characteristic angle, $^{\circ}$
 δ – thickness, m
 θ – temperature of wall material, $^{\circ}C$
 λ – thermal conductivity, $W/(m \cdot K)$
 ρ – density, kg/m^3
 σ_r – tangential stresses, N/m^2
 τ – time, s

Subscripts and Superscripts

in – inner
 j – spatial step number
 m – medium
 o – outer
 tw – tube wall
 τ – time step number
 w – wall

Experimental measurements performed for the ultra-supercritical boiler were presented in [3]. Authors of the publication monitored the main thermal and flow parameters of steam flowing inside the waterwall tube for different steam pressures and boiler loads. During the tests, steam reached subcritical, near-critical and supercritical pressures, so the conclusions drawn from the analysis of the results can be successfully applied to a wide range of steam flows.

Results of numerical simulations of thermo-physical parameters of supercritical steam were also presented in [4]. A four-pass steam superheater was used as an analysis system for the calculation of the specific heat of steam based on the temperature and pressure of steam, and the temperature of the waterwall tube wall. The heat transfer of a cross-counter-current superheater consisting of 48 tubes was analysed.

Spirally rifled waterwall tubes are used in power boilers to reduce thermal stresses and allow faster heat-transfer stabilisation. Results of numerical simulations of thermal and flow phenomena for smooth and internally ribbed tubes mounted vertically were presented by the authors of [5–8]. The calculations were performed to examine the temperature and thermal stress distributions in tubes, and the thermo-physical parameters of the working fluid. In [7], numerical data were also verified with experimental data from the measurement performed on the dedicated experimental stand.

Authors of [9] presented the results of experimental studies on the application of internally rifled vertical tubes in a combustion chamber. They have listed the advantages of this system of waterwall tubes compared to spirally wound smooth tubes. In the paper, the main advantages of using internal spirally rifled tubes can be found in the field of installation, flow characteristics, flow stability and load follow-up capability, as well as a short analysis of possible directions for optimising this type of system was presented.

Another possibility of minimising the thermal and pressure stresses can be the use of corrugated furnace tubes. Authors of [10] analysed stress distribution for different corrugations of

depths and spacings of the tube. It was found that in this high-power fire-tube boiler, a corrugated furnace tube with a greater corrugation depth should be used, because it allows for the reduction of the maximum value of stresses in the structure.

In publications [11–13], cases were analysed considering the influence of the variability of heat load with increasing height above the inlet to the waterwall tubes on the thermo-physical parameters of walls and tubes. In the analysed studies, an additional measuring insert was used, located on the front wall of the tube, which allowed for obtaining more accurate temperature values.

Waterwall tubes mounted on the combustion chamber walls are heated on one side, which results in a significant temperature difference between front and rear tube walls [14,15]. Due to the uneven temperature distribution in the cross-sections of the tubes, their division into smaller elements is often analysed. In publications [16–20], a model of the division of the half cross-section of the waterwall tube with the fin on the control volumes was presented. In the articles, temperature distributions in tubes and fins were presented, obtained by simulation performed for the operating parameters of a supercritical boiler.

Authors of [21] analysed the temperature distribution in the wall of the waterwall tube covered with ash deposits on the outer surface of the tube. The calculation of heat transfer for tubes with scale deposits on the inner surface of the tube was also presented. A complex numerical model can be used for calculations for steam, tube wall, and flue gas temperature for the entire superheater.

In the article [22], the results of numerical simulation carried out by using the Runge-Kutta method to solve the equations of energy, mass and momentum balance were presented. The authors proposed a calculation algorithm for the medium flowing inside the waterwall tubes of the furnace chamber. The presented results were based on parameters of the supercritical furnace with a steam capacity of 2400×10^3 kg/h.

The computational models proposed in the literature allow for carrying out calculations for one or several waterwall tubes,

with various calculation details. The novelty proposed in this publication will be the analysis of the change in temperature distribution depending on the width of spacing of the waterwall tubes under the same initial conditions.

3. Problem analysis

In the article, an analysis of temperature distribution in waterwall tubes connected by fins was presented. The proposed model allows for determining the three-dimensional temperature distribution, which will be presented for the cross-sections of waterwall tubes and fins between them, taking into account the variability of thermo-physical parameters of fluids and tubes in every control volume at each time step.

The analysis was performed by using parameters of water and steam calculated on the basis of the International Association for the Properties of Water and Steam table and the properties of 13CrMo4-5 steel received from its manufacturer.

As input data, the operating parameters from the supercritical boiler operating in one of the Polish power plants were used. Due to the fact that the boiler generates steam with supercritical parameters, the analysis took into account the uneven distribution of the heat flux density across the width of the combustion chamber wall. Based on experimental studies conducted by many teams, it was determined that the heat flux incident on the combustion chamber walls near the corners is up to 50% lower than the heat flux incident on the wall at half its width at the same height; a graphical interpretation of this condition is presented in Fig. 1 [1].

The model of the division of the cross-section of waterwall tubes connected by fins was proposed in [23,24]. In the cross-section of the tube with two halves of fins, 36 characteristic points are identified (Fig. 2); this method of division can be scaled to a system consisting of multiple tubes, as presented for three tubes in Fig. 3.

The energy balance equations written for all elements of the analysed system allow for the determination of the temperature

distribution in the waterwall tubes connected by fins. Equations for selected control volumes are presented below.

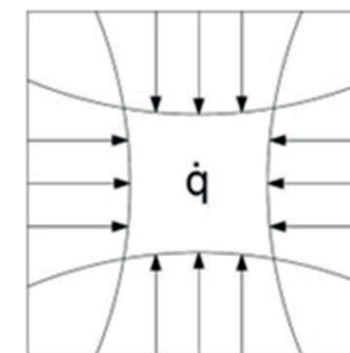


Fig. 1. The distribution of the thermal flux density in the cross-section of the supercritical boiler's furnace chamber.

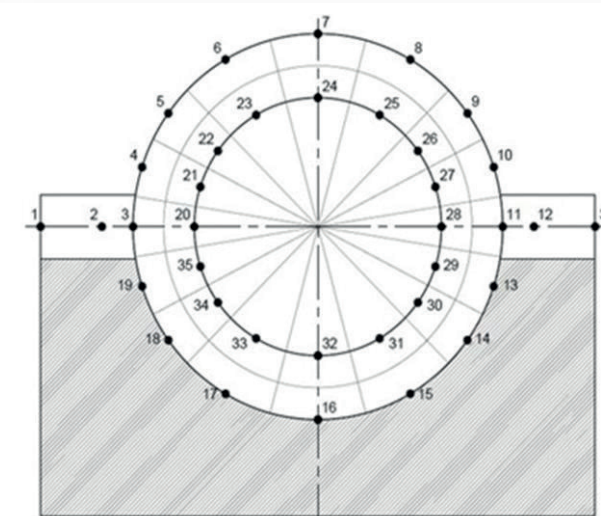


Fig. 2. The method of division of the cross-section of the waterwall tube with two halves of fins into control volumes.

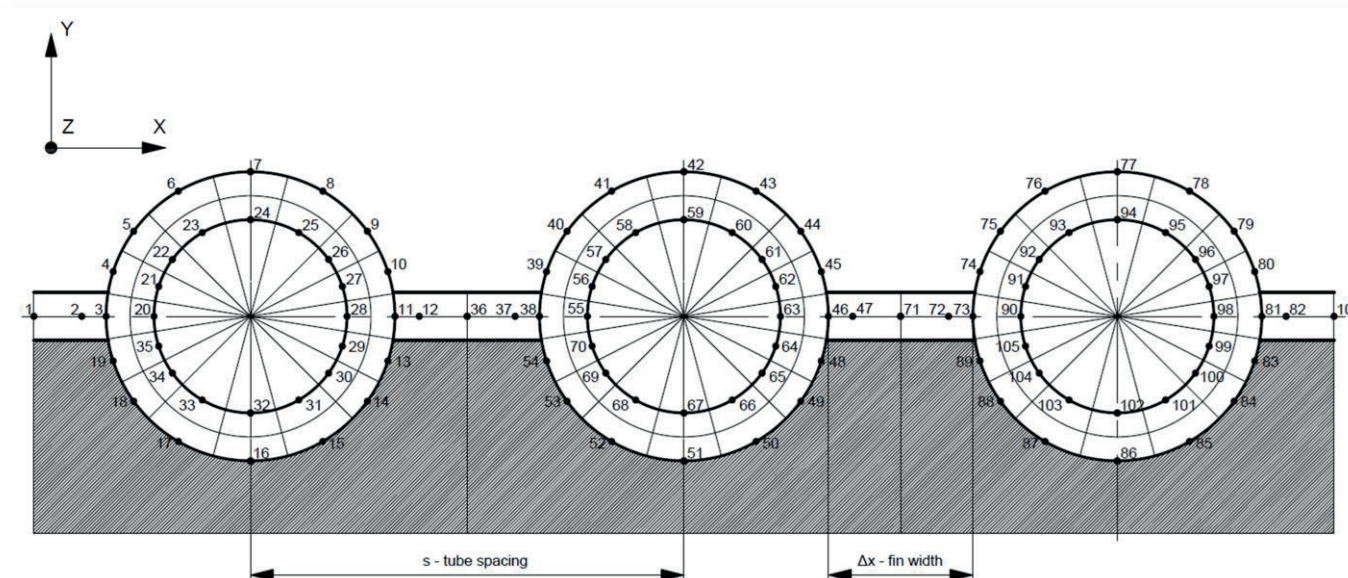


Fig. 3. The division of the cross-section of three waterwall tubes with fins into control volumes with the adopted coordinate system.

For the point on the centre of the outer surface of the front wall of the tube (“7” in Fig. 4), the energy balance equation takes the form of Eq. (1). Analogical equations can be obtained for points “42” and “77” by increasing the numbers in the lower index by 35 or 70, respectively.

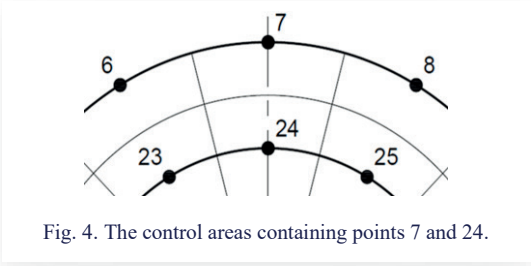


Fig. 4. The control areas containing points 7 and 24.

$$c_{7,j}\rho_{7,j}\frac{\Delta\hat{\phi}_1}{2}(r_o^2 - r_m^2)\Delta z\frac{d\theta_{7,j}}{d\tau} = \lambda_{7,j}\frac{\theta_{6,j}-\theta_{7,j}}{\Delta\hat{\phi}_1 r_o}\frac{\Delta r}{2}\Delta z + \lambda_{7,j}\frac{\theta_{8,j}-\theta_{7,j}}{\Delta\hat{\phi}_1 r_o}\frac{\Delta r}{2}\Delta z + \lambda_{7,j}\frac{\theta_{24,j}-\theta_{7,j}}{\Delta r}\Delta\hat{\phi}_1 r_m\Delta z + \lambda_{7,j}\frac{\Delta\hat{\phi}_1}{2}(r_o^2 - r_m^2)\frac{\theta_{7,j-1}-\theta_{7,j}}{\Delta z} + \lambda_{7,j}\frac{\Delta\hat{\phi}_1}{2}(r_o^2 - r_m^2)\frac{\theta_{7,j+1}-\theta_{7,j}}{\Delta z} + \dot{q}_{7,j}\Delta\hat{\phi}_1 r_o\Delta z. \quad (1)$$

For the point on the centre of the inner surface of the front wall of the tube (“24” in Fig. 4), the energy balance equation takes the form of Eq. (2). Analogical equations can be obtained for points “59” and “94” by increasing the numbers in the lower index by 35 or 70, respectively.

$$c_{24,j}\rho_{24,j}\frac{\Delta\hat{\phi}_1}{2}(r_m^2 - r_{in}^2)\Delta z\frac{d\theta_{24,j}}{d\tau} = \lambda_{24,j}\frac{\theta_{25,j}-\theta_{24,j}}{\Delta\hat{\phi}_1 r_{in}}\frac{\Delta r}{2}\Delta z + \lambda_{24,j}\frac{\theta_{23,j}-\theta_{24,j}}{\Delta\hat{\phi}_1 r_{in}}\frac{\Delta r}{2}\Delta z + \lambda_{24,j}\frac{\theta_{7,j}-\theta_{24,j}}{\Delta r}\Delta\hat{\phi}_1 r_m\Delta z +$$

$$\theta_{7,j}^{\tau+\Delta\tau} = \theta_{7,j}^{\tau} + \frac{\Delta\tau}{c_{7,j}\rho_{7,j}\frac{\Delta\hat{\phi}_1}{2}(r_o^2 - r_m^2)\Delta z} [\lambda_{7,j}^{\tau}\frac{\theta_{6,j}-\theta_{7,j}}{\Delta\hat{\phi}_1 r_o}\frac{\Delta r}{2}\Delta z + \lambda_{7,j}^{\tau}\frac{\theta_{8,j}-\theta_{7,j}}{\Delta\hat{\phi}_1 r_o}\frac{\Delta r}{2}\Delta z + \lambda_{7,j}^{\tau}\frac{\theta_{24,j}-\theta_{7,j}}{\Delta r}\Delta\hat{\phi}_1 r_m\Delta z + \lambda_{7,j}^{\tau}\frac{\Delta\hat{\phi}_1}{2}(r_o^2 - r_m^2)\frac{\theta_{7,j-1}-\theta_{7,j}}{\Delta z} + \lambda_{7,j}^{\tau}\frac{\Delta\hat{\phi}_1}{2}(r_o^2 - r_m^2)\frac{\theta_{7,j+1}-\theta_{7,j}}{\Delta z} + \dot{q}_{7,j}\Delta\hat{\phi}_1 r_o\Delta z], \quad (4)$$

$$\theta_{24,j}^{\tau+\Delta\tau} = \theta_{24,j}^{\tau} + \frac{\Delta\tau}{c_{24,j}\rho_{24,j}\frac{\Delta\hat{\phi}_1}{2}(r_m^2 - r_{in}^2)\Delta z} [\lambda_{24,j}^{\tau}\frac{\theta_{25,j}-\theta_{24,j}}{\Delta\hat{\phi}_1 r_{in}}\frac{\Delta r}{2}\Delta z + \lambda_{24,j}^{\tau}\frac{\theta_{23,j}-\theta_{24,j}}{\Delta\hat{\phi}_1 r_{in}}\frac{\Delta r}{2}\Delta z + \lambda_{24,j}^{\tau}\frac{\theta_{7,j}-\theta_{24,j}}{\Delta r}\Delta\hat{\phi}_1 r_m\Delta z + \lambda_{24,j}^{\tau}\frac{\Delta\hat{\phi}_1}{2}(r_m^2 - r_{in}^2)\frac{\theta_{24,j-1}-\theta_{24,j}}{\Delta z} + \lambda_{24,j}^{\tau}\frac{\Delta\hat{\phi}_1}{2}(r_m^2 - r_{in}^2)\frac{\theta_{24,j+1}-\theta_{24,j}}{\Delta z} + \alpha_j^{\tau}(t_j^{\tau} - \theta_{24,j}^{\tau})\Delta\hat{\phi}_1 r_{in}\Delta z], \quad (5)$$

$$\theta_{36,j}^{\tau+\Delta\tau} = \theta_{36,j}^{\tau} + \frac{\Delta\tau}{c_{36,j}\rho_{36,j}\Delta x\delta_w\Delta z} [\lambda_{36,j}^{\tau}\frac{\theta_{12,j}-\theta_{36,j}}{\Delta x}\delta_w\Delta z + \lambda_{36,j}^{\tau}\frac{\theta_{37,j}-\theta_{36,j}}{\Delta x}\delta_w\Delta z + \lambda_{36,j}^{\tau}\Delta x\delta_w\frac{\theta_{36,j+1}-\theta_{36,j}}{\Delta z} + \dot{q}_{36,j}\frac{\Delta x}{3}\Delta z]. \quad (6)$$

In order to determine the average temperature of the fluid in the cross-sections of waterwall tubes, the mass, energy and momentum balance equations were used. The balance equations were used in the form of separable differential equations – on the left side, expressions containing time derivatives were presented, while on the right side, spatial derivatives were replaced using the backward difference quotient method [22].

The mass balance equation takes the form:

$$\frac{\partial(\Delta V\rho)}{\partial\tau} = (A\rho v)|_{z-(\frac{\partial z}{2})} - (A\rho v)|_{z+(\frac{\partial z}{2})}. \quad (7)$$

$$+ \lambda_{24,j}\frac{\Delta\hat{\phi}_1}{2}(r_m^2 - r_{in}^2)\frac{\theta_{24,j-1}-\theta_{24,j}}{\Delta z} + \lambda_{24,j}\frac{\Delta\hat{\phi}_1}{2}(r_m^2 - r_{in}^2)\frac{\theta_{24,j+1}-\theta_{24,j}}{\Delta z} + \alpha_j(t_j - \theta_{24,j})\Delta\hat{\phi}_1 r_{in}\Delta z. \quad (2)$$

For the point in the centre of the fin between waterwall tubes (“36” in Fig. 5), the energy balance equation takes the form of Eq. (3). An analogical equation can be obtained for point “71” by increasing the number in the lower index by 35. Equations for points “1” and “106” located at the ends of the analysed fragment of the cross-section of the combustion chamber wall can be determined assuming the symmetry of parameters for adjacent characteristic points.

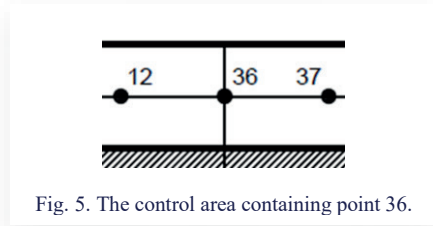


Fig. 5. The control area containing point 36.

$$c_{36,j}\rho_{36,j}\Delta x\delta_w\Delta z\frac{d\theta_{36,j}}{d\tau} = \lambda_{36,j}\frac{\theta_{37,j}-\theta_{36,j}}{\Delta x}\delta_w\Delta z + \lambda_{36,j}\frac{\theta_{12,j}-\theta_{36,j}}{\Delta x}\delta_w\Delta z + \lambda_{36,j}\Delta x\delta_w\frac{\theta_{36,j+1}-\theta_{36,j}}{\Delta z} + \lambda_{36,j}\Delta x\delta_w\frac{\theta_{36,j-1}-\theta_{36,j}}{\Delta z} + \dot{q}_{36,j}\frac{\Delta x}{3}\Delta z. \quad (3)$$

After transforming Eqs. (1)–(3), the dependencies for determining the temperature values are obtained in the form of Eqs. (4)–(6):

The momentum conservation law takes the form:

$$\frac{\partial(Av\rho\Delta z)}{\partial\tau} = (\dot{m}v)|_{z-(\frac{\partial z}{2})} - (\dot{m}v)|_{z+(\frac{\partial z}{2})} + A \cdot (p|_{z-(\frac{\partial z}{2})} - p|_{z+(\frac{\partial z}{2})}) - \sigma\tau U\Delta z - \rho g\Delta V \sin\varphi. \quad (8)$$

The energy of the fluid, related to unit mass, must be analysed as a sum of internal, kinetic and potential energy:

$$e = \frac{v^2}{2} + gz \sin\varphi + u. \quad (9)$$

For the purpose of this analysis, it was assumed that the work done by surface and volume forces is negligible. After rearran-

$$\frac{\partial i}{\partial\tau} = \left(1 - \frac{1}{\rho}\frac{\partial p}{\partial i}\right)^{-1} \left(\frac{\dot{m}}{A\rho}\left(\frac{1}{\rho}\frac{\partial p}{\partial z} - \frac{\partial i}{\partial z} + \frac{1}{\rho}\frac{\partial p_{\tau}}{\partial z}\right) + \frac{4\alpha(\theta - t)}{2r_{in}\rho} - \frac{1}{A\rho}\frac{\partial p}{\partial\rho}\frac{\partial\dot{m}}{\partial z}\right). \quad (10)$$

By using the method presented in [24], balance equations for fluid can be solved and written in a form presented below for each time step “ τ ” and each spatial step “ j ”, with space derivatives on the left side of the equation and time derivatives replaced with difference quotients on the right side. Therefore, the following equations were applied to the numerical model:

- the energy balance equation:

$$\frac{di_j^{\tau}}{dz} = \frac{A\rho_j^{\tau-\Delta\tau}}{\dot{m}_j^{\tau-\Delta\tau}} \left(-\frac{i_j^{\tau}-i_j^{\tau-\Delta\tau}}{\Delta\tau} + 4\alpha_j^{\tau-\Delta\tau} \cdot \frac{\theta_j^{\tau-\Delta\tau} - t_j^{\tau-\Delta\tau}}{2r_{in}\rho_j^{\tau-\Delta\tau}} \right), \quad (11)$$

where the density of the fluid can be found as a function of enthalpy and pressure of the fluid:

$$\rho_j^{\tau} = f(i_j^{\tau}, p_j^{\tau-\Delta\tau}), \quad (12)$$

- the mass balance equation:

$$\frac{d\dot{m}_j^{\tau}}{dz} = -A \frac{\rho_j^{\tau} - \rho_j^{\tau-\Delta\tau}}{\Delta\tau}, \quad (13)$$

- the momentum balance equation:

$$\frac{d}{dz} \left(\frac{(\dot{m}^2)_j^{\tau}}{A^2 \rho_j^{\tau}} + p_j^{\tau} \right) = -\frac{1}{A} \frac{(\dot{m}^2)_j^{\tau} - (\dot{m}^2)_j^{\tau-\Delta\tau}}{\Delta\tau} - \frac{dp_{\tau}}{dz} - \rho_j^{\tau} g \sin\varphi. \quad (14)$$

Using the relations (11)–(14), the parameters of the fluid in each cross-section of the tube can be determined in each time step. In the proposed model, the heat transfer coefficient for the supercritical fluid flow is calculated on the inner surface of the tube by using the dependency determined by Kitoh et al. [25].

In order to compare the temperature distributions in the waterwall tubes connected by fins of different widths, the operating conditions of a supercritical boiler working in the supercritical stage were used. The geometrical parameters of tubes and fins in the first case were adopted directly from the combustion chamber walls geometry; in the second case, the width of fins between tubes was increased. The working fluid is single-phase because its thermodynamic parameters exceed the water critical point parameters. A model was developed that allows online calculation of parameters of waterwall tubes and working fluid, in which the assumptions mentioned below were taken into account.

The fluid temperature inside the tubes was assumed to be constant; it is also assumed that the parameters of the fluid in the entire cross-section of the tube are the same. Equations for points “1” and “106” located at the ends of the analysed fragment of the cross-section of the combustion chamber wall (Fig. 3) can be determined assuming the symmetry of parameters for adjacent characteristic points. For the purpose of this analysis, it was assumed that the work done by surface and volume forces is negligible. To approximate the actual simulation conditions, a plane strain state was assumed, meaning that there was no deformation in the Z-axis direction perpendicular to the

gements presented in [22], Eq. (9) can be presented in the form of a differential equation with separable variables:

analysed cross-section. One side of the model was fixed to a point, and the other side was blocked from moving relative to the Y-axis.

The most important adopted parameters are:

- mass flow per tube: $\dot{m} = 0.77$ kg/s,
- water pressure at the tube inlet: $p_{in} = 299.6$ bar,
- waterwall tube material: 16Mo3 steel,
- outer tube diameter: $d_o = 33.7$ mm,
- tube wall thickness: $\delta_w = 6.1$ mm,
- tube spacing: $s_1 = 50$ mm (case 1), $s_2 = 58.15$ mm (case 2).

For the purpose of the analysis, it was assumed that the average heat flux would vary with the height of the combustion chamber according to the characteristics shown in Fig. 6.

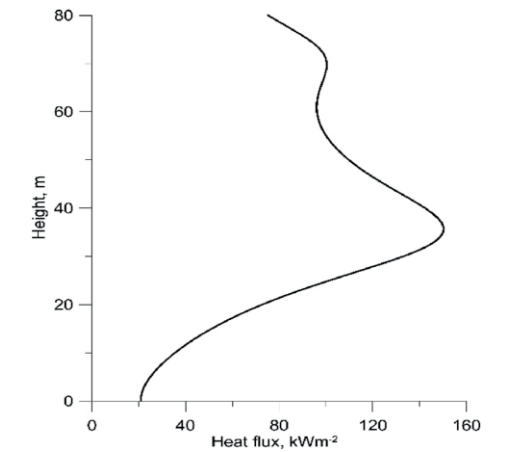


Fig. 6. Distribution of the average heat flux at the furnace waterwalls along the height of the combustion chamber.

The analysed waterwall tubes are 166.5 m long, and the assumed step along the tube length (along the Z axis) is 0.75 m. The entire analysed model was divided into 222 levels with a total number of characteristic points equal to 23 532.

The comparison of temperature values obtained in both cases is presented in the next chapter.

4. Results and discussion

Two cases of heating of the system were analysed, consisting of three tubes connected by fins. In each case, the heat flux on the centre tube was by 50% higher than that on the outer tubes. In the second case, the tube spacing was increased from 50 mm to 58.15 mm to compare the temperature and thermal stress distribution in cross-sections of the analysed tubes.

The temperature distribution at the characteristic points obtained from the developed numerical algorithm was imported into the ANSYS environment as input data and graphically interpolated. Thermodynamic parameters of the material were determined according to data provided by the steel manufacturer.

The mounting conditions for the analysed tube cross-sections and connecting fins were determined, allowing for the simulation of the operating conditions of the entire combustion chamber wall section. On the basis of the imported temperature distribution and determined parameters, the static distribution of the thermal stresses is calculated. Due to the relatively simple geometry analysed in this single case, a 0.25 mm quadrangular mesh was applied to the system.

The temperature distributions obtained using the numerical model for the case with “standard” tube spacing (fin width is equal to 16.3 mm) are presented in Figs. 7–9, for the case with wider tube spacing (fin width is equal to 24.45 mm) in Figs. 10–12. The thermal stress distributions for the case with “standard” tube spacing are presented in Figs. 13–15, for the case with a wider tube spacing in Figs. 16–18.

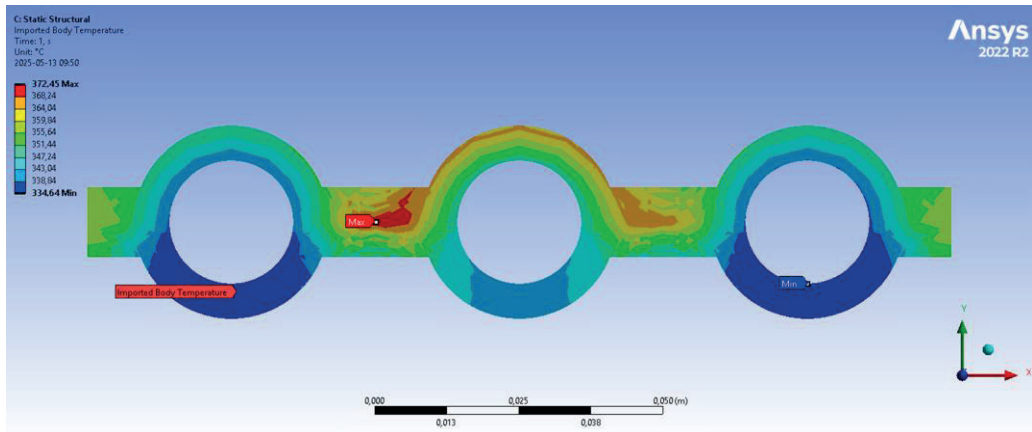


Fig. 7. The temperature distribution in the cross-section of the analysed system with "standard" tube spacing at a distance of 40 m from the tube inlet.

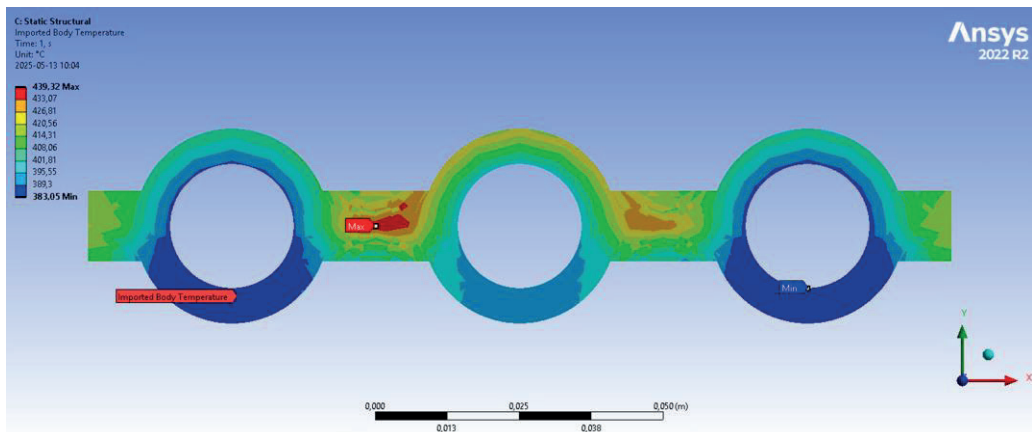


Fig. 8. The temperature distribution in the cross-section of the analysed system with "standard" tube spacing at a distance of 80 m from the tube inlet.

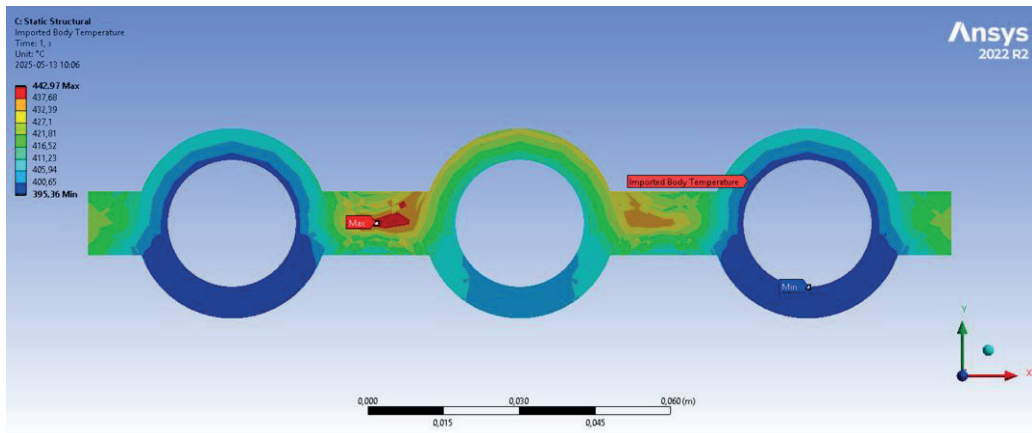


Fig. 9. The temperature distribution in the cross-section of the analysed system with "standard" tube spacing at a distance of 120 m from the tube inlet.

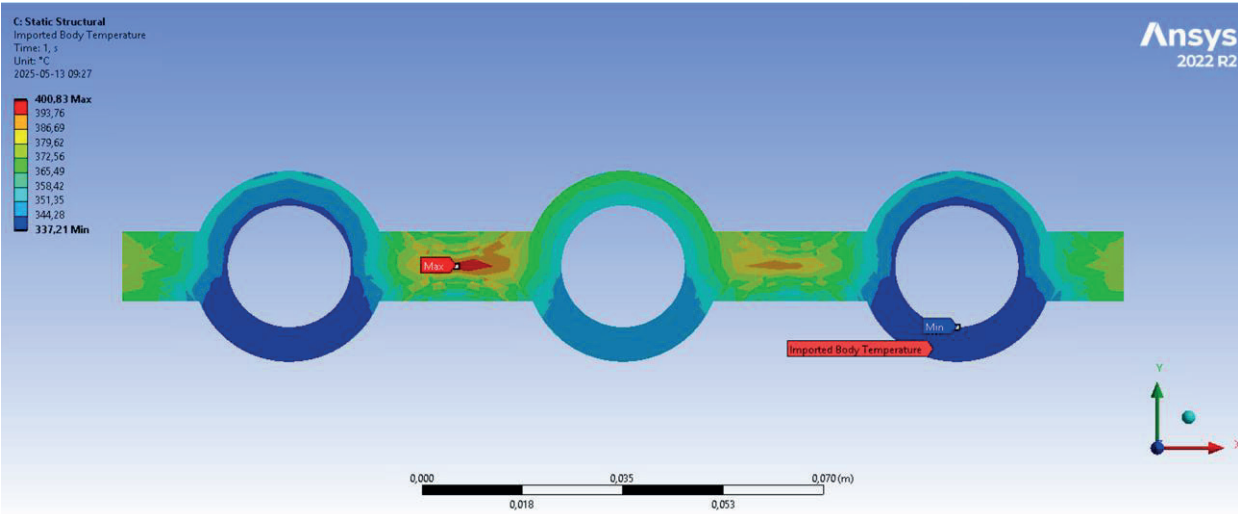


Fig. 10. The temperature distribution in the cross-section of the analysed system with a wider tube spacing at a distance of 40 m from the tube inlet.

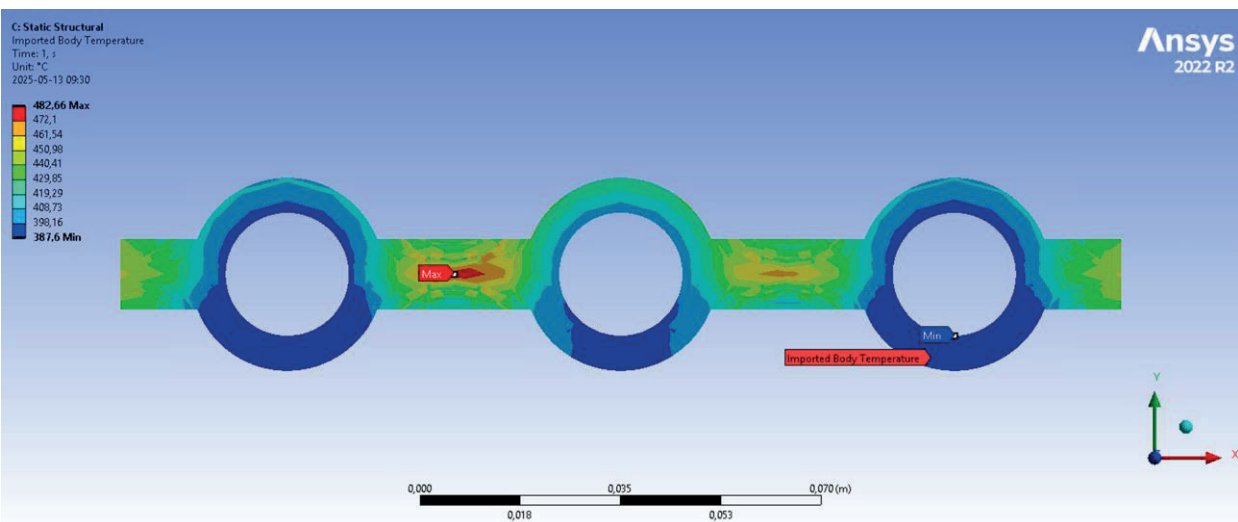


Fig. 11. The temperature distribution in the cross-section of the analysed system with a wider tube spacing at a distance of 80 m from the tube inlet.

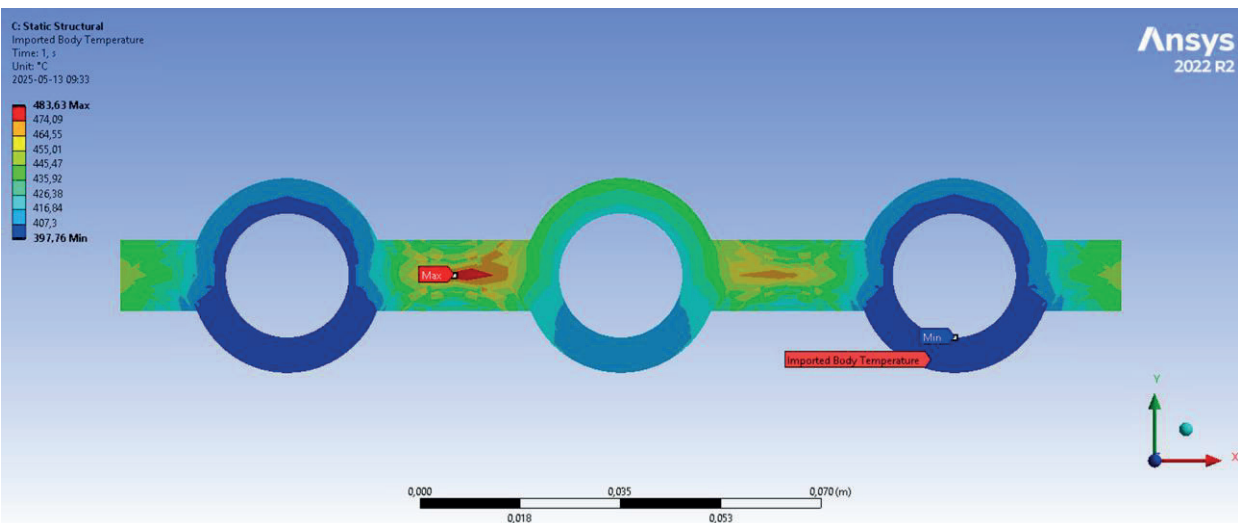


Fig. 12. The temperature distribution in the cross-section of the analysed system with a wider tube spacing at a distance of 120 m from the tube inlet.

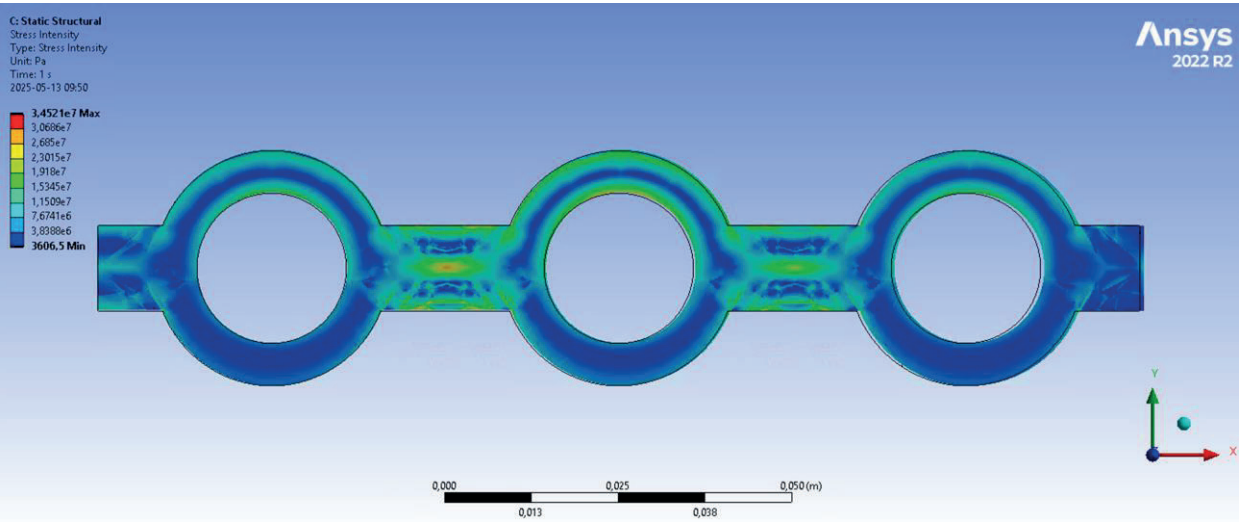


Fig. 13. The thermal stress distribution in the cross-section of the analysed system with "standard" tube spacing at a distance of 40 m from the tube inlet.

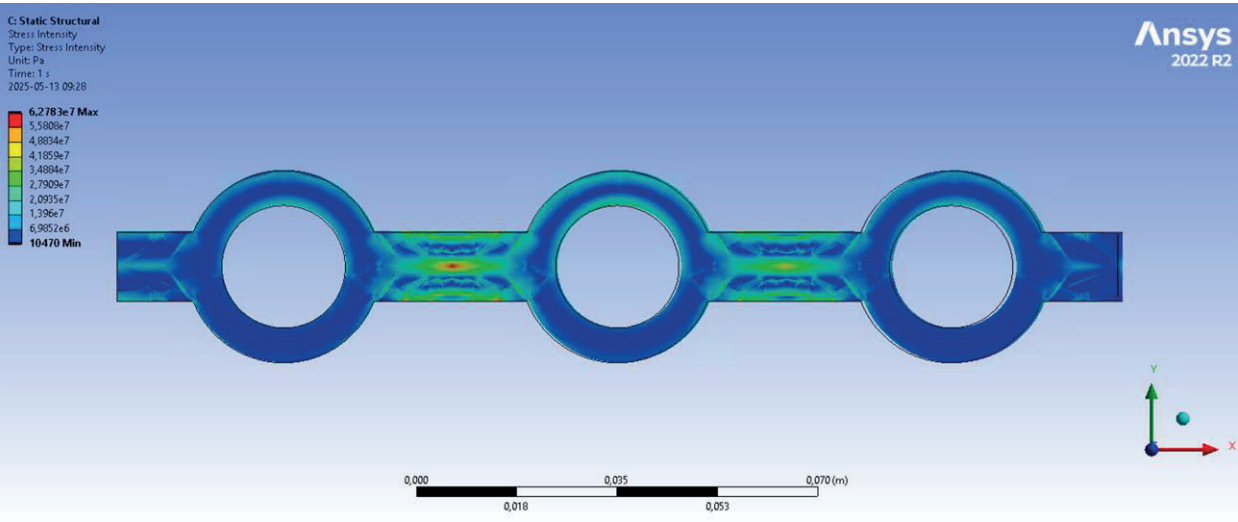


Fig. 16. The thermal stress distribution in the cross-section of the analysed system with wider tube spacing at a distance of 40 m from the tube inlet.

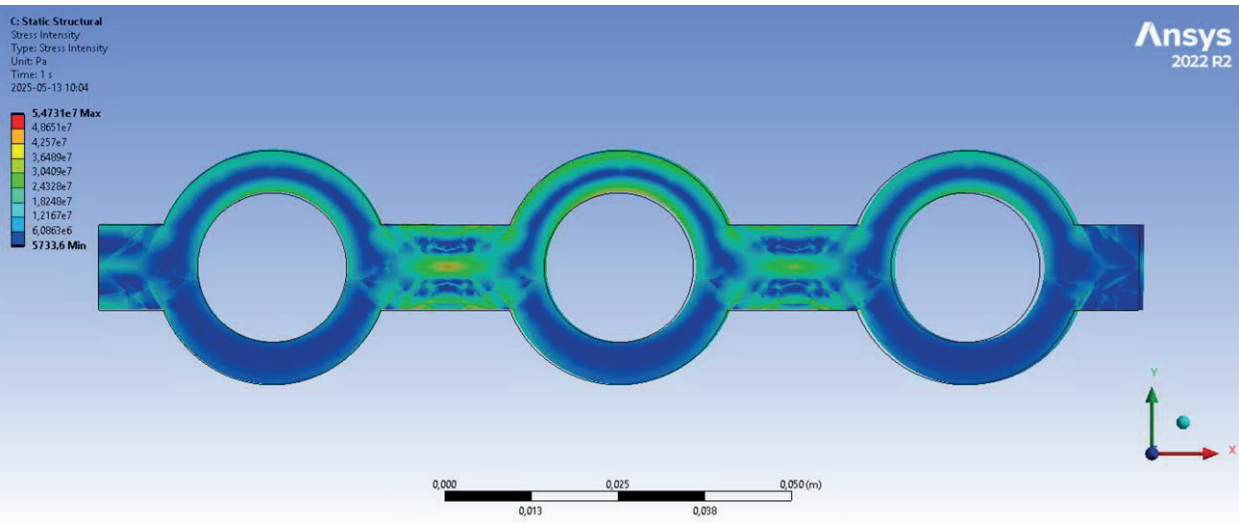


Fig. 14. The thermal stress distribution in the cross-section of the analysed system with "standard" tube spacing at a distance of 80 m from the tube inlet.

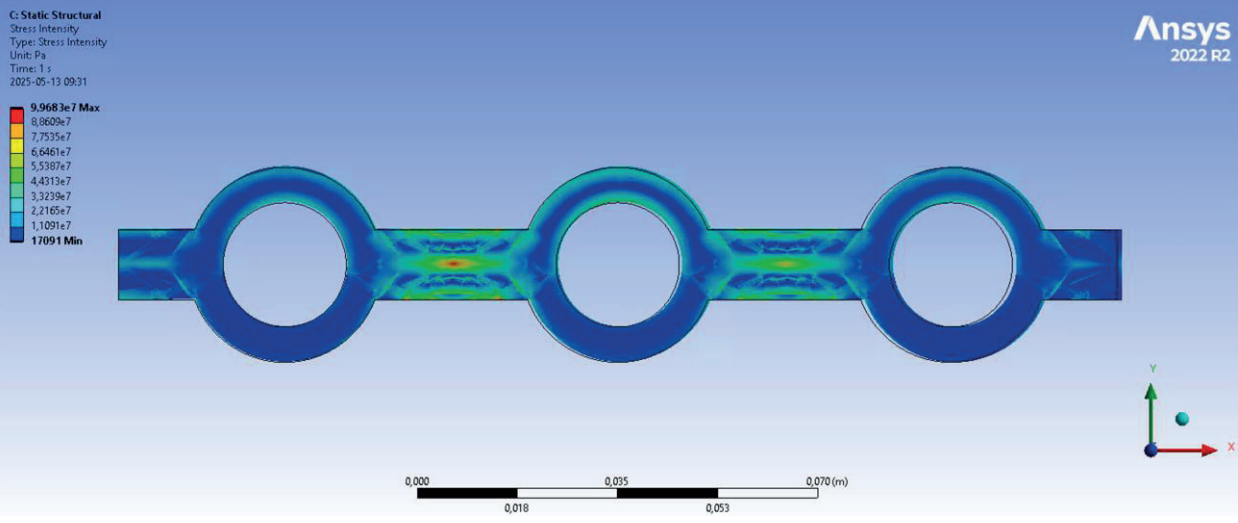


Fig. 17. The thermal stress distribution in the cross-section of the analysed system with wider tube spacing at a distance of 80 m from the tube inlet.

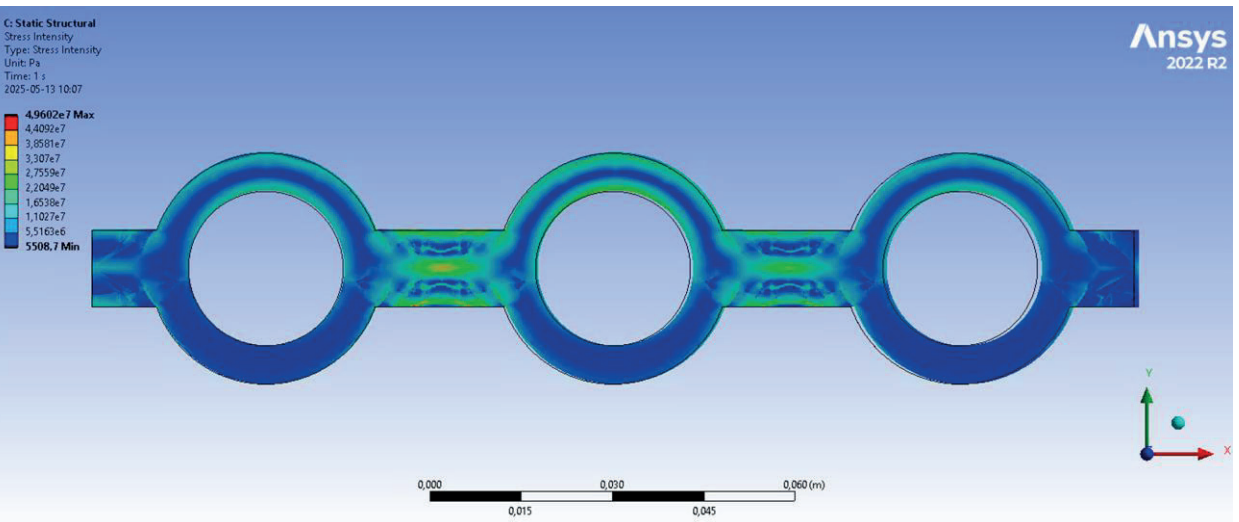


Fig. 15. The thermal stress distribution in the cross-section of the analysed system with "standard" tube spacing at a distance of 120 m from the tube inlet.

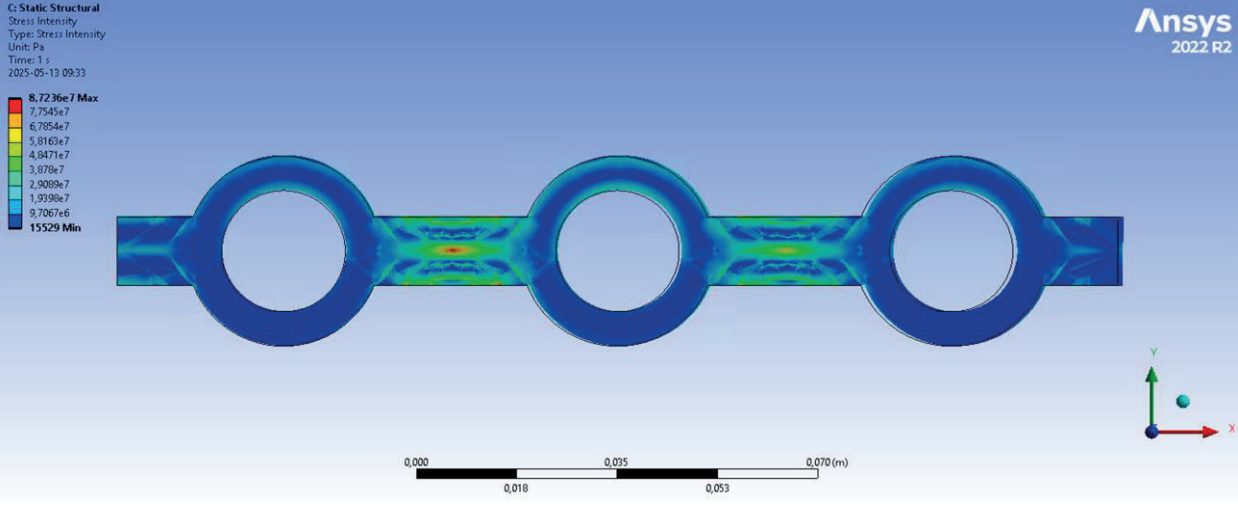


Fig. 18. The thermal stress distribution in the cross-section of the analysed system with wider tube spacing at a distance of 120 m from the tube inlet.

Due to the analysis of a small section of the combustion chamber wall, correctly representing the conditions occurring at the edges of the analysed area was a significant challenge. As it was mentioned before, to approximate the actual simulation conditions, a plane strain state was assumed, meaning that there was no deformation in the Z-axis direction perpendicular to the analysed cross-section. One side of the model was fixed to a point, and the other side was blocked from moving relative to the Y-axis.

To compare the stresses occurring in the cross-sections of the analysed system depending on the length of the fins connect-

ing the tubes, thermal stress values were determined at the centres of the cross-sections of both fins; they are presented in Table 1. Differences between the stress values for each fin in the same cross-section result from the previously described method of mounting the analysed model.

To facilitate the search for relationships between the obtained stress and temperature values at the analysed points of the system, Table 2 presents the temperature values at points marked with numbers "36" and "71", located at the centres of the fins, for each of the analysed cross-sections of the tubes and fins connecting them.

Table 1. The thermal stress values calculated for the analysed cross-sections of waterwall tubes connected by fins.

Distance from the tube inlet, [m]	Case 1 – „standard” tube spacing		Case 2 – wider tube spacing	
	Thermal stress value in point „36”, [MPa]	Thermal stress value in point „71”, [MPa]	Thermal stress value in point „36”, [MPa]	Thermal stress value in point „71”, [MPa]
40	20.69	25.72	44.59	57.34
80	32.07	40.33	70.12	89.23
120	29.19	36.62	62.38	80.16

Table 2. The temperature values calculated for the analysed cross-sections of waterwall tubes connected by fins.

Distance from the tube inlet, [m]	Case 1 – „standard” tube spacing		Case 2 – wider tube spacing	
	Temperature value in point „36”, [°C]	Temperature value in point „71”, [°C]	Temperature value in point „36”, [°C]	Temperature value in point „71”, [°C]
40	372.52	367.47	400.97	390.83
80	439.45	431.52	482.90	466.79
120	443.07	436.19	483.81	469.48

6. Conclusions

A new model for determining the distribution of thermal stresses in fragments of combustion chamber walls for the considered cases of tube assembly with different tube spacing was presented. The proposed algorithm allows calculating thermodynamic parameters of the working fluid and wall material for each control volume of the analysed model in each time step. By using this model, it is possible to calculate temperature values in 36 characteristic points of the cross-section of each tube connected with fins. The obtained temperature values are close to the temperature distributions obtained in other scientific papers and by using CFD software.

The novelty of this paper is the comparison of thermal stress values for systems with identical input parameters of the working medium and identical values of the specific heat flux incident on the combustion chamber walls, but with different spacing widths of the waterwall tubes. This allows for an analysis of the effect of the fin width between the tubes on the temperatures and thermal stresses obtained at the centres of fins.

The obtained temperature and thermal stress distributions indicate that as the fin width between the tubes increases, so do the temperatures and thermal stresses. The analysis allows us to conclude that in areas with high heat flux to the combustion chamber walls, it would be beneficial to reduce the spacing of the waterwall tubes to reduce thermal stresses and minimise the risk of damage resulting from exceeding the permissible operating temperatures for the metals from which the tubes and fins are made.

[8] Grądziel, S., & Majewski, K. (2019). Experimental determination of the friction factor in a tube with internal helical ribs. *Energies*, 12(2), 257. doi: 10.3390/en12020257

[9] Yamamoto, K., Suganuma, H., Domoto, K., Yamasaki, Y., Kanemaki, Y., & Nakaharai, H. (2013). Design Technology for Supercritical Sliding Pressure Operation Vertical Water Wall Boilers-First Report : History of Practical Application and Introduction of Enhanced Rifled Tube. *Mitsubishi Heavy Industries Technical Review*, 50(3).

[10] Gaćeša, B., Maneski, T.D., Milosevic-Mitic, V.O., Nestorovic, M.S., & Petrovic, A. (2014). Influence of furnace tube shape on thermal strain of fire-tube boilers. *Thermal Science*, 18(1), 39–47. doi: 10.2298/TSCI130317172G

[11] Taler, J., Duda, P., Węglowski, B., Zima, W., Grądziel, S., Sobota, T., & Taler, D. (2009). Identification of local heat flux to membrane water-walls in steam boilers. *Fuel*, 88, 305–311. doi: 10.1016/j.fuel.2008.08.011

[12] Grądziel, S. (2013). Determination of the heat load on the walls of the furnace chamber of the OP-210 boiler. In *Energy machines and devices* (pp. 71–90). Cracow, Poland (in Polish).

[13] Grądziel, S. (2015). Heat load determination of boiler furnace chamber. *Archivum Combustionis*, 35(2), 75–88.

[14] Zima, W., Grądziel, S., & Cebula, A. (2010). Modelling of heat and flow phenomena occurring in water-wall tubes of boilers for supercritical steam parameters. *Archives of Thermodynamics*, 31(3), 19–36. doi: 10.2478/v10173-010-0012-y

[15] Taler, J., Taler, D., & Kowal, A. (2011). Measurements of absorbed heat flux and water-side heat transfer coefficient in water wall tubes. *Archives of Thermodynamics*, 32(3), 77–88. doi: 10.2478/v10173-011-0004-6

[16] Zima, W., Nowak-Ocłoń, M., & Ocłoń, P. (2015). Simulation of fluid heating in combustion chamber water-walls of boilers for supercritical steam parameters. *Energy*, 92, 117–127. doi: 10.1016/j.energy.2015.02.111

[17] Zima, W., & Nowak-Ocłoń, M. (2016). A new 1D/3D model of conjugate heat transfer in waterwall tubes of power boiler combustion chamber. *Procedia Engineering*, 157, 200–206. doi: 10.1016/j.proeng.2016.08.357

[18] Zima, W., Nowak-Ocłoń, M., & Ocłoń, P. (2018). Novel online simulation-ready models of conjugate heat transfer in combustion chamber waterwall tubes of super-critical power boilers. *Energy*, 148, 809–823. doi: 10.1016/j.energy.2018.01.178

[19] Zima, W. (2019). Analysis of temperature drops on the wall thickness of a supercritical boiler water-wall tube. *Thermal Science*, 23(4), 1091–1100. doi: 10.2298/TSCI19S4091Z

[20] Zima, W., Grądziel, S., Cebula, A., Rerak, M., & Kozak-Jagięła, E. (2024). Simulation of the supercritical boiler evaporator operation under conditions of a rapid power increase. *Applied Thermal Engineering*, 243, 122592. doi: 10.1016/j.applthermaleng.2024.122592

[21] Trojan, M., & Taler, J. (2013). Effect of scale deposits on the internal surfaces of the tubes on the superheater operation. *Archives of Thermodynamics*, 34(4), 73–91. doi: 10.2478/aoter-2013-0030

[22] Grądziel, S., & Majewski, K. (2016). Simulation of heat transfer in combustion chamber waterwall tubes of supercritical steam boilers. *Chemical and Process Engineering*, 37(2), 199–213. doi: 10.1515/cpe-2016-0017

[23] Majdak, M., & Grądziel, S. (2020). Influence of thermal and flow conditions on the thermal stresses distribution in the evaporator tubes. *Energy*, 209, 118416. doi: 10.1016/j.energy.2020.118416

[24] Majdak, M., Grądziel, S., Zima, W., Cebula, A., Rerak, M., & Kozak-Jagięła, E. (2024). Numerical and experimental analysis of thermal and flow operating conditions of waterwall tubes connected by fins. *Archives of Thermodynamics*, 44(4), 231–252. doi: 10.24425/ather.2023.149719

[25] Kitoh, K., Koshizuka, S., & Oka, Y. (2001). Refinement of transient criteria and safety analysis for a high temperature reactor cooled by supercritical water. *Nuclear Technology*, 135, 252–264. doi: 10.13182/NT01-A3220



Co-published by
Institute of Fluid-Flow Machinery
Polish Academy of Sciences
Committee on Thermodynamics and Combustion
Polish Academy of Sciences

Copyright©2026 by the Authors under licence CC BY-NC-ND 4.0

<http://www.imp.gda.pl/archives-of-thermodynamics/>



Investigation of Heat Transfer and Flow Characteristics in Solar Air Heaters with a Spherical Turbulator Using Machine Learning Models

Vijay Singh Bisht^a, Desh Bandhu Singh^a, Pushpendra Kumar^b, Prabhakar Bhandari^{b,c*},
Rupesh Kumar Tipu^d, Sandeep Singh^e, Ankur Singh Bist^f

^aDepartment of Mechanical Engineering, Graphic Era (Deemed to be) University, Dehradun 248002, Uttarakhand, India

^bDepartment of Mechanical Engineering, K.R. Mangalam University, Gurugram 122103, Haryana, India

^cCentre of Excellence-Cyber Security, School of Engineering and Technology, K.R. Mangalam University, Gurugram 122103, Haryana, India

^dDepartment of Civil Engineering, School of Engineering and Technology, K.R. Mangalam University, Gurugram 122103, Haryana, India

^eO.P. Jindal Global University, Sonapat 131001, Haryana, India

^fCSE, Graphic Era Hill University, Bhimtal Campus, Uttarakhand, India

*Corresponding author email: prabhakar.bhandari40@gmail.com

Received: 25.12.2025; revised: 21.03.2026; accepted: 22.04.2026

Abstract

Solar air heaters are attractive for low-carbon thermal applications, but their performance is constrained by weak convective heat transfer in the near-wall region. In this work, a single-pass rectangular solar air heater duct equipped with spherical turbulators is investigated numerically and then accelerated using machine-learning surrogate models for rapid prediction of thermal–hydraulic responses. Five turbulator arrangements (V-, M-, W-shaped, inclined, and arc-shaped) were evaluated at three discrete spacing levels (pitch ratio $P/D = 3, 6, 9$) with a constant sphere diameter of 25 mm over $Re = 3500–23\,500$, consistent throughout all the simulations. The computational fluid dynamics model employed a constant heat flux of 1000 W/m^2 and standard pressure–velocity coupling/discretisation practices, and was validated against established previous work. A leakage-safe machine-learning dataset (675 samples, 36 variables) was constructed from computational fluid dynamics outputs and physics-informed engineered features; models were trained using geometry-grouped cross-validation to ensure generalisation across turbulator arrangements and spacing levels. Among candidate regressors, histogram gradient boosting provided the best Nu surrogate ($OOF\ RMSE = 3.299$, $R^2 = 0.9908$, $MAPE = 2.44\%$), while ridge regression yielded the most accurate friction factor surrogate ($OOF\ RMSE = 4.70 \times 10^{-4}$, $R^2 = 0.9886$, $MAPE = 1.12\%$). The combined computational fluid dynamics and machine-learning framework enables fast, reliable evaluation of solar air heater thermo-hydraulic performance for configuration screening and optimisation within the validated operating envelope.

Keywords: Solar air heater; Spherical turbulator; Thermo-hydraulic performance; Surrogate modelling; Gradient boosting; Ridge regression

Vol. 47(2026), No. 2, 99–118; doi: 10.24425/ather.2026.158677

Cite this manuscript as: Bisht, V.S., Singh, D.B., Bhandari, P., Kumar, P., Tipu, R.K., Singh, S., & Bist, A.S. (2026). Investigation of Heat Transfer and Flow Characteristics in Solar Air Heaters with a Spherical Turbulator Using Machine Learning Models. *Archives of Thermodynamics*, 47(2), 99–118.

1. Introduction

In recent years, the combination of rapid population expansion and intensified industrial activity has significantly increased

global energy consumption. This rising demand continues to depend largely on traditional fuels – coal, petroleum and natural gas – which are not only diminishing quickly but also causing substantial environmental harm. Consequently, the search for

Nomenclature

A – cross section area, m²
 C_p – specific heat of air, J/(kg·K)
 D – diameter, m
 F – friction factor
 h – heat transfer coefficient, W/(m²·K)
 k – turbulent kinetic energy, J/kg
 k_a – thermal conductivity, W/(m·K)
 M – mass flow rate of air, kg/s
Nu – Nusselt number
 p – pressure, kPa
 Δp – pressure drop, N/m²
 P – pitch, perimeter, m
 P/D – relative roughness pitch
Pr – Prandtl number
 q – heat flux, W/m²
 R^2 – coefficient of determination or R -squared
Re – Reynolds number
 t – thickness, m
 T – temperature, K
 U_L – head loss coefficients, W/(m²·K)
 u, v, w – velocity components in x, y and z direction, m/s
 V – mean flow velocity, m/s
 W – channel width, mm
 x, y, z – coordinates, m

Greek symbols

ε – turbulence-specific dissipation rate, m²/s³
 μ – dynamic viscosity of air, N s/m²
 ν, ν_T – kinematic viscosity, eddy viscosity, m²/s

ρ – density of air, kg/m³

Subscripts and Superscripts

in – inlet
 out – outlet
 r – roughened
 s – smooth

Abbreviations and Acronyms

ANN – artificial neural networks
ASHRAE– American Society of Heating, Refrigerating and Air Conditioning Engineers
CFD – computational fluid dynamics
CNN – convolutional neural network
ML – machine-learning
EDA – exploratory data analysis
GA – genetic algorithms
HistGB – histogram-based gradient boosting
 MAE – mean absolute error
 $MAPE$ – mean absolute percentage error
MLP – multi-layer perceptron
OOF – out-of-fold
PSO – particle swarm optimisation
RF – random forest
 $RMSE$ – root mean squared error
RNG – re-normalisation group
SAH – solar air heater
SIMPLE – semi-implicit method for pressure-linked equation
SST – shear stress transport
ST – spherical turbulators
SVM – support vector machine
SVR – support vector regression
THPP – thermo-hydraulic performance parameter

clean and sustainable energy options has become critical. Renewable resources such as solar, biomass and hydropower have gained attention as viable alternatives, with solar energy emerging as particularly important because of its wide availability, especially in high-insolation regions like India. Thus, utilising solar energy has become essential in the worldwide move toward environmentally responsible power production. Solar energy serves numerous purposes, including water heating [1], space cooling [2], cooking [3], and purification [4]. Among these applications, solar air heating is especially significant in colder regions. A solar air heater (SAH) is a cost-effective and environmentally friendly system that absorbs solar radiation to enable heat transfer [5].

Its straightforward structure, minimal maintenance needs, and freedom from fossil fuels make it suitable for both domestic and industrial heating. Despite these advantages, SAHs face a key drawback: their thermal efficiency remains low because of insufficient heat transfer between the absorber plate and the air passing over it [6].

SAH systems typically consist of an absorber plate that collects incoming solar radiation, a transparent cover that reduces thermal losses, and an insulated enclosure that helps retain the captured heat. Within a SAH, the air flowing close to the absorber plate encounters a turbulent boundary layer, creating a thin viscous sublayer where heat transfer is limited due to reduced air velocity and low thermal conductivity. As a result, the

overall thermal performance of the system remains restricted under conventional operating conditions. To address this limitation, a variety of passive enhancement techniques have been introduced, such as adding surface roughness features, integrating turbulence-generating components, and altering duct configurations. These methods are designed to increase airflow turbulence and consequently improve heat transfer. Among the different enhancement strategies, artificial roughness has become one of the most commonly used and highly effective solutions for boosting the thermal efficiency of SAHs [7]. This method involves placing features such as ribs, dimples, or baffles on the absorber surface to disrupt the laminar sublayer and strengthen heat transfer [8].

Artificial roughness provides a passive, energy-free means of improving thermal output; it is simple to apply, economical, and substantially enhances the heat-exchange process. However, it can also lead to higher frictional losses, making careful design optimisation necessary to achieve an appropriate balance between increased heat transfer and pressure drop. Numerous studies have investigated the ideal configuration of roughness elements in SAHs through both computational analyses and experimental studies.

Chaube et al. [9] carried out a two-dimensional numerical analysis using the shear stress transport (SST) $k-\omega$ turbulence model to examine the effects of various rib geometries on heat transfer and airflow characteristics in a single-pass solar air

heater. Among the rib configurations tested – rectangular, chamfered, square, circular and semi-circular – the chamfered design produced the greatest heat transfer enhancement, whereas rectangular ribs delivered the highest thermo-hydraulic efficiency by offering an optimal balance between thermal improvement and pressure drop. This study highlights the critical role of rib shape in boosting the performance of solar air heaters. In another investigation, Kumar and Saini [10] performed numerical simulations on a SAH equipped with arc-shaped ribs over a Reynolds number range of 6000–18 000 and relative roughness heights between 0.0299 and 0.0426. Their work demonstrated that, among multiple turbulence models assessed, the re-normalisation group (RNG) $k-\varepsilon$ model provided the most reliable predictions. Yadav and Bhagoria [11] numerically evaluated the influence of circular transverse wire ribs on SAH performance and found that the ribbed arrangement increased the Nusselt number by up to 2.6 times, achieving a maximum thermo-hydraulic performance factor of 1.8. Al-Chlaihawi et al. [12] also carried out a numerical study in which they employed a combination of transverse trapezoidal ribs and chamfered grooves to enhance thermal behaviour in a SAH duct. Their analysis explored parameters such as relative roughness pitch, relative roughness height and Reynolds number, leading to optimised values of improved performance. Additionally, they proposed new correlations for predicting both the Nusselt number and friction factor.

Manjunath et al. [13] investigated the use of spherical turbulence promoters and reported as much as a 2.45-fold increase in heat transfer, accompanied by a moderate rise in friction factor and a thermo-hydraulic performance factor (THPF) of 1.74, demonstrating their effectiveness in enhancing system performance. Maithani et al. [14] employed wavy delta-winglet vortex generators in rectangular ducts, producing swirling airflow that improved heat transfer by 30–40% compared to smooth ducts while maintaining the performance evaluation criterion (PEC) greater than 1, despite the associated rise in pressure drop. In a separate study, Manjunath et al. [15] examined absorber plates incorporating sinusoidal wave patterns and found that an optimised configuration – with a wavelength of 1.0 and an aspect ratio of 1.5 – raised the thermal efficiency by 12.5%, although an increased pumping power was required. Researchers have also explored a wide range of fin configurations to enhance SAH performance, including pin fins [16], stepped cylindrical turbulence generators [17], helicoidal spring-shaped fins [18], and others.

Drawing inspiration from geometric shapes of the English alphabet letters, numerous researchers have investigated how alphabet-shaped ribs affect the heat transfer and flow characteristics of solar air heaters (SAHs). Taslim et al. [19] and Olsson and Sunden [20] analysed heat transfer in square channels and found that V-shaped ribs performed better than both inclined and transverse ribs in enhancing thermal output. Momin et al. [21] experimentally evaluated the effects of rib height and attack angle on heat transfer and flow behaviour in SAHs equipped with V-shaped wire ribs, confirming that V-shaped ribs achieve higher thermal efficiency than inclined rib configurations under comparable conditions. Lanjewar et al. [22] carried out an experimental study on SAHs incorporating W-sha-

ped wire rib roughness and reported that W-shaped ribs surpassed V-shaped ribs in overall thermo-hydraulic performance. Ghildyal et al. [23] compared D-shaped, reverse D-shaped and U-shaped turbulators, determining that U-shaped designs achieved the greatest Nusselt number and overall thermo-hydraulic performance, while the D-shaped turbulator exhibited the highest friction factor. In a subsequent study, Ghildyal et al. [24] showed that W-contoured turbulators delivered the best thermal efficiency when benchmarked against tapered and reverse-tapered variants over Reynolds numbers ranging from 4000 to 18 000. Bohra et al. [25] demonstrated that Z-shaped baffles mounted on the absorber plate significantly improved SAH performance, with the highest efficiency obtained at a blockage ratio of 0.3, a pitch ratio of 1.5, and a baffle angle of 45° for Reynolds numbers between 5000 and 10 000. Similarly, Chaudhari et al. [26] identified that inverted L-shaped ribs with a roughness pitch of 17.86 achieved maximum exergy efficiency at a heat flux of 1000 W/m². Kumar et al. [27] analysed a hybrid roughness configuration combining S-shaped ribs and protrusions to further improve heat transfer.

Alhilali et al. [28] employed I-shaped ribs in a two-dimensional numerical study and reported that they enhanced the Nusselt number by 2.498–2.978 times and increased the friction factor by 2.812–4.393 times. Extending these concepts to parallel-plate heat exchangers, Chtourou et al. [29] utilised Y-shaped and C-shaped obstacles, observing a maximum thermo-hydraulic performance parameter (THPP) of approximately 1.44 for Y-shaped ribs and around 2.6 for C-shaped ribs. In a follow-up study, Chtourou et al. [30] refined the Y-shaped geometry into classical, fragmental and fractal variants and assessed their behaviour in rectangular and triangular arrangements. Among all designs, the fragmental Y configuration arranged triangularly yielded the highest thermo-hydraulic performance. Beyond these, researchers have also explored novel rib geometries such as plus (+)-shaped turbulators [31] and hook-shaped ribs [32].

Beyond rib geometry, the orientation or layout of ribs on the absorber plate significantly influences the underlying flow behaviour. When a transverse rib is placed in the passage, it generates a pair of vortices on its upstream and downstream sides. These vortices tend to remain relatively stationary compared with the bulk airflow, causing elevated temperatures within the vortex regions and near the wall, which ultimately decreases the local heat transfer rate. In contrast, an inclined rib allows the vortical structures to convect along the rib’s length – fluid enters near the leading edge and exits toward the trailing edge, reattaching to the primary stream. This motion introduces secondary flow patterns that sweep across the channel width, producing marked spatial differences in heat transfer. Consequently, the rib's inclination angle becomes just as important as its height and pitch in enhancing thermal transport performance [33]. Several rib orientations have been examined in the literature, including inclined [34], transverse [35], V-shaped [36], and others.

Artificial neural networks (ANNs) have become useful tools in SAH research because they can model nonlinear relationships among operating conditions, geometry, heat transfer, friction loss and thermal performance. Ghritlahre et al. [37] used relevant input parameters to predict the performance of a solar air

heater through an ANN model. Bhattacharyya et al. [38] developed an ANN-based model for a solar-air-heater tube and predicted the Nusselt number and thermo-hydraulic efficiency with high accuracy. Ghritlahre and Prasad [39] also used MLP, GRNN and RBF neural-network models to predict the exergetic performance of solar air heaters, showing the suitability of ANN-based models for solar thermal performance estimation.

Recent studies have extended this direction by combining CFD-generated data with machine-learning models. Alatawi [40] integrated CFD and ML for solar air heaters with distinct rib configurations and reported that a CNN model outperformed random forest regression and support vector regression in predicting heat transfer coefficients. Review evidence also shows that intelligent methods are increasingly used for solar heater prediction and optimisation [41]. However, many earlier CFD-ML studies still focus mainly on prediction accuracy and do not fully test geometry-wise generalisation under leakage-safe grouped validation. The present study addresses this limitation by using a geometry-grouped surrogate framework for spherical-turbulator solar air heaters.

Despite these advances, three important gaps remain in the current CFD-ML literature on solar air heaters. First, most prior studies emphasise prediction accuracy for one or more performance parameters, but do not evaluate whether the surrogate can generalise to previously unseen geometry groups under leakage-safe grouped validation. Second, many studies focus on conventional ribbed or finned geometries, whereas geometry-specific surrogate modelling for spherical-turbulator layouts and their arrangement-dependent thermo-hydraulic ranking remains limited. Third, several CFD-ML studies report fast prediction, but do not connect surrogate performance to a clear design outcome through frozen-holdout testing, uncertainty bounds, and thermo-hydraulic selection of the best configuration. The present work addresses these gaps by combining validated CFD simulations with a grouped and leakage-safe surrogate framework for five spherical-turbulator arrangements, followed by frozen-holdout evaluation, bootstrap-based uncertainty quantification, and thermo-hydraulic performance parameter (THPP)-based design ranking.

The objectives of the present work are fivefold:

- 1) to construct a supervised CFD-derived dataset for solar air heaters equipped with spherical turbulators, using operating and geometric descriptors, including the Reynolds number, configuration, and pitch ratio, to predict the Nusselt number (Nu) and friction factor (f);
- 2) to establish a transparent and reproducible data-preparation workflow with explicit feature documentation and leakage control for tabular thermo-fluid surrogate modelling;
- 3) to design a geometry-aware evaluation framework based on configuration $\times$ pitch grouping, grouped cross-validation, and a frozen geometry-grouped holdout test so that generalisation is assessed on unseen layouts rather than near-duplicate cases;
- 4) to benchmark multiple regression surrogates and identify the most reliable predictor for each target, together with bootstrap-based uncertainty estimates for design-screening reliability;
- 5) to translate the CFD-ML workflow into a practical thermo-hydraulic design outcome by ranking the investigated spherical-turbulator configurations using THPP and identifying the best-performing arrangement within the investigated design space.

In this way, the contribution of the study lies not only in replacing repeated CFD runs with fast surrogates, but in providing a geometry-aware, uncertainty-supported and design-oriented CFD-ML framework for configuration screening of spherical-turbulator solar air heaters.

2. Solar air heater geometry and different configurations

The present work examines a SAH equipped with spherical turbulators through both analytical formulation and numerical simulation. A schematic representation of the system is shown in Fig. 1, where spherical elements are positioned beneath the absorber plate to intensify heat transfer.

The duct has a rectangular cross-section with a width of 0.15 m, a height of 0.03 m and an overall length of 1.85 m, con-

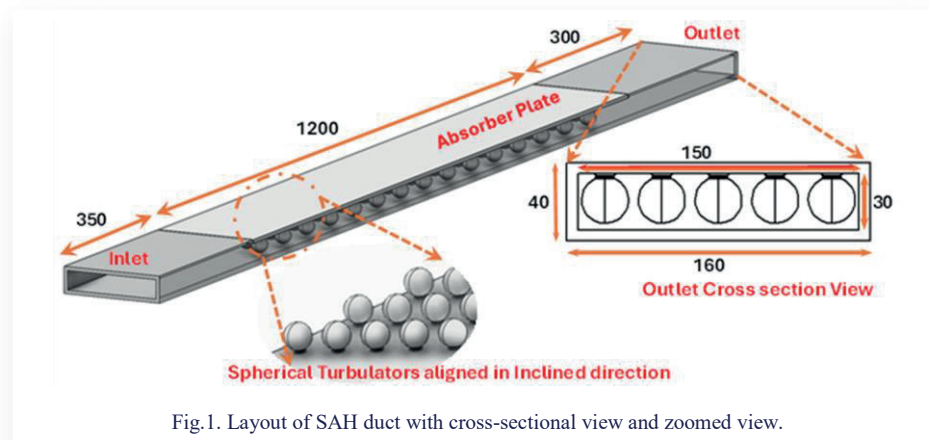


Fig. 1. Layout of SAH duct with cross-sectional view and zoomed view.

sisting of a 0.35 m inlet, a 1.20 m heating region and a 0.30 m outlet. The design of SAH follows the recommendations specified in American Society of Heating, Refrigerating and Air Con-

ditioning Engineers (ASHRAE) Standard 93–77 [13,42].

The absorber plate extends across the active heating zone and captures solar radiation before conveying thermal energy to

the air flowing beneath it. Spherical turbulators with a diameter of 25 mm are arranged in uniformly spaced rows along the underside of the plate. Their placement is intended to disrupt the boundary layer, induce turbulence, and thereby elevate the convective heat transfer coefficient. The cross-sectional view illustrates the positioning of the spheres within the flow passage, while the magnified view depicts their specific layout pattern. Each configuration of spherical turbulators is designed to enhance heat transfer effectiveness while controlling pressure losses to achieve improved thermo-hydraulic performance.

This study investigates how various orientations of spherical turbulators influence the thermal and flow behaviour of SAHs.

To facilitate this analysis, five separate configurations were formulated, each featuring a distinct pattern of turbulators positioned beneath the absorber plate. The examined arrangements include V-shaped, M-shaped, W-shaped, inclined and arc-shaped layouts, all intentionally designed to modify airflow characteristics and intensify turbulence within the duct. A top-view representation of these configurations is shown in Fig. 2, illustrating the placement and directional alignment of the spherical turbulators across the absorber plate region for each design. This comparative framework allows for a systematic assessment of the impact that different turbulator patterns have on heat transfer augmentation and overall thermo-hydraulic performance.

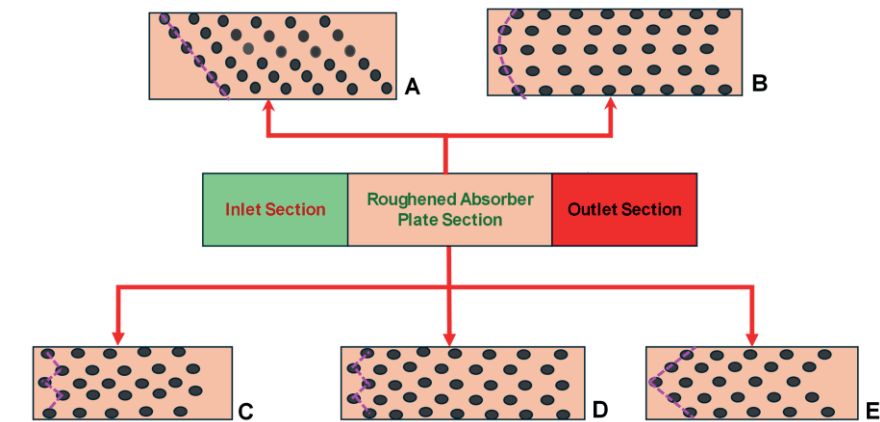


Fig. 2. Various arrangement of spherical turbulators in SAHs (a) inclined arrangement, (b) arc-shaped arrangement, (c) M-shaped arrangement, (d) W-shaped arrangement (e) V-shaped arrangement.

In every design considered, the spherical turbulators were kept at a constant diameter of 25 mm to maintain uniformity in geometry. This specific size was chosen because earlier findings by Manjunath et al. [13] indicated that a 25 mm sphere provides the highest thermal and effective efficiency. For all configurations, the angle of attack was maintained at 60°. Additionally, each turbulator pattern was evaluated at three different relative roughness pitch values – 3, 6 and 9 – to assess how variations in spacing and arrangement affect the overall thermo-hydraulic performance of the system.

3. CFD and ML modelling

3.1. CFD numerical model

To perform the numerical analysis of SAHs, several foundational assumptions were introduced to streamline the computational framework and maintain reliable simulation results. These assumptions include:

- The working fluid – air – is treated as a Newtonian, continuous medium with consistent flow characteristics across the entire domain;
- The airflow is considered incompressible, meaning density variations during motion are assumed to be negligible;
- A no-slip condition is applied to all solid boundaries, ensuring the fluid velocity at the wall is equal to the wall velocity;

- The thermo-physical properties of air, such as density, viscosity, thermal conductivity and specific heat, are assumed to remain constant throughout the simulation;
- Heat transfer is modelled through conduction and convection only, while thermal radiation effects are regarded as minimal and therefore omitted from the analysis.

3.1.1. Conservation equations and boundary conditions

Any thermodynamic system operating over a specified duration must satisfy the fundamental conservation principles of mass, momentum and energy. The Cartesian form of these governing equations – continuity, momentum, and energy – is expressed as follows [43,44]:

- mass conservation:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0, \quad (1)$$

- momentum conservation:

– (M_x):

$$\frac{\partial u}{\partial t} + \frac{\partial(uu)}{\partial x} + \frac{\partial(vu)}{\partial y} + \frac{\partial(wu)}{\partial z} = \frac{\partial}{\partial x} \left[(\nu + \nu_T) \frac{\partial u}{\partial x} \right] + \frac{\partial}{\partial y} \left[(\nu + \nu_T) \frac{\partial u}{\partial y} \right] + \frac{\partial}{\partial z} \left[(\nu + \nu_T) \frac{\partial u}{\partial z} \right] + \left(-\frac{1}{\rho} \frac{\partial p}{\partial x} + s'_u \right), \quad (2)$$

– (M_y):

$$\frac{\partial v}{\partial t} + \frac{\partial(uv)}{\partial x} + \frac{\partial(vv)}{\partial y} + \frac{\partial(wv)}{\partial z} = \frac{\partial}{\partial x} \left[(\nu + \nu_T) \frac{\partial v}{\partial x} \right] + \frac{\partial}{\partial y} \left[(\nu + \nu_T) \frac{\partial v}{\partial y} \right] + \frac{\partial}{\partial z} \left[(\nu + \nu_T) \frac{\partial v}{\partial z} \right] + \left(-\frac{1}{\rho} \frac{\partial p}{\partial y} + s'_v \right), \quad (3)$$

– (M_z):

$$\frac{\partial w}{\partial t} + \frac{\partial(uw)}{\partial x} + \frac{\partial(vw)}{\partial y} + \frac{\partial(ww)}{\partial z} = \frac{\partial}{\partial x} \left[(\nu + \nu_T) \frac{\partial w}{\partial x} \right] + \frac{\partial}{\partial y} \left[(\nu + \nu_T) \frac{\partial w}{\partial y} \right] + \frac{\partial}{\partial z} \left[(\nu + \nu_T) \frac{\partial w}{\partial z} \right] + \left(-\frac{1}{\rho} \frac{\partial p}{\partial z} + s'_w \right), \quad (4)$$

• energy equation:

$$\frac{\partial T}{\partial t} + \frac{\partial(uT)}{\partial x} + \frac{\partial(vT)}{\partial y} + \frac{\partial(wT)}{\partial z} = \frac{\partial}{\partial x} \left[\left(\frac{\nu}{Pr} + \frac{\nu_T}{Pr_T} \right) \frac{\partial T}{\partial x} \right] + \frac{\partial}{\partial y} \left[\left(\frac{\nu}{Pr} + \frac{\nu_T}{Pr_T} \right) \frac{\partial T}{\partial y} \right] + \frac{\partial}{\partial z} \left[\left(\frac{\nu}{Pr} + \frac{\nu_T}{Pr_T} \right) \frac{\partial T}{\partial z} \right] + S_T, \quad (5)$$

where s'_u , s'_v , s'_w and S_T are additional source terms in the momentum equation in the x -direction, y -direction, z -direction, and in the energy equation, respectively.

• turbulence equations:

– (k):

$$\frac{\partial k}{\partial t} + \frac{\partial(uk)}{\partial x} + \frac{\partial(vk)}{\partial y} + \frac{\partial(wk)}{\partial z} = \frac{\partial}{\partial x} \left[\left(\frac{\nu_T}{\sigma_k} \frac{\partial k}{\partial x} \right) \right] + \frac{\partial}{\partial y} \left[\left(\frac{\nu_T}{\sigma_k} \frac{\partial k}{\partial y} \right) \right] + \frac{\partial}{\partial z} \left[\left(\frac{\nu_T}{\sigma_k} \frac{\partial k}{\partial z} \right) \right] + (P - D), \quad (6)$$

– (ε):

$$\frac{\partial \varepsilon}{\partial t} + \frac{\partial(u\varepsilon)}{\partial x} + \frac{\partial(v\varepsilon)}{\partial y} + \frac{\partial(w\varepsilon)}{\partial z} = \frac{\partial}{\partial x} \left[\left(\frac{\nu_T}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x} \right) \right] + \frac{\partial}{\partial y} \left[\left(\frac{\nu_T}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial y} \right) \right] + \frac{\partial}{\partial z} \left[\left(\frac{\nu_T}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial z} \right) \right] + \left(\frac{\varepsilon}{k} (C_{\varepsilon 1} P - C_{\varepsilon 2} D) \right), \quad (7)$$

where

$$P = 2\nu_T \left[\left(\frac{\partial u}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial y} \right)^2 + \left(\frac{\partial w}{\partial z} \right)^2 \right] + \nu_T \left[\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right)^2 + \left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \right)^2 \right], \quad (8)$$

$$D = \varepsilon,$$

where $C_{\varepsilon 1}$ and $C_{\varepsilon 2}$ are the model constants with values of 1.44 and 1.92, respectively. The turbulence kinetic energy magnitude is indicated by (k), turbulence-specific dissipation rate is shown by (ε), and the eddy viscosity is indicated by (ν_T).

The k - ε turbulence model is selected for the CFD evaluation of the solar air heater because it delivers reliable, experimentally validated predictions of turbulent flow and heat transfer in ducts with surface roughness, while also offering good computational efficiency. Numerous numerical investigations and review studies on solar air heaters have shown that the re-normalisation group (RNG) variant of the k - ε model closely matches experimental findings for absorber plates equipped with roughness elements [23–25,27,28,31,32,38].

A constant heat flux of 1000 W/m² was applied to the absorber plate. The inlet flow conditions were defined for Reynolds numbers ranging from 3500 to 23 500 [13,15,16]. In this analysis, velocity inlet and pressure outlet conditions were imposed at the entrance and exit of the duct. The incoming air entered with a steady, uniform velocity and an initial temperature of 300 K. Since the flow is driven by pressure, the outlet pressure was maintained at 1.013×10⁵ Pa. The boundary conditions used in the numerical simulations are summarised in Table 1.

Table 1. Computational domain boundary conditions [44].

Boundary domain	Zone type	Boundary condition
Inlet	Velocity inlet	The velocity for a fully developed flow is computed according to the associated Reynolds number. $w = V_{in}$, $u = v = 0$, $T = T_{in} = 300$ K
Outlet	Pressure outlet	$\left(\frac{\partial v}{\partial z} \right)_{z=L} = 0$, $\left(\frac{\partial T}{\partial z} \right)_{z=L} = 0$, At the outlet, the pressure is set to zero gauge, and the remaining flow parameters are estimated based on values from the domain.
Bottom duct wall	Wall	Adiabatic wall condition with a no-slip boundary. $u = v = w = 0$, $q = 0$ W/m ²
Spheres	Wall	Adiabatic wall condition with a no-slip boundary. $u = v = w = 0$, $q = 0$ W/m ²
Top heated wall	Wall	Uniform constant heat flux condition with a no-slip boundary $u = v = w = 0$, $q = 1000$ W/m ²

The interaction between the pressure and velocity fields in the numerical simulation was handled through the SIMPLE (semi-implicit method for pressure-linked equations) algorithm. This iterative procedure maintains mass conservation by repeatedly adjusting both pressure and velocity until a stable solution is reached. To solve the discretised momentum and energy equations, a second-order upwind scheme was employed, offering improved numerical precision by accounting for flow direction and reducing artificial diffusion.

The geometric setups and operating parameters applied throughout the analysis are compiled in Table 2, which presents the essential inputs utilised in the simulations.

Table 2. Geometrical and operating parameters [44].

Geometrical and operating parameters	Symbol	Value/range
Diameter of sphere (mm)	D	25
Relative roughness pitch	P/D	3, 6 and 9
Specific heat of air (J/(kg·K))	C_p	1008
Thermal conductivity of air (W/(m·K))	k_a	0.026
Dynamic viscosity of air (kg/(s·m))	μ	1.865×10^{-5}
Reynolds number	Re	3500–23 500

The Nusselt number (Nu) is widely used to evaluate the efficiency of convective heat transfer inside a SAH channel [45]:

$$Nu = \frac{hD}{k_a}, \quad (9)$$

where the hydraulic diameter D for a rectangular duct is:

$$D = \frac{4A}{P}. \quad (10)$$

The heat transfer coefficient h is found using the relation:

$$h = \frac{q}{(T_s - T_b)}, \quad (11)$$

where q is the heat flux on the absorber plate, T_s is the average surface temperature of the heated plate, and T_b represents the bulk mean temperature of the airflow.

The friction factor (f) for fully developed turbulent flow through a roughened duct can be found as [46]:

$$f = 2 \frac{[\Delta p]}{\rho V^2} D, \quad (12)$$

where Δp is the difference between the static pressure at the entrance and at the exit of the duct

$$\Delta p = p_{in} - p_{out}. \quad (13)$$

Lewis [47] introduced the thermo-hydraulic performance parameter (THPP), which states the following and compares the value of thermal gain to that of the frictional loss:

$$THPP = (Nu_r / Nu_s) / (f_r / f_s)^{1/3}, \quad (14)$$

where Nu and f are the Nusselt number and friction factor, respectively. The parameters with suffix r and s refer to roughened and smooth configurations of SAHs.

3.1.2. Meshing and grid independence test

The different design configurations were first modelled in SolidWorks V23, after which the mesh generation and numerical simulations were carried out using ANSYS V21.0. The computational domain was discretised using an unstructured tetrahedral mesh, which allowed accurate representation of the complex geometric features of the spherical turbulators and the duct surfaces. While prismatic (wedge-type) layers were introduced near the walls (top and bottom absorber walls as well as around the

spherical turbulators) to properly resolve the near-wall flow physics like flow separation, recirculation zones and boundary layer development, which are critical for predicting heat transfer enhancement and frictional losses in the solar air heater duct. A total of 5 inflation layers were applied near the solid surfaces with a first layer thickness of approximately 0.02 mm and a growth rate of 1.2, ensuring adequate resolution of the velocity and thermal boundary layers. The resulting dimensionless wall distance (y^+) values were maintained within the acceptable turbulence modelling range of 30–200 for the selected turbulence models. The inlet boundary conditions were specified using a turbulence intensity of 5% (0.05), which is commonly adopted for generic high-speed and internal turbulent flows. The turbulence length scale was defined as 0.07 times the hydraulic diameter of pipe to represent the characteristic size of turbulent eddies at the inlet.

These mesh structures were created through the ICEM CFD module in ANSYS, ensuring smooth element transitions and faithful representation of the geometric features. As shown in Fig. 3, inflation layers were applied along both the spherical turbulators (ST) and duct walls to effectively resolve near-wall flow characteristics during the analysis. The simulations were then performed using the Fluent solver integrated in ANSYS V21.0.

A grid independence analysis was performed to verify the dependability and accuracy of the computational setup. All configurations were independently subjected to grid-independence testing; however, for clarity and readability, only the inclined configuration is presented in the manuscript as a representative case. The inclined spherical turbulator configuration of the solar air heater (SAH) was evaluated under multiple mesh densities while maintaining a constant Reynolds number of 11 000. The goal was to determine how changes in mesh resolution influenced the predicted thermal behaviour. Table 3 reports the variation in Nusselt number as a function of the total cell count. The results showed that when the mesh consisted of 1 889 780 elements, the Nusselt number obtained with the k - ε turbulence model differed by only 0.95% from those predicted with finer grids. Since this difference is insignificant, additional refinement would not meaningfully improve the accuracy while significantly increasing computational time. Therefore, the mesh containing 1 889 780 cells was selected for all further simulations to achieve an optimal balance between computational cost and solution reliability.

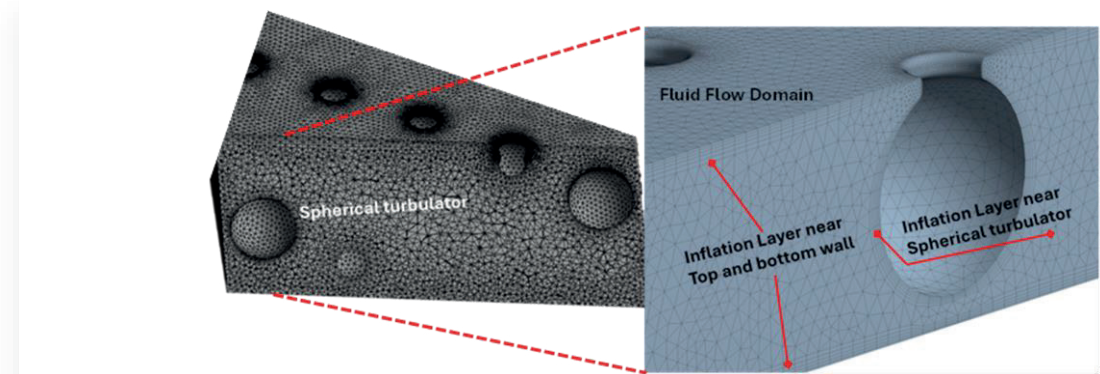


Fig. 3. Meshing of SAH fluid flow domain with zoomed view near spherical turbulators.

Table 3. Grid independence test.

No.	Number of cells	Nusselt number values	Deviation (%)
1	121 5439	96.45	—
2	129 4081	92.87	3.71
3	134 8725	88.29	4.93
4	144 2977	86.97	1.49
5	169 5313	85.74	1.41
6	188 9780	84.2	1.79
7	201 2542	83.4	0.95

3.1.3. Validation of the present numerical work

To ensure the accuracy and reliability of the numerical approach, a validation study was performed in which the results from the present SAH simulations were compared with established findings from earlier research. The validation involved modelling a smooth solar air heater duct without any roughness elements. The numerically obtained Nusselt number and friction factor were evaluated against well-known theoretical correlations for smooth ducts, specifically, the Gnielinski and Dittus-Boelter correlations for heat transfer, and the Blasius equation for friction factor estimation [45]. For Reynolds numbers below 10 000, the Gnielinski correlation was applied, while the Dittus-Boelter correlation was used for Reynolds numbers exceeding this range. The Nusselt number and friction factor predicted using standard $k-\epsilon$, RNG $k-\epsilon$, realizable $k-\epsilon$, standard $k-\omega$, and SST $k-\omega$ turbulence models were evaluated over a Reynolds number range of 4000–20 000 and compared with well-established empirical correlations (the Gnielinski and Dittus-Boelter correlations for the Nusselt number, and the modified Blasius equation for the friction factor). The comparison plots in Fig. 4 added in the revised manuscript show that the RNG $k-\epsilon$ model provides predictions that closely follow the empirical correlations while maintaining stable trends across the entire Reynolds number range. Other models either slightly overpredict or deviate from the correlations. Based on this comparative assessment, the RNG $k-\epsilon$ model was selected as it provides reliable agreement with established correlations for both heat transfer and flow friction characteristics in the present solar air heater configuration. The comparison showed excellent consistency, with average deviations of 3.1% for the friction factor and 4.7% for the Nusselt number.

The Nusselt number for a smooth duct has been calculated using the Dittus-Boelter equation for $Re > 10\,000$ and the Gnielinski correlation for $Re < 10\,000$, as expressed in Eqs. (15) and (16), respectively [39]:

$$Nu_s = 0.023 Re^{0.8} Pr^{0.4}, \quad (15)$$

$$Nu_s = \frac{\left(\frac{f}{2}\right)(Re - 1000)Pr}{1 + 12.7\left(\frac{f}{2}\right)^{\frac{1}{2}}(Pr^{\frac{2}{3}} - 1)}, \quad (16)$$

where $f = (1.58 \ln(Re) - 3.82)^{-2}$ and Pr is the Prandtl number.

The modified Blasius equation presents the friction factor in a smooth duct as [40]:

$$f_s = 0.085 Re^{-0.25}. \quad (17)$$

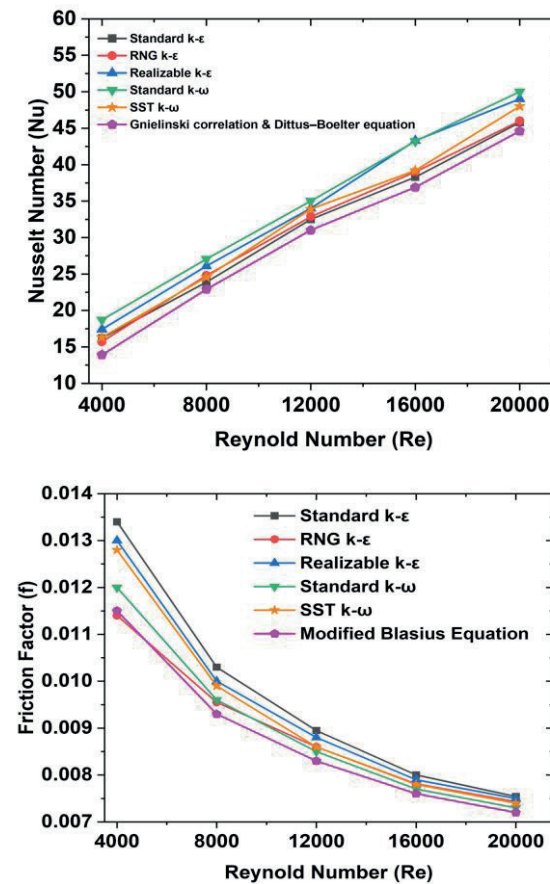


Fig. 4. Comparison of numerical results with correlations for a smooth duct.

An additional validation study was conducted using experimental data, in which the numerical simulation results of a solar air heater equipped with V-shaped ribs having a symmetrical gap were compared with the corresponding experimental findings reported in [48]. It can be observed from Fig. 5 that the Nusselt number increases consistently with the increasing Reynolds number for both approaches, indicating enhanced convective heat transfer at higher flow rates. The numerical predictions closely follow the experimental trend across the entire Reynolds number range (4000–20 000), demonstrating good agreement between the computational model and the experimental observations. The deviation between the two datasets appears relatively small, suggesting that the adopted numerical model and boundary conditions are capable of reliably predicting the heat transfer characteristics of the ribbed channel. However, minor discrepancies between experimental and numerical results may arise due to several possible sources of error. These include experimental uncertainties such as thermocouple measurement errors, heat losses to the surroundings, and non-uniform heating conditions. Additionally, numerical errors may result from mesh discretisation, turbulence model limitations, and simplified assumptions such as steady-state flow or uniform material properties. Variations in rib geometry during fabrication and slight differences in boundary conditions between the experiment and simulation may also contribute to the observed differences. Overall, the small deviation indicates acceptable validation of

the numerical model for predicting the heat transfer behaviour of the investigated configuration.

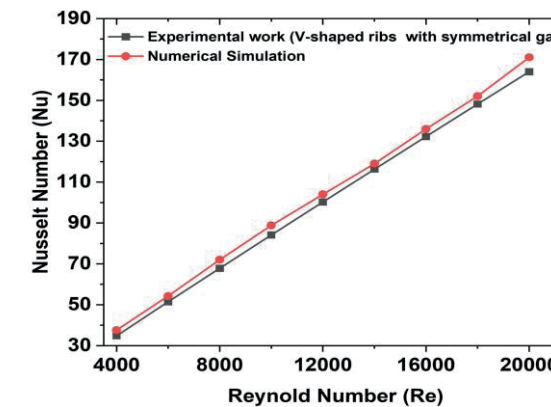


Fig. 5. Validation of the CFD model through comparison with experimental data for V-shaped rib configuration.

3.2. ML surrogate modelling

High-fidelity CFD provides reliable thermo-hydraulic performance predictions for solar air heaters (SAHs) equipped with spherical turbulators; however, repeated CFD evaluations become computationally expensive when configuration screening, sensitivity analyses, or optimisation are required. To enable

rapid evaluation over the investigated design space, a supervised machine-learning (ML) surrogate model was developed to predict the Nusselt number (Nu) and friction factor (f) directly from the governing operating and geometric descriptors (the Reynolds number and turbulator arrangement, together with the dimensionless spacing/ratio parameter used in the numerical campaign). The surrogate is trained on the CFD-derived dataset spanning over the range of $Re = 3500$ – $23\,500$ and the three discrete levels of the dimensionless spacing/ratio parameter (3, 6 and 9), consistent with the simulation campaign.

For clarity, the ML workflow can be understood as four sequential stages: dataset assembly from CFD outputs, cleaning and feature construction, leakage-safe grouped model development, and final reliability assessment on a frozen holdout set. First, CFD cases were compiled into a tabular dataset in which each row corresponds to one simulated operating condition and geometry combination. Second, the raw descriptors were cleaned, standardised, and expanded into engineered features that reflect known thermo-fluid scaling behaviour. Third, the processed data were split by geometry identity so that similar cases from the same configuration-pitch pair did not appear in both training and evaluation subsets. Finally, the selected surrogate models were assessed using grouped cross-validation and a frozen holdout test with uncertainty estimation. A schematic overview of this workflow is shown in Fig. 6.

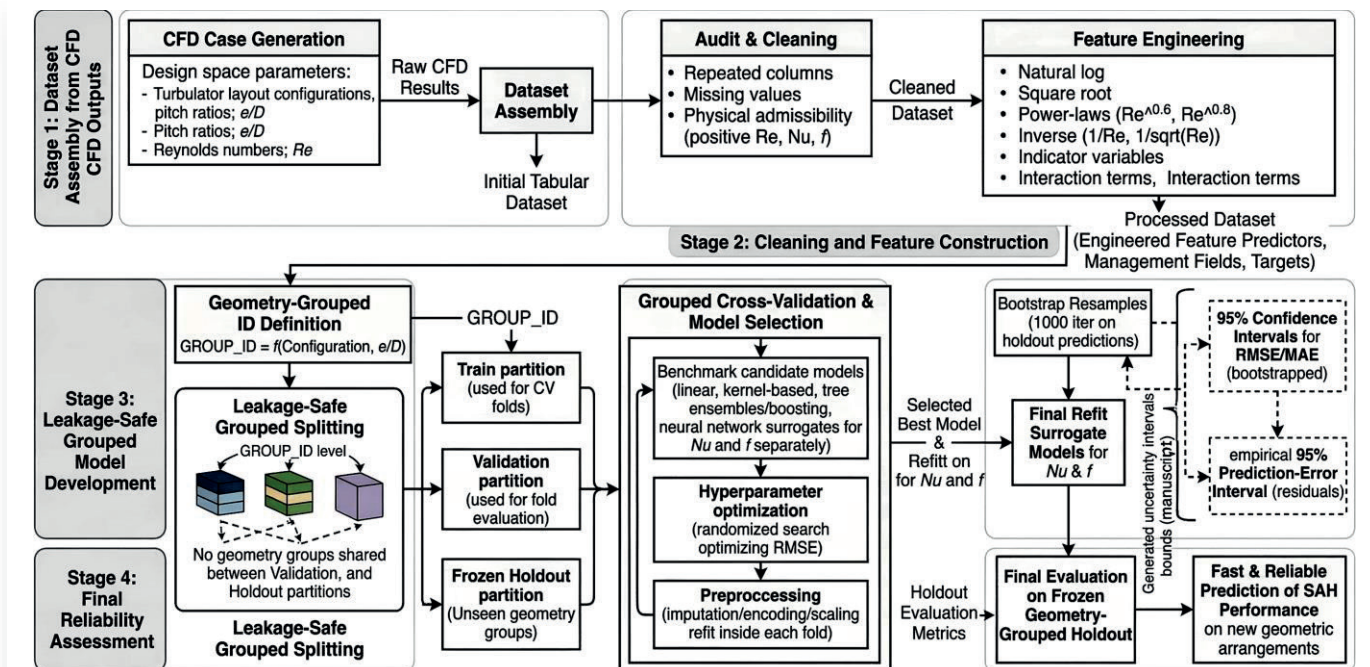


Fig. 6. Schematic overview of the CFD-derived surrogate-modelling workflow.

The ML workflow follows reporting for data-driven surrogates: (i) transparent dataset audit and physically consistent cleaning, (ii) leakage-safe train-test splitting using geometry-level grouping, and (iii) exploratory data analysis (EDA) to verify coverage of the operating envelope and to interpret dominant trends prior to model training, as shown in Fig. 7.

In contrast to many earlier CFD-ML studies that mainly report pointwise prediction accuracy, the present surrogate work-

flow is structured around geometry-aware generalisation and design screening. The methodological contribution here is the combination of grouped geometry-level splitting, explicit leakage control, frozen-holdout validation, uncertainty estimation, and post-prediction thermo-hydraulic ranking for spherical-turbulator configurations. This makes the surrogate framework more directly relevant for engineering selection tasks than a purely accuracy-oriented regression exercise.

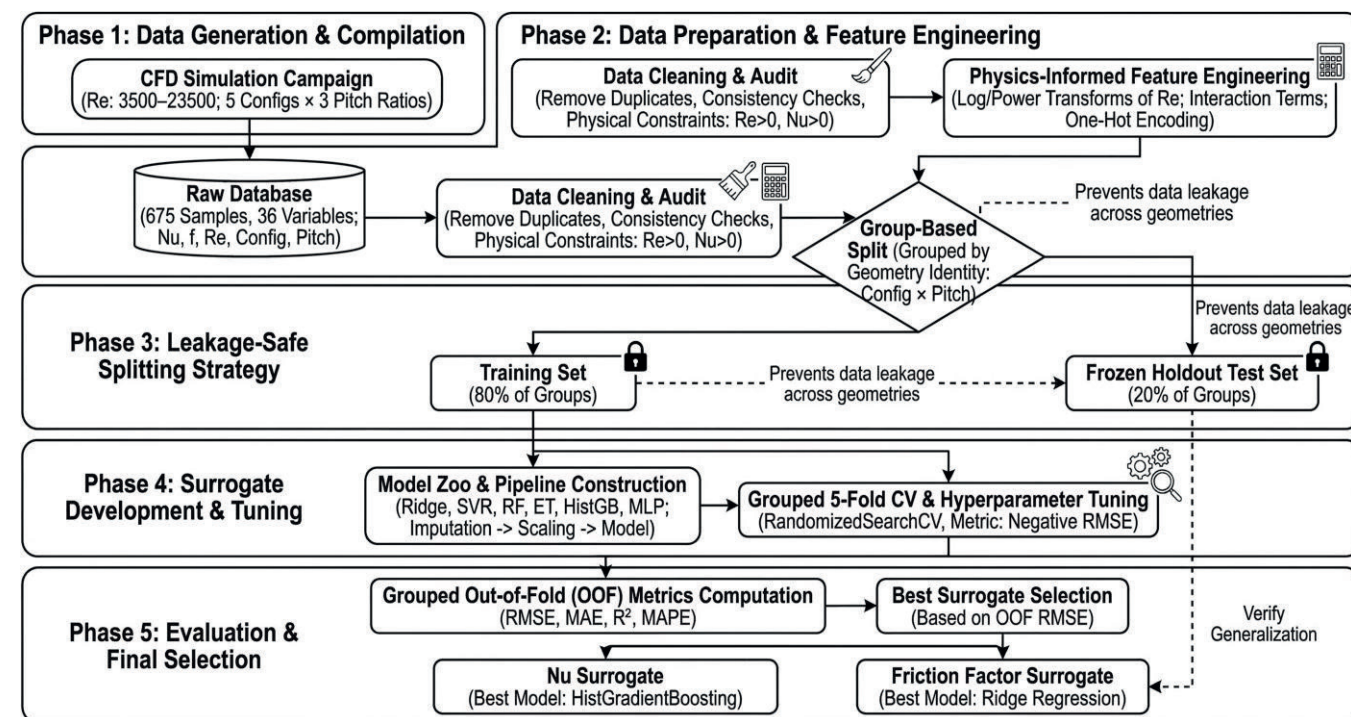


Fig. 7. Overall ML approach flowchart.

3.2.1. Dataset description

The compiled dataset contains 675 samples and 36 columns, including two supervised targets (Nu and friction factor), categorical descriptors of the turbulator arrangement (configuration, 5 levels: V-, M-, W-shaped, inclined, arc-shaped), and a dimensionless spacing/ratio parameter stored as P_{over_D} (three levels: 3, 6, 9). These five configurations correspond to the five spherical-turbulator arrangements investigated in the CFD study.

To improve learnability on tabular models, the raw physical inputs were augmented with physics-informed feature engineering, including logarithmic, inverse, square-root and power-law transforms of Reynolds number, together with interaction terms involving the spacing ratio and indicator variables for configuration class. The complete feature scheme used for ML, including raw variables, engineered variables, units, ranges and transformation definitions, is documented in Table 4. To avoid leakage, only pre-target operating and geometric descriptors were used as predictors; the response variables (Nu and friction factor), thermo-hydraulic performance metrics derived from them, and split-management fields were excluded from model inputs.

Table 4 distinguishes clearly between raw design inputs, engineered predictors, split-management fields, target variables, and derived thermo-hydraulic metrics. Variables such as Nu, friction factor and THPP were never supplied to the ML models as predictors. Likewise, GROUP_ID and SPLIT were used only for grouped evaluation and frozen holdout construction, not for training. This separation was enforced to eliminate inadvertent target or partition leakage.

In practical terms, preprocessing was performed in a simple sequence. The compiled CFD table was first checked for repeated columns, inconsistent names, and missing target values.

After this audit, categorical labels for configuration were standardised so that each layout appeared under a single consistent name. Numerical inputs were then screened for physical admissibility, retaining only rows with positive Reynolds number, positive Nusselt number and positive friction factor. Next, the raw design descriptors were transformed into a richer predictor set using interpretable mathematical operations such as logarithm, inverse, square-root and power-law terms, together with configuration indicators and selected interaction terms. These engineered variables were created only from pre-target quantities and were intended to help the regression models represent known nonlinear thermo-fluid behaviour more effectively.

A conservative audit-and-cleaning protocol was applied to ensure that the surrogate is trained on physically meaningful and internally consistent samples. Because aggregated datasets may occasionally contain repeated fields (e.g., duplicate “Re” columns from concatenation), columns were standardised by trimming/normalising names, and then checked for duplication. Perfectly duplicated columns (value-by-value identical) were removed to prevent redundant signals and unstable parameter estimation in linear models. When multiple Reynolds-number columns existed, a single canonical Re was selected based on numeric type and completeness. Furthermore, since the task is supervised regression, any sample missing either target (Nu or friction factor) was excluded. In the final compiled dataset used here, the targets are fully populated (675/675 non-null), which avoids bias from imputation on the dependent variables. Samples violating basic physical admissibility were removed using non-aggressive guards: $Re > 0$, $Nu > 0$, friction factor > 0 . These checks are lightweight but important: they prevent accidental inclusion of corrupted rows (e.g. sign errors) that can distort loss surfaces and inflate error metrics.

Table 4. Summary of variables used for ML surrogate modelling (targets, raw inputs and engineered features).

Variable	Type	Range / Levels	Transformation / Definition	Used as predictor	Leakage risk
Re	Raw input	3 500–23 500	Reynolds number from CFD case definition	Yes	No
P_{over_D} (or P/D as stored in dataset)	Raw input	3, 6, 9	Dimensionless pitch/spacing ratio used in the design study	Yes	No
Configuration	Raw categorical input	V, M, W, Inclined, arc	Turbulator layout label	Yes	No
Config_V	Engineered categorical flag	0, 1	One-hot encoding of V-shaped layout	Yes	No
Config_M	Engineered categorical flag	0, 1	One-hot encoding of M-shaped layout	Yes	No
Config_W	Engineered categorical flag	0, 1	One-hot encoding of W-shaped layout	Yes	No
Config_inclined	Engineered categorical flag	0, 1	One-hot encoding of inclined layout	Yes	No
Config_ARC	Engineered categorical flag	0, 1	One-hot encoding of arc-shaped layout	Yes	No
$\ln(Re)$	Engineered numeric	Computed	Natural logarithm of Re	Yes	No
$\sqrt{Re}$	Engineered numeric	Computed	Square root of Re	Yes	No
$Re^{0.6}$	Engineered numeric	Computed	Power-law transform of Re	Yes	No
$Re^{0.8}$	Engineered numeric	Computed	Power-law transform of Re	Yes	No
$1/Re$	Engineered numeric	Computed	Inverse Reynolds number	Yes	No
$1/\sqrt{Re}$	Engineered numeric	Computed	Inverse square-root Reynolds term, if used	Yes/No	No
$\ln(P_{over_D})$	Engineered numeric	$\ln(3)$, $\ln(6)$, $\ln(9)$	Natural logarithm of pitch ratio, if used	Yes/No	No
$1/P_{over_D}$	Engineered numeric	$1/9$ – $1/3$	Inverse pitch ratio, if used	Yes/No	No
$\sqrt{P_{over_D}}$	Engineered numeric	Computed	Square-root pitch term, if used	Yes/No	No
$Re \times P_{over_D}$	Engineered interaction	Computed	Interaction between flow regime and spacing ratio	Yes	No
$\ln(Re) \times P_{over_D}$	Engineered interaction	Computed	Interaction between transformed Re and spacing ratio	Yes	No
$Re^{0.8} \times P_{over_D}$	Engineered interaction	Computed	Power-law Reynolds–pitch interaction	Yes/No	No
Re bin_low	Engineered categorical flag	0, 1	Low-Reynolds operating bin	Yes/No	No
Re bin_mid	Engineered categorical flag	0, 1	Mid-Reynolds operating bin	Yes/No	No
Re bin_high	Engineered categorical flag	0, 1	High-Reynolds operating bin	Yes/No	No
GROUP_ID	Split-management field	15 groups	Geometry identity = Configuration $\times P_{over_D}$	No	Yes
SPLIT	Split-management field	Train / Test	Dataset partition label	No	Yes
Nu	Target	35.7–177.8	CFD-derived Nusselt number	No	Yes
Friction factor	Target	Dataset-specific	CFD-derived friction factor	No	Yes
THPP	Derived performance metric	Dataset-specific	Calculated from Nu and FrictionFactor	No	Yes

Categorical labels were standardised (case/whitespace normalisation) so that the same configuration is not split across multiple string variants. This is essential for stable one-hot encoding and for geometry-level grouping. A key risk in surrogate modelling for engineered thermal systems is data leakage: if multiple samples are generated from the same geometry (same arrangement and spacing ratio) across many Reynolds numbers, a naive random split may place near-identical geometry patterns in both training and test sets, inflating apparent generalisation.

To address this, the dataset was partitioned using a geometry identity group:

$$GROUP_ID = f(\text{Configuration}, e/D),$$

where e/D denotes the discrete spacing/ratio level used in the parametric campaign (stored as P_{over_D} in the dataset). This grouping produces 15 geometry groups (= 5 configurations $\times$ 3 spacing levels), aligning with the CFD design-of-experiments structure. From the full compiled table, only variables available before target evaluation were allowed to enter the predictor matrix. Specifically, the Reynolds number, spacing-ratio information, categorical configuration labels, and their engineered transforms were retained, whereas Nu, friction factor, THPP, SPLIT, and GROUP_ID were not used as predictors. This rule was adopted to prevent target leakage and to ensure that the surrogate learns only from quantities that would be known at design-screening time. A group shuffle split strategy was then used to create a frozen holdout test set (20% by groups), ensuring that the test set contains geometry groups unseen during training.

This makes the reported test performance meaningful for design screening, because the surrogate is evaluated on genuinely new geometry identities rather than interpolating within the same group.

Interpretation and significance. The split-check histogram (Fig. 8) is an important diagnostic: it verifies that, even after enforcing group disjointness, the Reynolds-number envelope remains comparable across training and test partitions. This reduces the chance that performance differences are driven simply by different Re ranges rather than by geometry generalisation. In other words, the surrogate is forced to learn transferable relationships between the inputs and (Nu, f) , rather than memorising a specific configuration’s curve.

Figure 9 confirms that the dataset spans the complete Reynolds-number and friction factor range investigated in the CFD campaign. This coverage is crucial because Nu and friction factor in roughened ducts are strongly Reynolds-dependent; a surrogate trained on a narrower window would extrapolate unreliably outside its calibration range. From a modelling standpoint, the broad Re coverage motivates the inclusion of nonlinear Reynolds transforms (e.g. $\ln Re$, $Re^{0.6}$, $Re^{0.8}$, $1/Re$), which mirror classical convective scaling behaviour and help reduce the functional complexity that the regression models must learn.

As expected for forced convection in internal flows, Nu increases with Re across all configurations (Fig. 10), reflecting enhanced mixing and reduced thermal boundary-layer thickness at higher flow rates. The separation between configuration-specific point clouds highlights that the turbulator arrangement is not a minor categorical label but a first-order determinant of

heat-transfer augmentation: at a fixed Re , different arrangements produce systematically different Nu levels. This validates two dataset-design decisions: (i) retaining configuration explicitly (categorical + indicators), and (ii) adding interaction terms combining Re with the spacing/ratio feature, because layout-induced turbulence generation interacts with the flow regime rather than acting as a simple additive offset.

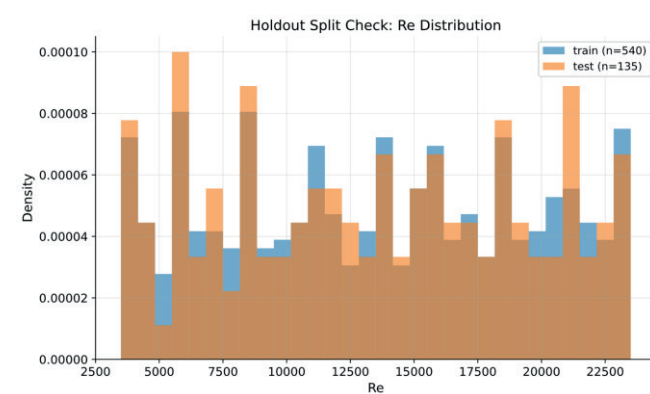


Fig. 8. Train-test split check showing similar Reynolds-number coverage in both splits (leakage-safe group split).

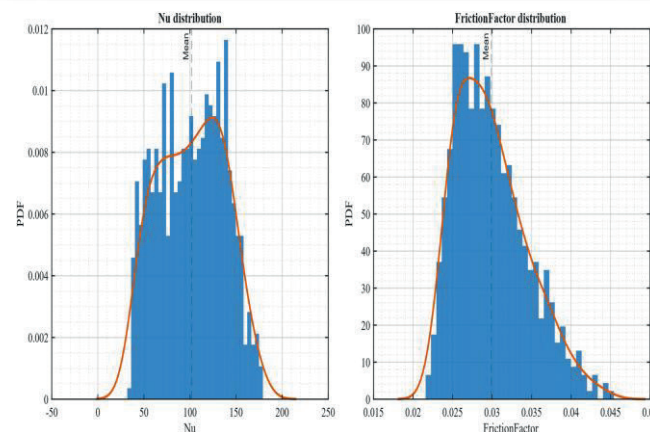


Fig. 9. Re number and friction factor distribution of the compiled dataset.

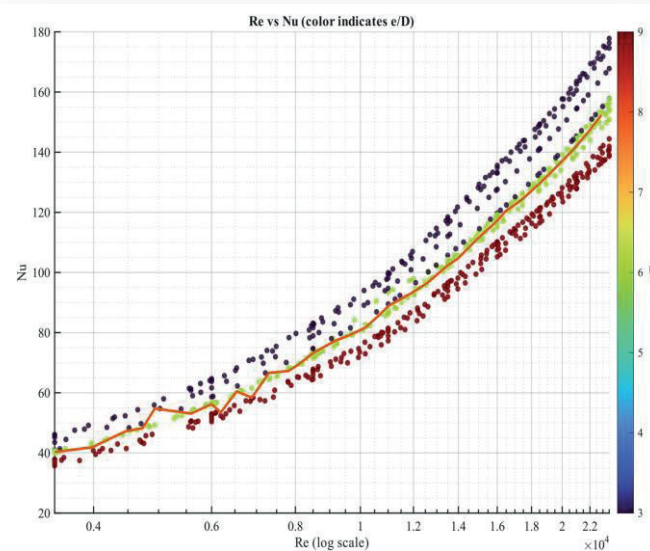


Fig. 10. Nusselt number versus Reynolds number for each spherical-turbulator configuration.

Figure 11 shows that friction factor decreases with increasing Re in the investigated turbulent range, consistent with internal-flow behaviour where inertial effects dominate and the relative contribution of viscous shear diminishes at higher Re . Importantly, the configuration-dependent offsets indicate that the same arrangement choices that increase turbulence and heat transfer also modify pressure losses. For surrogate modelling, this implies a multi-physics mapping: the model must capture both monotonic trends with Re and the geometry-driven shifts in the friction- Re relationship. This is a typical scenario where tree-based ensemble models and gradient boosting often perform well, because they can represent nonlinear interactions and categorical splits without requiring an explicit functional form.

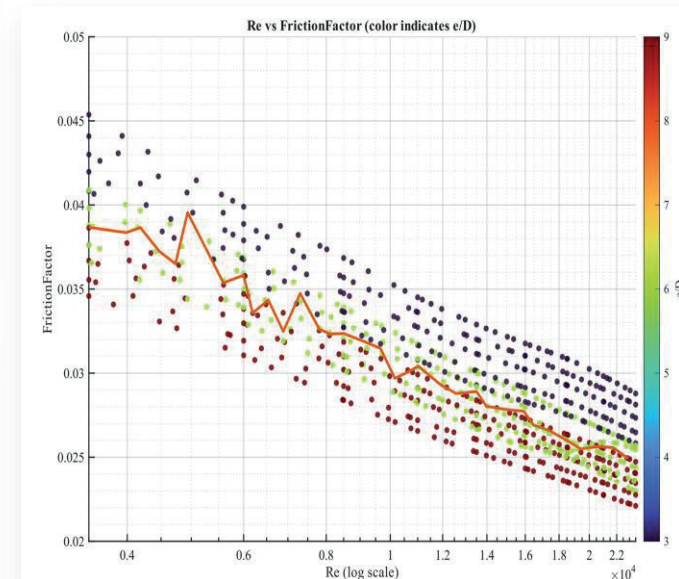


Fig. 11. Friction factor versus Reynolds number for each spherical-turbulator configuration.

The trade-off map (see Fig. 12) provides a compact thermo-hydraulic perspective: points with higher Nu are generally associated with higher f , reflecting the classical enhancement penalty where turbulence promoters improve convective transfer at the expense of pressure drop.

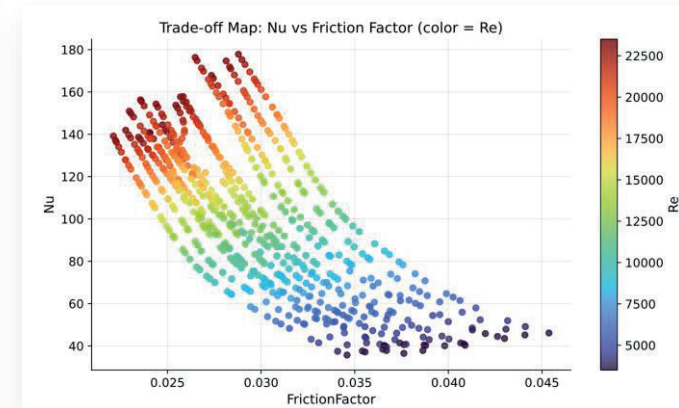


Fig. 12. Trade-off map between the Nusselt number and friction factor (colour-coded by Reynolds number).

The Reynolds-number colour gradient further clarifies the operating-regime dependence: as Re increases, Nu tends to rise

while f tends to moderate, indicating that some high- Re operating points can occupy a relatively favourable region of high heat transfer with less severe friction penalties. For the ML workflow, this figure justifies treating Nu and friction factor as separate regression targets (rather than forcing a single composite metric during training), because design optimisation and THPP evaluation can then be performed post-prediction using physically interpretable objectives.

3.2.2. ML model development and evaluation

This subsection describes the development of data-driven surrogate models for rapid prediction of the two CFD-derived performance indicators, i.e. Nu and f , for the investigated SAH equipped with spherical turbulators. The surrogate was trained over the same design space explored in the numerical campaign, comprising five turbulator arrangements (V-, M-, W-shaped, inclined and arc-shaped layouts) and three discrete spacing levels (relative roughness pitch values of 3, 6 and 9), while keeping the spherical diameter constant to preserve geometric uniformity.

Two independent supervised regression models were formulated:

$$\hat{y} = \mathcal{M}(\mathbf{x}), y \in \{Nu, f\}, \quad (18)$$

where $\mathbf{x}$ denotes the input feature vector containing operating and geometric descriptors. Separate models were trained for Nu and f (rather than a single multi-output regressor) to allow target-specific inductive bias and hyperparameter tuning, as the thermo-hydraulic response surfaces for heat transfer and pressure losses can exhibit different nonlinearities and noise characteristics.

All available non-target variables were considered as candidate predictors, excluding the data-management fields used for splitting and grouping (SPLIT and GROUP_ID). A complete variable dictionary, including units, ranges, transformation formulas and leakage status, is provided in Table 4. The predictors comprised: (i) primary physical inputs (e.g. Reynolds number and the spacing/ratio parameter e/D), (ii) configuration descriptors (categorical configuration label and its indicator repre-

sentation), and (iii) physics-informed engineered features (logarithmic, power-law and interaction terms), which aim to linearise known scaling relationships and represent coupled effects between flow regime and spacing.

To prevent leakage during cross-validation and to ensure consistent transformations across folds, preprocessing was implemented within the training pipeline and re-fitted inside each cross-validation split. The pipeline consisted of: (i) median imputation for missing values, followed by optional standardisation (zero mean, unit variance) for scale-sensitive models, (ii) most-frequent imputation, followed by one-hot encoding with “unknown category” handling to ensure robustness if an unseen configuration label appears at inference time (e.g. due to label normalisation or future extensions).

The preprocessing pipeline can be summarised as follows. For numerical variables, missing values were replaced with the median computed from the training fold only; scale-sensitive models then received standardised inputs with zero mean and unit variance, again fitted only on the training fold. For categorical variables, missing labels were replaced with the most frequent category from the training fold, followed by one-hot encoding to convert configuration labels into binary indicators. The preprocessing steps were embedded inside the model pipeline so that the validation fold and holdout set were never used to estimate imputation values, scaling factors or category mappings. This procedure ensures that model evaluation reflects true generalisation rather than information leakage from preprocessing.

This design ensures that all preprocessing parameters (imputation statistics, scaling factors and categorical mappings) are learned exclusively from the training portion of each fold. Grouped 5-fold cross-validation (GroupKFold, $n_splits = 5$) using GROUP_ID; randomised hyperparameter search with 30 iterations per model (RandomizedSearchCV, $n_iter = 30$); primary scoring = negative $RMSE$; random seed = 42; preprocessing (imputation/encoding/scaling) refit inside each fold (see Table 5).

Table 5. Candidate ML models, preprocessing choice, grouped CV protocol and hyperparameter search space used for surrogate development (Nu and friction factor are modelled separately).

Model family	Preprocessing (numeric)	Fixed settings	Hyperparameters tuned (randomised search space)
Ridge regression	Standardisation ON	—	$\alpha \in [10^{-4}, 10^3]$ (60 log-spaced values)
SVR (RBF kernel)	Standardisation ON	kernel = RBF	$C \in [10^{-1}, 10^3]$ (40 log-spaced); $\gamma \in [10^{-4}, 10^0]$ (40 log-spaced); $\epsilon \in [10^{-4}, 10^{-1}]$ (25 log-spaced)
Random forest regressor	Standardisation OFF	$n_estimators = 600$	$max_depth \in \{none, 10, 16, 24, 32\}$; $min_samples_split \in \{2, 4, 8, 12\}$; $min_samples_leaf \in \{1, 2, 4, 8\}$; $max_features \in \{“sqrt”, 0.5, 0.7, 1.0\}$
Extra trees regressor	Standardisation OFF	$n_estimators = 600$	$max_depth \in \{none, 8, 12, 18, 26\}$; $min_samples_split \in \{2, 4, 8, 12\}$; $min_samples_leaf \in \{1, 2, 4, 8\}$; $max_features \in \{“sqrt”, 0.5, 0.7, 1.0\}$
Histogram gradient boosting regressor	Standardisation OFF	—	$learning_rate \in [10^{-2.5}, 10^{-0.2}]$ (40 log-spaced; ~ 0.00316 – 0.631); $max_depth \in \{none, 4, 6, 8, 10\}$; $max_leaf_nodes \in \{15, 31, 63, 127\}$; $min_samples_leaf \in \{10, 20, 30, 50\}$; $l2_regularization \in [10^{-6}, 10^1]$ (30 log-spaced; 10^{-6} to 10)
MLP regressor (optional benchmark)	Standardisation ON	ReLU activations; MSE loss; AdamW optimiser; batch size = 256	Architecture: [256, 128, 64] hidden units; dropout = 0.10; learning rate = 1×10^{-3} ; epochs ≤ 250 with early stopping (patience = 30)

Median imputation was applied to numerical features; most-frequent imputation and one-hot encoding were applied to categorical features; standardisation was enabled only for scale-sensitive models. A compact but diverse model set (“model zoo”) was selected to balance interpretability, nonlinear capacity and sample efficiency on tabular data:

1. **Ridge regression (L2-regularised linear model):** a strong baseline for engineered features; regularisation mitigates multicollinearity among transformed predictors,
2. **Support vector regression (SVR) with RBF kernel:** a nonlinear margin-based learner that can fit smooth response surfaces; applied with standardised inputs,
3. **Random forest regression:** bagged decision trees providing robust nonlinear modelling and limited sensitivity to monotonic transforms,
4. **Extra trees regression:** randomised tree ensembles that typically yield strong accuracy and reduced variance on tabular datasets,
5. **Histogram-based gradient boosting regression:** a boosting approach well-suited to capturing complex interactions with efficient training,
6. **Optional multi-layer perceptron (MLP):** a small feed-forward neural network trained with early stopping; used as an additional nonlinear benchmark.

Ridge (α); SVR (C, γ, ϵ); random forest / extra trees (depth, minimum samples, max features); gradient boosting (learning rate, depth/leaf nodes, minimum leaf samples, L2 regularisation); MLP (hidden sizes, dropout, learning rate, epochs with early stopping).

Model development was performed strictly on the training split, while the holdout test split remained frozen for final assessment (reported in Results). Cross-validation was conducted using GroupKFold based on GROUP_ID, i.e., the geometry identity formed by configuration and spacing level. This grouped strategy is essential for surrogate modelling of parametric CFD datasets because multiple Reynolds-number samples from the same geometry are highly correlated; grouping forces validation folds to contain geometry identities not present in the corresponding training fold, producing a more realistic estimate of generalisation to unseen configurations within the investigated design space.

The grouped validation strategy can be interpreted in a simple way: all CFD samples belonging to the same geometry family, defined by a specific configuration and pitch ratio, were kept together during splitting. Thus, when one geometry group was placed in a validation fold or in the frozen holdout set, the model had not seen any samples from that same group during training. This is stricter than random splitting and is more appropriate for engineering screening, because the real task is to predict performance for a new configuration-pitch combination rather than for a repeated point from a geometry already seen during training. A schematic of this grouped split strategy is shown in Fig. 13. It illustrates how grouped splitting was enforced at the geometry level so that all Reynolds-number samples from one configuration-pitch pair remained together during validation and frozen-holdout testing.

Unless otherwise stated, five folds were used. A fixed random seed was set for reproducibility of randomised procedures

(hyperparameter sampling and any model-internal randomness). For each candidate model and for each target (Nu and f), hyperparameters were optimised using randomised search under grouped cross-validation. The primary tuning objective was minimisation of root mean squared error ($RMSE$), chosen because it penalises large errors and is commonly used for regression surrogates intended for engineering screening. After the search, the best hyperparameter setting was refit on the full training portion.

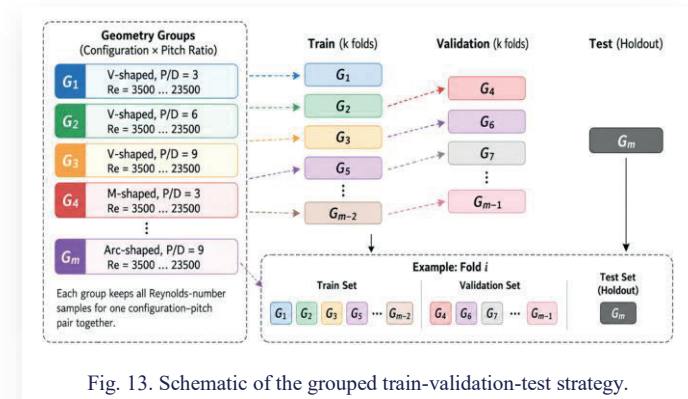


Fig. 13. Schematic of the grouped train-validation-test strategy.

To enable reproducible and fair comparison across models, the same grouped fold structure and the same metric definition were used during tuning for all candidates. Although final numerical results are presented in the Results section, evaluation methodology is defined here. For each tuned model, out-of-fold (OOF) predictions were generated under grouped cross-validation. OOF prediction provides a nearly unbiased estimate of model performance on unseen groups because each training sample is predicted by a model that did not observe its geometry group during fitting.

Four complementary metrics were computed:

$$RMSE = \sqrt{\frac{1}{n} \sum_{i=1}^n (y_i - \hat{y}_i)^2}, \quad (19)$$

$$MAE = \frac{1}{n} \sum_{i=1}^n |y_i - \hat{y}_i|, \quad (20)$$

$$R^2 = 1 - \frac{\sum_{i=1}^n (y_i - \hat{y}_i)^2}{\sum_{i=1}^n (y_i - \bar{y})^2}, \quad (21)$$

$$MAPE(\%) = \frac{100}{n} \sum_{i=1}^n \left| \frac{y_i - \hat{y}_i}{\max(|y_i|, \epsilon)} \right|. \quad (22)$$

Here, ϵ is a small constant used to avoid numerical instability when y_i is very small. $RMSE$ served as the primary model-selection criterion, while MAE , R^2 and $MAPE$ were used to provide complementary views of error magnitude, explained variance and relative error.

Metrics were computed from out-of-fold predictions generated under GroupKFold (5 folds), ensuring that each prediction is produced by a model trained without access to the corresponding geometry group (GROUP_ID). This provides a leakage-safe estimate of generalisation prior to frozen-test evaluation. $RMSE$, MAE , R^2 , and $MAPE$ were computed from grouped out-of-fold predictions; $RMSE$ was used for model selection.

To complement the classical ML models, an optional MLP regressor was trained using the same grouped cross-validation protocol and the same preprocessing (imputation, one-hot encoding and standardisation). The network employed multiple fully connected layers with ReLU activations and dropout regularisation. Training used the mean squared error loss and an adaptive optimiser (AdamW) with early stopping based on validation loss to reduce overfitting risk on the moderately sized dataset. This benchmark was treated as an additional candidate and compared on the same OOF metrics.

All experiments were executed in a controlled Python environment, recording software versions and hardware availability (CPU/GPU) for transparency. The best-performing pipeline for each target was archived for later inference and for frozen-test evaluation, ensuring that the exact preprocessing + model mapping can be reproduced without manual reconstruction.

After model selection under grouped cross-validation, the best-performing surrogate for each target was refit on the full training partition and evaluated once on the frozen geometry-grouped holdout test set. To quantify predictive reliability, uncertainty in the holdout error metrics was estimated using nonparametric bootstrapping of the holdout predictions with 1000 resamples. For each target, 95% confidence intervals were computed for $RMSE$ and MAE from the bootstrap distributions. In addition, an empirical 95% prediction-error interval was estimated from the holdout residuals to indicate the typical range within which new predictions may deviate from the CFD reference values inside the investigated design space.

4. Results and discussion

4.1. ML prediction results

The trained ML surrogates were assessed using leakage-safe, grouped out-of-fold (OOF) predictions, where geometry groups were defined by configuration and pitch/relative spacing (GROUP_ID), consistent with the operating envelope of the numerical dataset (Re = 3500–23 500; pitch ratio 3, 6 and 9). This grouped evaluation is intentionally stricter than point-wise random splits because it tests whether the model generalises to unseen geometry groups rather than merely interpolating among near-duplicate samples within the same configuration family.

In addition to grouped out-of-fold evaluation, the final selected models were assessed on a frozen geometry-grouped holdout test set containing configuration-pitch groups not used during training. On this stricter test, the hist gradient boosting model for Nu achieved an $RMSE$ of 3.45, MAE of 2.48, R^2 of 0.988, and $MAPE$ of 2.55%, with a bootstrapped 95% $RMSE$ interval of 3.10–3.85. For friction factor, the ridge model achieved an $RMSE$ of 0.000495, MAE of 0.000342, R^2 of 0.986, and $MAPE$ of 1.18%, with a bootstrapped 95% $RMSE$ interval of 0.000430–0.000570. The corresponding empirical 95% prediction-error intervals were [−6.5, 7.1] for Nu and [−0.00092, 0.00098] for friction factor, indicating stable predictive behaviour on previously unseen geometry groups.

The documented feature set also supports interpretability of the reported accuracy. Because the predictor matrix contains only pre-target operating and geometric variables and their transparent mathematical transforms, the reported performance

cannot be attributed to target-derived shortcuts. This strengthens confidence that the surrogates are learning transferable thermo-fluid relationships rather than exploiting hidden leakage.

Table 6 summarises the OOF accuracy for predicting Nu and f . Two key observations emerge: Firstly, the Nu prediction is best captured by a non-linear boosting model. The hist gradient boosting model (HistGB) achieved the best Nu accuracy ($RMSE = 3.299$; $R^2 = 0.9908$, $MAPE = 2.44\%$). Compared with the closest baseline (ridge regression, $RMSE = 3.928$), HistGB reduces $RMSE$ by ~16%. This indicates that even after extensive feature engineering (e.g. log and power transforms of Re and interaction terms with pitch), Nu retains non-linear interactions across the Reynolds number and configuration that are well represented by boosting ensembles. Secondly, the friction factor is highly predictable with a simple linear model. The ridge regression provided the lowest error for friction factor ($RMSE = 0.000470$; $R^2 = 0.9886$, $MAPE = 1.12\%$), outperforming the next-best model (HistGB, $RMSE = 0.000740$) by a ~36% $RMSE$ reduction. This suggests that within the present design space, f behaves as a smooth and near-linear function in the engineered feature space (e.g. inverse/logarithmic transforms of Re and discrete pitch levels), consistent with the fact that friction correlations in internal flows often become close to linear after appropriate transformations.

Table 6. Grouped OOF performance of ML surrogates for predicting Nu and friction factor (f).

(a) Target: Nu				
Model	OOF $RMSE$	OOF MAE	OOF R^2	OOF $MAPE$ (%)
HistGB	3.2990	2.3348	0.9908	2.4414
Ridge	3.9281	2.9664	0.9870	3.3052
ExtraTrees	4.0612	2.9025	0.9861	2.8326
SVR_RBF	4.1796	3.5516	0.9853	3.7464
RandomForest	4.2551	2.9797	0.9847	2.9317
TorchMLP_CPU	13.7627	11.5722	0.6773	12.2304

(b) Target: friction factor (f)				
Model	OOF $RMSE$	OOF MAE	OOF R^2	OOF $MAPE$ (%)
Ridge	0.000470	0.000325	0.9886	1.1233
HistGB	0.000740	0.000596	0.9717	2.0201
SVR_RBF	0.000744	0.000564	0.9714	1.9412
ExtraTrees	0.001177	0.001040	0.9284	3.5082
RandomForest	0.001255	0.001090	0.9186	3.6909
TorchMLP_CPU	0.002161	0.001710	0.6393	5.8947

Importantly, the CFD model used to generate the underlying dataset was previously validated against correlations/benchmarks, with reported deviations of the order of ~7–8% for Nu and friction factor in comparable validation tests. The surrogate modelling errors reported here therefore add only a small incremental uncertainty on top of the validated CFD responses, enabling fast screening/optimisation while preserving CFD-level fidelity. These results show that the contribution of the present study is not limited to achieving high regression accuracy. The key improvement over a conventional CFD-ML workflow is that the trained surrogates remain reliable under geometry-grouped cross-validation and frozen-holdout testing, while also supporting direct thermo-hydraulic ranking of candidate configurations. This combination of predictive fidelity, leakage-safe validation, uncertainty reporting, and design selection provides a more complete engineering workflow than studies that stop at

surrogate accuracy alone. Based on the CFD-derived thermo-hydraulic performance trends, the configuration ranking by THPP showed that the W-shaped arrangement performed best, followed by the V-shaped, M-shaped, arc-shaped, and inclined configurations under the investigated operating range, with the top-ranked case occurring at $P/D = 6$ and $Re = 8000$.

4.2. Best surrogate for Nu: hist gradient boosting

The best-performing Nu surrogate (HistGB) reaches $R^2 \approx 0.991$, indicating that the model explains nearly all variance in Nu within the grouped validation regime. Given the Nu range in the dataset (≈ 35.7 – 177.8), an $RMSE$ of 3.30 corresponds to only a small fraction of the operating span, supporting use as a high-fidelity surrogate for design-space exploration.

The OOF parity plot for HistGB (Fig. 14) provides a direct visualisation of calibration quality over the full Nu range, while the residual scatter and residual histogram (Fig. 15) diagnose systematic bias and error symmetry. Together, these diagnostics are essential for surrogate use in optimisation: unbiased residuals and near-identity parity behaviour imply that the model can be used not only for ranking candidate designs but also for estimating absolute performance metrics with minimal distortion.

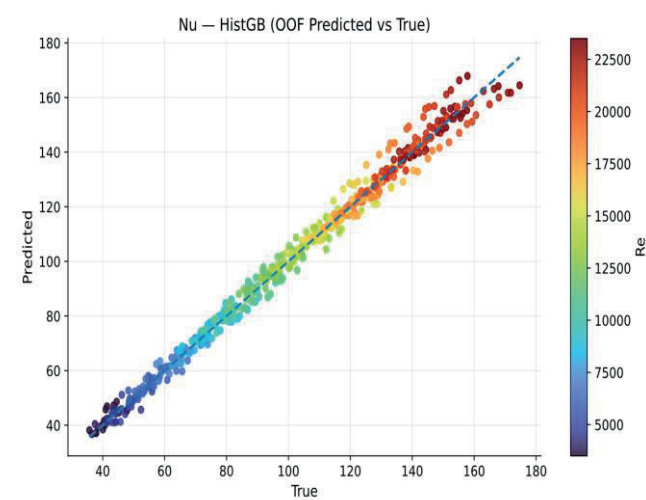


Fig. 14. OOF predicted vs. true Nu for HistGB (colour-coded by Reynolds number).

That boosting domination for Nu is physically plausible: heat transfer enhancement in roughened SAHs is governed by coupled, non-linear effects of turbulence augmentation, reattachment and secondary flow structures that vary with Reynolds number and turbulator arrangement. Even when Reynolds number is transformed (log, power laws) and combined with pitch-related interaction features, the mapping to Nu remains non-additive, making tree-based boosting an appropriate function class.

The frozen-holdout results confirm that the strong grouped cross-validation performance is retained when the model is evaluated on unseen geometry groups. The narrow bootstrap interval around holdout $RMSE$ indicates that the reported Nu accuracy is not driven by a favourable single split. This supports the use of the Nu surrogate for rapid design screening within the investigated Reynolds-number and pitch-ratio ranges.

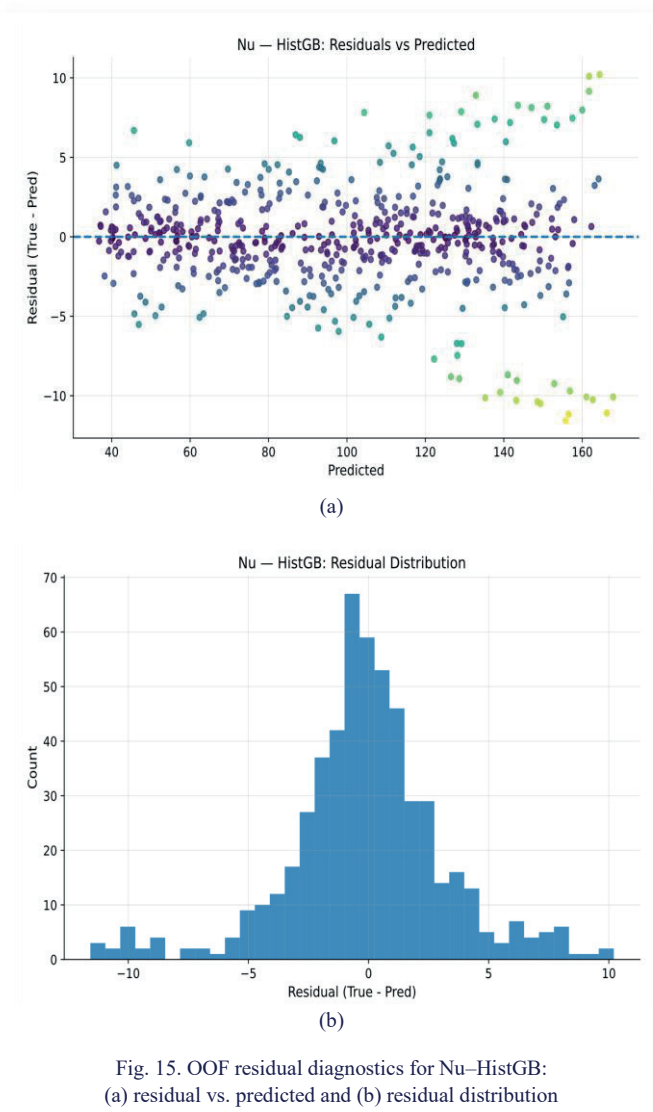


Fig. 15. OOF residual diagnostics for Nu-HistGB: (a) residual vs. predicted and (b) residual distribution

4.3. Best surrogate for friction factor: ridge regression

For friction factor, ridge regression yields the strongest performance ($MAPE \approx 1.12\%$; $R^2 \approx 0.989$). This result is particularly valuable because the friction factor directly drives pumping power and thus the hydraulic penalty that must be traded off against heat transfer enhancement in SAH design.

Within the present operating envelope, the friction factor is typically dominated by smooth trends with respect to the Reynolds number and discrete roughness/pitch levels. After feature engineering (e.g. $1/Re$, $\ln(Re)$, $\sqrt{Re}$ and pitch-related transforms), these trends become approximately linear in the engineered space. Ridge regression then provides (i) strong generalisation under grouped splitting and (ii) stable estimates that are less sensitive to small dataset idiosyncrasies than more flexible models. This stability is advantageous when the surrogate will be embedded in optimisation loops.

Thermo-hydraulic performance metrics explicitly couple Nu and friction factor; for example, the widely used THPP formulation compares thermal gain against frictional loss using Nu and f . Therefore, high-accuracy prediction of f (along with Nu) is not merely a statistical achievement – it directly improves the reliability of multi-objective optimisation (maximise heat trans-

fer while limiting pressure drop) and reduces the risk of selecting designs that appear optimal due to surrogate bias.

The frozen-holdout error and its bootstrap confidence interval further show that the friction-factor surrogate remains stable on unseen geometry groups. This reliability is important because even small bias in friction-factor prediction can distort thermo-hydraulic ranking. The holdout uncertainty bounds therefore strengthen confidence in using the surrogate for THPP-based screening and trade-off analysis.

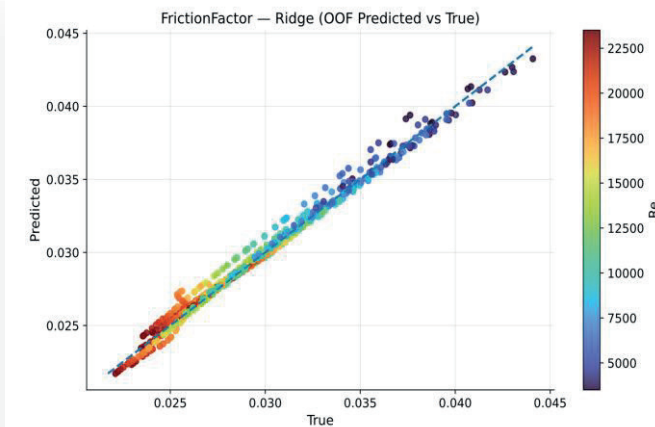


Fig. 16. OOF predicted vs. true friction factor for ridge regression (colour-coded by Reynolds number).

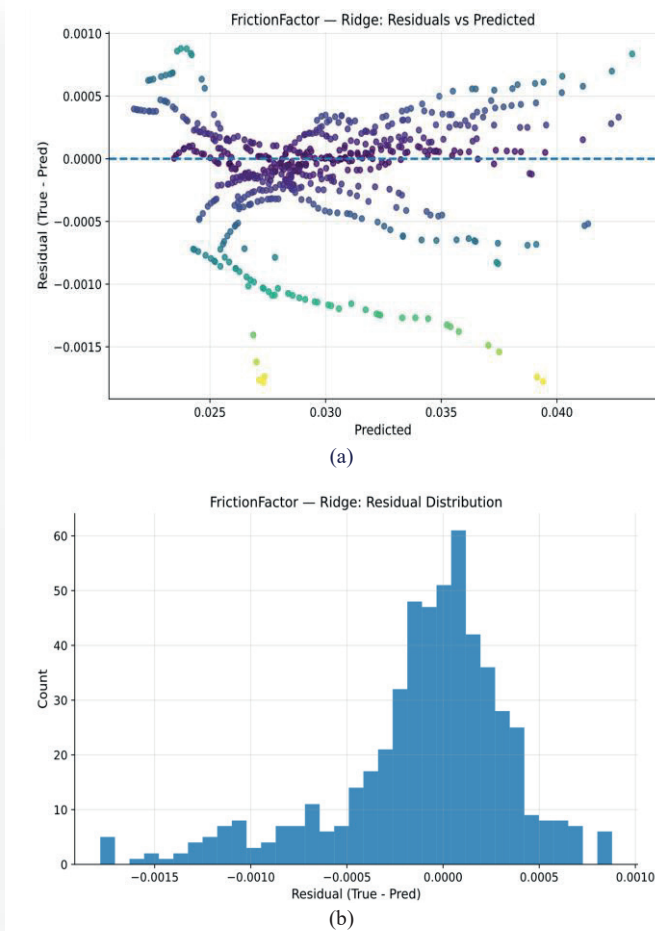


Fig. 17. OOF residual diagnostics for friction factor – ridge regression: (a) residual vs. predicted and (b) residual distribution.

4.4. Discussion of underperforming deep MLP baseline

The torch MLP benchmark underperformed substantially for both Nu and friction factor, despite using standardised numeric inputs and early stopping. In small-to-moderate tabular datasets with heterogeneous engineered features and discrete categorical structure (configuration families and pitch levels), deep MLPs often require careful architecture and regularisation tuning (and sometimes embedding strategies for categorical inputs) to match the inductive bias of tree ensembles. In contrast, boosting and linear models can be more data-efficient and robust for this class of structured regression problems, which is consistent with the performance ranking observed in Table 6.

4.5. Practical implications for SAH design and optimisation

From a domain perspective, these surrogates enable near-instant evaluation of Nu and f across the validated operating envelope ($Re = 3500$ – $23\,500$; pitch ratio 3–9), replacing repeated CFD calls during design screening. Because both Nu and f are predicted with high accuracy under leakage-safe grouped testing, the surrogates are suitable for:

- rapid parametric sweeps across Re and turbulator arrangements without rerunning CFD,
- multi-objective optimisation where Nu improvement must be balanced against friction penalties via THPP-style metrics,
- sensitivity studies to identify which geometric and operating factors most strongly shift the heat transfer-pressure drop trade-off.

To convert the thermo-hydraulic analysis into a design recommendation, the investigated spherical-turbulator layouts were ranked using the thermo-hydraulic performance parameter (THPP). Among all tested cases, the highest THPP was obtained for the W-shaped arrangement at $P/D = 6$ and $Re = 8000$, with a THPP of 2.14. This case provided the most favourable balance between heat-transfer enhancement and friction penalty. In contrast, the V-shaped configuration produced the highest heat-transfer augmentation, but its larger friction loss reduced the overall thermo-hydraulic advantage. Likewise, the inclined configuration showed lower pressure drop but did not achieve the strongest overall performance because of its lower Nusselt-number gain.

Across the tested operating range, the W-shaped configuration remained the most favourable choice for thermo-hydraulic design, particularly in the range of $Re = 3500$ – $12\,000$ and at $P/D = 6$. Therefore, for practical SAH implementation, the W-shaped configuration with $P/D = 6$ is recommended when the objective is to maximise the overall thermo-hydraulic performance rather than heat transfer alone.

Furthermore, from a methodological standpoint, the present study extends existing CFD-ML work by linking surrogate modelling directly to design decision-making. Rather than treating the surrogate as only a fast predictor of Nu and friction factor, the framework uses validated and uncertainty-supported predictions to rank configuration performance and identify the most

favourable spherical-turbulator arrangement. This design-oriented use of the surrogate is a central contribution of the study.

5. Conclusions

This study developed and validated a combined CFD-ML framework to evaluate and rapidly predict the thermo-hydraulic behaviour of a single-pass solar air heater (SAH) equipped with spherical turbulators arranged in five patterns (V-, M-, W-shaped, inclined, and arc-shaped) over three pitch ratios ($P/D = 3, 6, 9$) and a Reynolds-number range of 3500–23 500. The principal conclusions are:

1. The CFD formulation and numerical settings yielded predictions that agreed well with established smooth-duct heat-transfer and friction correlations, and also showed acceptable deviation relative to benchmark roughened-duct results. This validation supports the use of the CFD database as a high-fidelity reference for surrogate training and configuration screening.
2. The compiled dataset captures physically consistent behaviour: Nu increases with the Reynolds number, while the friction factor decreases with the Reynolds number, and both responses exhibit clear dependence on turbulator arrangement and pitch ratio. These configuration-dependent offsets confirm that layout is a first-order driver of SAH performance.
3. By using geometry-aware grouping (configuration $\times$ pitch) and grouped cross-validation, the ML workflow explicitly targets generalisation across geometry identities, not merely interpolation among near-duplicate samples. This provides a more realistic estimate of surrogate robustness for design-space exploration.
4. Thermo-hydraulic ranking based on THPP identified the W-shaped configuration as the best overall spherical-turbulator layout within the investigated design space. The most favourable case occurred at $P/D = 6$ and $Re = 8000$, where the configuration achieved the best balance between heat-transfer enhancement and friction penalty.
5. Although the V-shaped configuration delivered strong heat-transfer enhancement, its higher friction loss reduced its overall thermo-hydraulic benefit. Similarly, the inclined configuration offered lower hydraulic resistance but a weaker overall thermal advantage. This confirms that configuration selection should be based on THPP rather than Nu or friction factor alone.
6. For practical SAH design within the investigated Reynolds-number range of 3500–23 500 and pitch ratios of 3, 6 and 9, the W-shaped configuration with $P/D = 6$ is the recommended design choice for overall performance, while the trained surrogates can be used for rapid screening of similar operating conditions without repeated CFD simulations.

The present models are valid within the investigated configuration set, pitch ratios and Reynolds-number range. The added frozen holdout-test results and bootstrap-based uncertainty bounds confirm that the selected surrogates retain strong predictive reliability on unseen geometry groups within this design space. Future work should expand the database to additional

sphere diameters and geometric degrees of freedom and integrate the trained surrogates with formal multi-objective optimisers to generate robust design maps for practical SAH deployment.

Acknowledgements

The research has been duly facilitated by the Department of Mechanical Engineering Graphic Era Deemed to be University, Dehradun, Uttarakhand. The authors sincerely acknowledge the support of K.R. Mangalam University’s Central Instrumentation Facility in enabling and assisting with this research.

Data and code availability

The CFD-derived dataset, feature-definition sheet and the Python scripts used for preprocessing, grouped cross-validation, frozen-holdout testing and surrogate-model training will be made available in a public repository upon acceptance of the manuscript. The repository will include the final tabular dataset, variable dictionary, train-test grouping protocol and scripts required to reproduce the reported results.

References

- [1] Uniyal, A., Prajapati, Y.K., Ranakoti, L., Bhandari, P., Singh, T., Gangil, B., Sharma, S., Upadhyay, V.V., & Eldin, S.M. (2022). Recent advancements in evacuated tube solar water heaters: A critical review of the integration of phase change materials and nanofluids with ETCs. *Energies*, 15, 8999. doi: 10.3390/en15238999
- [2] Kumar, A., Karn, A., McGregor, C., & Singh, V.P. (2025). A critical analysis of paradigm shifts from domestic solar cooking to institutional solar cooking technologies. *Renewable and Sustainable Energy Reviews*, 218, 115840. doi: 10.1016/j.rser.2025.115840
- [3] Al-Yasiri, Q., Szabó, M., & Arıcı, M. (2022). A review on solar-powered cooling and air-conditioning systems for building applications. *Energy Reports*, 8, 2888–2907. doi: 10.1016/j.egyr.2022.01.172
- [4] Negi, A., Ranakoti, L., Verma, R.P., Kumar, V., Bhandari, P., Khargotra, R., & Singh, T. (2025). Enhancing solar still productivity with organic phase change materials: A literature review. *Energy Conversion and Management: X*, 26, 100984. doi: 10.1016/j.ecmx.2025.100984
- [5] Bisht, V.S., Patil, A.K., & Gupta, A. (2018). Review and performance evaluation of roughened solar air heaters. *Renewable and Sustainable Energy Reviews*, 81, 954–977. doi: 10.1016/j.rser.2017.08.036
- [6] Saini, R.P., & Singal, S.K. (2007). A review on roughness geometry used in solar air heaters. *Solar Energy*, 81(11), 1340–1350. doi: 10.1016/j.solener.2007.01.017
- [7] Chamoli, S., Thakur, N.S., & Saini, J.S. (2012). A review of turbulence promoters used in solar thermal systems. *Renewable and Sustainable Energy Reviews*, 16, 3154–3175. doi: 10.1016/j.rser.2012.01.021
- [8] Gawande, V.B., Dhoble, A.S., Zodpe, D.B., & Chamoli, S. (2016). A review of CFD methodology used in literature for predicting the thermo-hydraulic performance of a roughened solar air heater. *Renewable and Sustainable Energy Reviews*, 54, 550–605. doi: 10.1016/j.rser.2015.10.025
- [9] Chaube, A., Sahoo, P.K., & Solanki, S.C. (2006). Analysis of heat transfer augmentation and flow characteristics due to rib rough-

- ness over absorber plate of a solar air heater. *Renewable Energy*, 31, 317–331. doi: 10.1016/j.renene.2005.01.012
- [10] Kumar, S., & Saini, R.P. (2009). CFD based performance analysis of a solar air heater duct provided with artificial roughness. *Renewable Energy*, 34, 1285–1291. doi: 10.1016/j.renene.2008.09.015
- [11] Yadav, A.S., & Bhagoria, J.L. (2013). CFD based heat transfer and fluid flow analysis of a solar air heater provided with circular transverse wire rib roughness. *Energy*, 55, 1127–1142. doi: 10.1016/j.energy.2013.03.066
- [12] Al-Chlahiawi, K., Alyas, B.H., & Abdullah, B.N. (2024). Optimizing heat transfer and flow characteristics within a roughened solar air heater duct with compound turbulators. *Asia-Pacific Journal of Chemical Engineering*, 19(5), e3126. doi: 10.1002/apj.3126
- [13] Manjunath, M.S., Karanth, K.V., & Sharma, N.Y. (2017). Numerical analysis of spherical turbulence generators on heat transfer enhancement of flat plate solar air heater. *Energy*, 121, 616–630. doi: 10.1016/j.energy.2017.01.032
- [14] Maithani, R., Silori, A., Rana, J., & Chamoli, S. (2017). Numerical analysis of heat transfer and fluid flow of wavy delta winglets in a rectangular duct. *Thermal Science and Engineering Progress*, 2, 15–25. doi: 10.1016/j.tsep.2017.04.002
- [15] Manjunath, M.S., Karanth, K.V., & Sharma, N.Y. (2018). Numerical investigation on heat transfer enhancement using sinusoidal corrugations. *International Journal of Mechanical Sciences*, 138, 219–228. doi: 10.1016/j.ijmecsci.2018.01.037
- [16] Manjunath, M.S., Karanth, K.V., & Sharma, N.Y. (2019). Numerical Analysis of Flat Plate Solar Air Heater Integrated With an Array of Pin Fins on Absorber Plate for Enhancement in Thermal Performance. *Journal of Solar Energy Engineering*, 141(5), 051008. doi: 10.1115/1.4043517
- [17] Antony, A.L., Shetty, S.P., Madhwesh, N., Sharma, N.Y., & Karanth, K.V. (2020). Influence of stepped cylindrical turbulence generators on the thermal enhancement factor of a flat plate solar air heater. *Solar Energy*, 198, 295–310. doi: 10.1016/j.solener.2020.01.065
- [18] Arunkumar, H.S., Kumar, S., & Karanth, K.V. (2020). Analysis of a solar air heater for augmented thermohydraulic performance using helicoidal spring shaped fins-A numerical study. *Renewable Energy*, 160, 297–311. doi: 10.1016/j.renene.2020.06.098
- [19] Taslim, M.E., Li, T., & Kercher, D.M. (1996). Experimental Heat Transfer and Friction in Channels Roughened With Angled, V-Shaped, and Discrete Ribs on Two Opposite Walls. *Journal of Turbomachinery*, 118(1), 20–28. doi: 10.1115/1.2836602
- [20] Olsson, C.O., & Sunden, B. (1998). Thermal and Hydraulic Performance of a Rectangular Duct With Multiple V-Shaped Ribs. *Journal of Turbomachinery*, 120(4), 1072–1077. doi: 10.1115/1.2825892
- [21] Momin, A.M.E., Saini, J.S., & Solanki, S.C. (2002). Heat transfer and friction in solar air heater duct with V-shaped ribs. *International Journal of Heat and Mass Transfer*, 45(16), 3383–3396. doi: 10.1016/S0017-9310(02)00046-7
- [22] Lanjewar, A., Bhagoria, J.L., & Sarviya, R.M. (2011). Heat transfer and friction in W-shaped rib roughness. *Energy*, 36(7), 4531–4541. doi: 10.1016/j.energy.2011.03.054
- [23] Ghildyal, A., Bisht, V.S., Rawat, K.S., & Bhandari, P. (2023). Effect of D-shaped turbulators on performance. *Archives of Thermodynamics*, 44(2), 3–20. doi: 10.24425/ather.2023.146556
- [24] Ghildyal, A., Bisht, V.S., Bhandari, P., Thapliyal, S., Kaushik, S., Ranakoti, L., Bangari, R.S., Srivastav, A., Kanojia, N., & Raj Paul, A. (2024). Comparative numerical evaluation of solar air heater. *Archives of Thermodynamics*, 45(4), 189–196. doi: 10.24425/ather.2024.152008

- [25] Bohra, J., Bisht, V.S., Bhandari, P., Rawat, K.S., Singh, J., Kumar, K., & Rawat, B. (2022). Effect of blockage height ratio. *Materials Today: Proceedings*, 69, 153–157. doi: 10.1016/j.matpr.2022.08.279
- [26] Chaudhari, M., Sharma, S.L., & Debbarma, A. (2023). Exergetic performance analysis of solar air heater with L-shaped ribs as roughness element. *Archives of Thermodynamics*, 44(3), 241–267. doi: 10.24425/ather.2023.147546
- [27] Kumar, S., Bisht, V.S., Bhandari, P., Ranakoti, L., Negi, A., Bist, A.S., & Padalia, D. (2025). Computational analysis of modified solar air heater. *International Journal on Interactive Design and Manufacturing*, 19(5), 3467–3478. doi: 10.1007/s12008-024-01972-2
- [28] Alhilali, H.A.A., Al-Chlahiawi, K., Zarzoor, A.K., & Flayyih, M.A. (2025). Optimization of thermohydraulic parameters in solar air heater ducts featuring I-shaped ribs. *Heat Transfer*, 54(6), 3790–3814. doi: 10.1002/htj.23384
- [29] Chtourou, S., Djemel, H., Kaffel, M., & Baccar, M. (2021). Predicting rib pitch effect in plate heat exchangers. *SN Applied Sciences*, 3, 497. doi: 10.1007/s42452-021-04473-z
- [30] Chtourou, S., Djemel, H., Kaffel, M., & Baccar, M. (2022). Numerical investigation for the optimal plate heat exchangers Thermo-hydraulic efficiency fitted with fragmental and fractal y-shaped obstacles. *Computational Thermal Sciences: An International Journal*, 14(3), 29–57. doi: 10.1615/computthermalsci.2022039597
- [31] Dhamudia, K., & Senapati, J.R. (2025). A computational study for enhancement in thermal performance of plus(+) shaped ribbed solar air heater duct. *International Communications in Heat and Mass Transfer*, 161, 108411. doi: 10.1016/j.icheatmasstransfer.2024.108411
- [32] Dhamudia, K., & Senapati, J.R. (2025). Thermo-fluid performance with hook-shaped promoters. *International Communications in Heat and Mass Transfer*, 169, 109538. doi: 10.1016/j.icheatmasstransfer.2025.109538
- [33] Varun, Saini, R.P., & Singal, S.K. (2007). A review on roughness geometry used in solar air heaters. *Solar Energy*, 81(11), 1340–1350. doi: 10.1016/j.solener.2007.01.017
- [34] Gupta, D., Solanki, S.C., & Saini, J.S. (1997). Thermohydraulic performance of solar air heaters with roughened absorber plates. *Solar Energy*, 61(1), 33–42. doi: 10.1016/S0038-092X(97)00005-4
- [35] Tanda, G. (2004). Heat transfer in rectangular channels with transverse and V-shaped broken ribs. *International Journal of Heat and Mass Transfer*, 47(2), 229–243. doi: 10.1016/S0017-9310(03)00414-9
- [36] Karwa, R. (2003). Experimental studies of augmented heat transfer and friction in asymmetrically heated rectangular ducts with ribs on the heated wall in transverse, inclined, V-continuous and V-discrete pattern. *International Communications in Heat and Mass Transfer*, 30(2), 241–250. doi: 10.1016/S0735-1933(03)00035-6
- [37] Ghritlahre, H.K., Chandrakar, P., & Ahmad, A. (2020). Application of ANN model to predict the performance of solar air heater using relevant input parameters. *Sustainable Energy Technologies and Assessments*, 40, 100764. doi: 10.1016/j.seta.2020.100764.
- [38] Bhattacharyya, S., Sarkar, D., Roy, R., Chakraborty, S., Goel, V., & Almatrafi, E. (2021). Application of New Artificial Neural Network to Predict Heat Transfer and Thermal Performance of a Solar Air-Heater Tube. *Sustainability*, 13(13), 7477. doi: 10.3390/su13137477.
- [39] Ghritlahre, H.K., & Prasad, R.K. (2018). Exergetic performance prediction of solar air heater using MLP, GRNN and RBF models

of artificial neural network technique. *Journal of Environmental Management*, 223, 566–575. doi: 10.1016/j.jenvman.2018.06.033.

[40] Alatawi, E.S. (2025). Computational fluid dynamics analysis and machine learning study of heat transfer in solar air heaters with distinct ribs configuration. *Case Studies in Thermal Engineering*, 73, 106559. doi: 10.1016/j.csite.2025.106559.

[41] Alhuyi Nazari, M., Mukhtar, A., Yasir, A.S.H., Rashidi, M.M., Ahmadi, M.H., Blazek, V., Prokop, L., & Misak, S. (2023). Applications of intelligent methods in solar heaters: an updated review. *Engineering Applications of Computational Fluid Mechanics*, 17(1), 2229882. doi: 10.1080/19942060.2023.2229882.

[42] Bisht, V.S., Singh, D.B., Kumar, P., Jain, S., Bhandari, P., Mumtaz, M.A., & Rahman, M.H. (2025). Innovative spherical turbulators configurations for enhanced solar air heater performance: A numerical and analytical study. *Energy*, 342, 139601. doi: 10.1016/j.energy.2025.139601

[43] Tu, J., Yeoh, G.H., & Liu, C. (2019). *Computational fluid dynamics* (3rd ed.). Butterworth-Heinemann.

[44] Chamoli, S., Lu, R., Xu, D., & Yu, P. (2018). Thermal performance improvement using winglet generators. *Solar Energy*, 159, 966–983. doi: 10.1016/j.solener.2017.11.046

[45] Incropera, F.P., & DeWitt, D.P. (2002). *Fundamentals of heat and mass transfer*. John Wiley & Sons.

[46] Kakac, S., Shah, R.K., & Aung, W. (1987). *Handbook of single-phase convective heat transfer*. Wiley, New York.

[47] Lewis, M.J. (1975). Optimising thermohydraulic performance of rough surfaces. *International Journal of Heat and Mass Transfer*, 18(11), 1243–1248. doi: 10.1016/0017-9310(75)90232-X

[48] Maithani, R., & Saini, J.S. (2016). Heat transfer and friction correlations for V-rib roughness. *Experimental Thermal and Fluid Science*, 70, 220–227. doi: 10.1016/j.expthermflusci.2015.09.010



Co-published by
Institute of Fluid-Flow Machinery
Polish Academy of Sciences
Committee on Thermodynamics and Combustion
Polish Academy of Sciences

Copyright©2026 by the Authors under licence CC BY-NC-ND 4.0

<http://www.imp.gda.pl/archives-of-thermodynamics/>



**ARCHIVES
OF THERMODYNAMICS**

Preliminary zero-dimensional thermodynamic analysis of a gas turbine using EBSILON Professional software

Kamil Bokisz^a, Adam Ruziewicz^{a*}, Maciej Cholewiński^b

^aDepartment of Thermodynamics and Renewable Energy Sources, Faculty of Mechanical and Power Engineering, Wrocław University of Science and Technology, Wybrzeże Wyspiańskiego 27, 50-370 Wrocław, Poland
^bDepartment of Cryogenics and Aerospace Engineering, Faculty of Mechanical and Power Engineering, Wrocław University of Science and Technology, Wybrzeże Wyspiańskiego 27, 50-370 Wrocław, Poland
*Corresponding author email: adam.ruziewicz@pwr.edu.pl

Received: 06.11.2025; revised: 02.03.2026; accepted: 02.04.2026

Abstract

The application of zero-dimensional mathematical modelling offers significant advantages in the domain of energy systems. Firstly, it serves as a crucial support tool for the design process. Secondly, it facilitates the optimisation of operating parameters, thereby enhancing the efficiency of energy systems. Furthermore, it offers, depending on the accuracy of a model, a comprehensive analysis of the impact of various factors on the performance of a specific system. The present article focuses on a model of a gas turbine unit that has been developed using the EBSILON Professional software. The model was employed to assess the operating parameters that are characteristic of gas turbines, including the turbine inlet temperature and compression ratio, that are crucial for the efficiency and unit power of the abovementioned power system. Moreover, this approach enables the determination of the optimal operating points for the considered assumptions. Next, for the selected case, an analysis was conducted to determine the impact of changes in atmospheric air parameters on the operation of the gas turbine, applying two different approaches. Research has demonstrated that daily, and particularly seasonal, fluctuations in temperature can exert a substantial influence on the efficiency and capacity of the system, while fluctuations in pressure and relative humidity have a much smaller impact on turbine operation. Finally, the simplified model was utilised to ascertain the effect of incorporating hydrogen into fuel on the operational parameters of a gas turbine, indicating the effect of both quantitative and qualitative changes of stream in the combustion chamber and turbine.

Keywords: Gas turbine; Thermodynamic analysis; EBSILON Professional; Zero-dimensional analysis; Numerical modelling

Vol. 47(2026), No. 2, 119–130; doi: 10.24425/ather.2026.158678

Cite this manuscript as: Bokisz, K., Ruziewicz, A., & Cholewiński, M. (2026). Preliminary zero-dimensional thermodynamic analysis of a gas turbine using EBSILON Professional software. *Archives of Thermodynamics*, 47(2), 119–130.

1. Introduction

The ongoing development of natural gas-based energy is driven by several factors [1]. These include the need to diversify generation sources, reduce direct industrial emissions, and reorganise the geopolitical landscape of existing technology parks [2]. Consequently, gas installations are implemented to generate usable forms of energy, such as electricity and heat. These systems are designed for the high-efficiency conversion of chemical energy from fuels and energy carriers in a gaseous state into mechanical or electrical energy, on a given scale and with a given

degree of system complexity (including combined cycles [3–6]). These primarily include gas turbines, which are energy systems that enable the implementation of a real cycle similar to the Joule-Brayton cycle [7,8].

The simplified gas turbine unit consists of an inlet, compressor, combustion chamber, gas turbine and an exhaust system [4,9]. Its thermal efficiency is not far from that of other energy systems, such as piston engines and the Rankine steam cycle. However, they also have several advantageous features, including ecological benefits (the combustion of natural gas results in lower emissions of carbon dioxide and sulphur oxides per unit

Nomenclature

h – specific enthalpy, kJ/kg
 LCV – low calorific value, MJ/kg
 $\dot{m}$ – mass flow, kg/s
 p – pressure, Pa
 P – power, kW
 TIT – turbine inlet temperature, °C
 t – temperature, °C

Greek symbols

ε – relative pressure drop, %
 η – efficiency, %
 κ – heat capacity ratio
 π – compression ratio
 φ – air relative humidity, %

Subscripts and Superscripts

a – air

C – compressor
 el – electrical
 f – fuel
 g – flue gas
 G – electric generator
 GT – gas turbine (turbogenerator)
 m – mechanical
 n – polytropic
 s – isentropic
 T – turbine
 u – per unit

Abbreviations and Acronyms

CCGT– combined cycle gas turbines
 LCV – low calorific value
 TIT – turbine inlet temperature
 UAV – unmanned aerial vehicle
 UPS – uninterruptible power supplies

of chemical energy introduced into the combustion chamber), operational benefits (quick start-up), logistical benefits (transportation, fuel availability, and extraction costs), and technological benefits (the ability to create cascade installations, such as combined cycle gas turbines (CCGT), with efficiencies above 60%, as well as high development potential). Consequently, an increasing number of countries worldwide, including Poland (as evidenced by the PEP2040 document [2]), have chosen to prioritise research and implementation projects focusing on technological resources and fuel reserves in favour of gas carriers based on methane and other hydrocarbons.

Additionally, there is a growing interest in micro gas turbines, defined as systems dedicated to small stationary installations with a capacity of several hundred kW. These systems are most often coupled with electric generators, which are used in uninterruptible power supplies (UPS) and so-called SKI-Dach systems. Micro gas turbines also include mobile units, such as military vehicles, including the M1 Abrams tank family, which is equipped with Avco-Lycoming AGT1500 turboshaft multi-fuel engines. Other mobile units include guided missiles and unmanned aerial vehicles (UAVs), which burn aviation kerosene-type fuel [10].

Hydrogen, regarded as the fuel of the future, possesses technological potential related to combustion in gas turbines (both stationary and mobile, including aircraft). Dedicated combustion chambers, turbines and exhaust systems, along with solutions that allow for satisfactory temperatures (strength and thermal issues of expansion devices) and composition (environmental issues) of exhaust gases, have led to its current consideration as a component of the machinery park of many countries in the second half of the 21st century, similar to fuel cells. However, the technical implementation of hydrogen comes with challenges related to production, storage and use, including in gas turbines.

The research domain of every development process is becoming increasingly dependent on the availability and quality of computer-aided tools. The second of the aforementioned features is increasingly identified not only with the distinctive convergence of results generated by a given program or solver, but

also with their satisfactory level in terms of the time needed to perform them and the computer resources used. Consequently, a wide array of models, ranging from complex 2D and 3D numerical ones to their one-dimensional or dimensionless counterparts, have become integral to research endeavours. These models provide a foundation for decision-making processes and the utilisation of advanced tools. Additionally, they play a crucial role in the education and training of engineering and technical personnel, as well as in the construction of computer models for parametric diagnostics.

In recent years, as a result of the aforementioned progressive development and popularisation of various classes of computer-aided design tools and ongoing control of energy systems, there have been many numerical analyses of gas systems and power machines and devices that consider gas-fuel-based systems, implementing real cycles identified with gas-steam units [3] or aviation turbine power units [9,11,12]. They have been part of various research and development initiatives, focusing on a selected group of physical quantities and issues related to the ongoing operation of technical systems. This confirms their important role in the broadly understood decision-making system. A review of the extant literature reveals the existence of commercial tools that facilitate the aforementioned identification work in both stationary and aviation contexts. These include programs such as EngineSim (manufacturer or distributor – NASA Glenn Research Center) [13], ASTRA (Samara National Research University) [14], DVIGwT (Ufa State Aviation Technical University) [14], GasTurb (J. Kurzke, MTU) [15], GSP (Royal Netherlands Aerospace Centre) [16], Aspen HYSYS [6,17], and Uni_TTF (A. Lyulka Experimental Design Bureau) [14].

Nevertheless, despite its undeniable and steadily growing popularity, it remains advisable to repeat previously published numerical studies and verify the theses put forward with their help [6–9,11,12,18–22]. This is both because of the multi-threaded but often limited nature of the objects described by them in the course of computer tests and because of the simplifications used in many programs. For instance, existing literature continues to employ estimations that disregard the moisture content of the ambient air introduced into the flow machine dur-

ing the estimation process. Additionally, these analyses often underestimate the influence of climatic characteristics on the resulting operating parameters, treating them as negligible factors.

One of the software programs currently available that addresses the aforementioned issues is EBSILON Professional. It has been identified by numerous independent researchers as an interesting and practical tool for simulating and evaluating the operation of power systems, including those with a high degree of technological complexity and non-linear operating characteristics of the power machines and equipment included in it [23,24]. The software is characterised by its graphical interface, which has been in development since the early 1990s. It is equipped with a range of function codes dedicated to simulations of various systems, including gas and gas-steam units [25]. The model under consideration is 'based on a structured model of connections and time-operational dependencies. This provides a realistic basis for identifying a number of optimisation and research directions for the modelled gas turbine. In addition, it offers insight into the specifics of the energy conversion chain and its accompanying technical implementations.

In this article, the EBSILON Professional program was implemented to conduct preliminary analyses of the selection of operating parameters (combustion chamber outlet temperature, compression) for a model stationary gas turbine. The impact of environmental conditions (ambient temperature and pressure) on the indicators obtained by the associated energy system was also investigated, including cycle efficiency, gross unit power, and the air flow necessary to feed the turbine compressor. Additionally, the impact of enriching conventional fuel (methane) with hydrogen is presented using a previously created parameterised gas turbine model. The results obtained were subsequently subjected to a comprehensive evaluation, encompassing both comments and an assessment of their applicability in terms of the simplifications adopted in the software.

2. Mathematical model of the gas turbine

As illustrated in Fig. 1, the considered gas turbine unit consists of a compressor, a combustion chamber, a turbine and an electric generator. The flows of individual gases: air, fuel and flue gas,

are marked with separate colours, while the numbers indicate characteristic points in the system where thermodynamic parameters are determined. The computational model of the system for its individual components is presented in the following subsections.

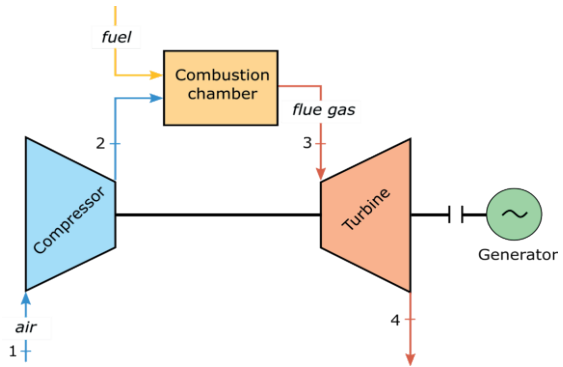


Fig. 1. Scheme of the modelled gas turbine.

The EBSILON Professional software facilitates the execution of numerical simulations of a variety of predefined thermodynamic processes and cycles. In addition, it enables the construction of user-defined models of energy systems, which can be employed for the purpose of thermodynamic analysis. In the latter case, the development of a model principally consists of the appropriate selection and combination of components available in the program and the determination of preliminary parameters and assumptions for the model. The calculations are based on the principle of mass and energy conservation, among other considerations, and are performed iteratively. These calculations are informed by the relationships that result from the thermochemical parameters of fluids. During the computational process, the program notifies the user of its convergence status. The mass and energy balance equations employed, along with the assumptions underlying the individual elements of the system, will be presented below.

The modelled gas turbine comprises four fundamental functional elements available in the program, which were combined into a single system, as illustrated in Fig. 2.

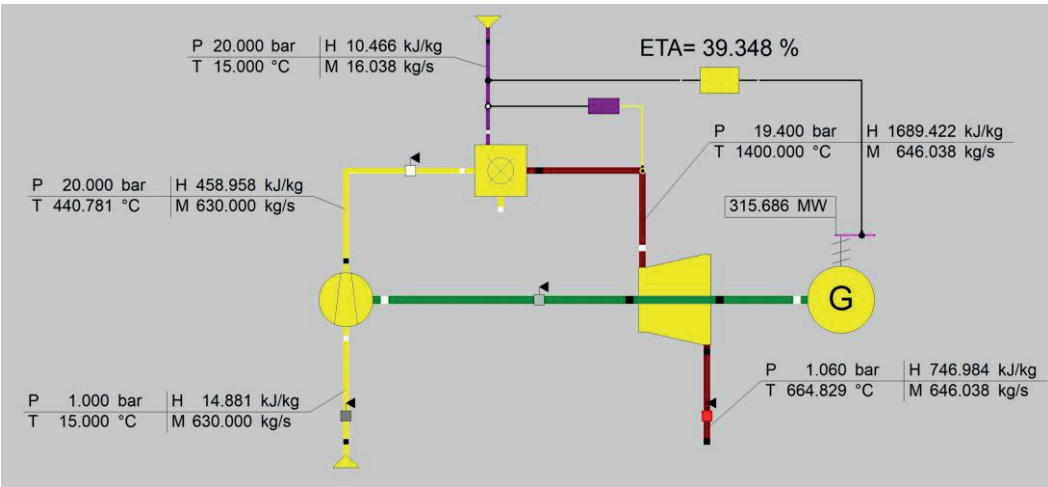


Fig. 2. Scheme of the gas turbine model implemented in the EBSILON Professional program with operational parameters.

Each component was assigned specific input parameters, which constituted the boundary conditions for the entire model. In the case of air parameters at the compressor inlet, standardised parameters for gas turbines according to ISO-2314 were utilised. The remaining parameters were assumed, and their values were mostly based on the actual parameters of similar facilities. Pure methane, the primary component of methane-rich natural gas (type E according to the PN-C-04750:2011 standard), was presumed to be the fuel utilised during the combustion process within the gas turbine. Furthermore, in order to execute a more extensive evaluation of the gas turbine's performance under diverse operating conditions, the predetermined turbine inlet temperature (TIT) was achieved through the automatic adjustment of the air or the fuel mass flow rate supplied to the combustion chamber alternatively. The program employs a dedicated control component for this purpose (forcing the implementation of the set change function). A comparative analysis was conducted on the results obtained from the implementation of both methodologies.

The following subsections contain a detailed description of the equations and the input values implemented in the program to solve the entire analysed system. The mathematical model is based on the balance equations of individual machines and devices forming the gas turbine.

2.1. Air compressor

The initial component of the modelled gas turbine is the compressor. According to the calculation code employed in the program, the variables necessary to determine the specifics of the process taking place in the compressor are considered to be:

- 1) air inlet (static) temperature t_1 ,
- 2) air inlet (static) pressure p_1 ,
- 3) air inlet relative humidity φ ,
- 4) air outlet (static) pressure p_2 ,
- 5) polytropic efficiency of the compressor $\eta_{n,C}$,
- 6) mechanical efficiency of the compressor η_m ,
- 7) air mass flow $\dot{m}_a$.

Assuming no bleed flows and a leak-tight compressor, the calculations presume a constant air mass flow throughout its entirety, as depicted by (control cross-sections are delineated in Fig. 1):

$$\dot{m}_a = \dot{m}_1 = \dot{m}_2. \quad (1)$$

In addition, under the assumption that compression occurs without heat exchange with the environment (adiabatically) and that the change in kinetic energy of the flow is negligible within the scope of the analyses conducted, the energy balance of the compressor is calculated using the following equation:

$$\dot{m}_a h_1 + P_c = \dot{m}_a h_2, \quad (2)$$

where P_c denotes the power supplied to the working fluid in the compressor, and $h_i = f(t_i, p_i)$ denotes the specific enthalpy of the stream determined in accordance with the fluid properties libraries adopted in the program.

The procedure for determining the parameters behind the compressor (in point 2) necessitates the calculation of the isen-

tropic compression efficiency. In the case of the analysed program, this efficiency is calculated using the following formula:

$$\eta_{s,C} = \frac{\pi^{\frac{\kappa_a - 1}{\kappa_a}} - 1}{\pi^{\kappa_a \eta_{n,C}} - 1}. \quad (3)$$

In order to determine it, it is therefore necessary to assume a directly correlated polytropic efficiency of compression $\eta_{n,C}$, specific heat ratio for air κ_a , and to identify the static compression ratio resulting directly from the assumed values of static pressure before and behind the compressor:

$$\pi = \frac{p_2}{p_1}. \quad (4)$$

Accordingly, the specific enthalpy of the air behind the compressor is determined by the following formula:

$$h_2 = h_1 - \Delta h_{12}, \quad (5)$$

where Δh_{12} is the actual increase in specific enthalpy, taking into account isentropic efficiency, determined from the following relationship:

$$\Delta h_{12} = \frac{h_{2s} - h_1}{\eta_{s,C}}, \quad (6)$$

where $h_{2s} = f(s_1, p_2)$ is the enthalpy after the compressor during isentropic compression. Knowing the specific enthalpy for the non-isentropic case, it is possible to determine the air temperature behind the compressor $t_2 = f(h_2, p_2)$, and the power supplied to the air flow in the compressor (Eq. 2). After taking into account the mechanical efficiency of the machine, the value of the mechanical power required to drive the compressor is obtained with:

$$P_{m,C} = \frac{P_c}{\eta_m}. \quad (7)$$

2.2. Combustion chamber

The second component of a gas turbine system, situated between the compressor and the turbine, forming a single rotor assembly, is the combustion chamber. This device facilitates the controlled oxidation of the supplied fuel in a compressed air atmosphere. The following assumptions were made in order to conduct this analysis:

- Combustion is considered to be complete;
- The entire air flow from the compressor goes to the combustion chamber (there are no air leaks between the two components of the system);
- The fuel is pure methane, and its lower calorific value (LCV), specified in the program's physico-chemical tables, is 50.015 MJ/kg.

In the calculation of the combustion chamber balance, it is essential to ascertain values of the following quantities:

- 1) fuel mass flow $\dot{m}_f$,
- 2) fuel pressure – assumed to be: $p_f = p_2$,
- 3) relative pressure drop in the combustion chamber ε .

In accordance with the principle of energy conservation, the mass flow of exhaust gases leaving the combustion chamber is determined using the following formula:

$$\dot{m}_g = \dot{m}_a + \dot{m}_f. \quad (8)$$

Given an assumed fuel flow rate, the gas parameters behind the combustion chamber can be determined from the energy balance equation:

$$\dot{m}_a h_2 + \dot{m}_f (h_f + LCV) = \dot{m}_g h_3, \quad (9)$$

where $h_f = f(t_f, p_f)$ is the specific physical enthalpy of the fuel.

Knowing the enthalpy of the exhaust gas determined from Eq. (9), its temperature $t_3 = f(p_3, h_3)$ can be calculated. A pressure drop in the combustion chamber is assumed according to the formula:

$$p_3 = p_2 (1 - \varepsilon). \quad (10)$$

The temperature of the exhaust gases behind the combustion chamber is described in the literature as the turbine inlet temperature (TIT), and will be presented in Section 3 using this acronym. This is a parameter of significant importance, as it directly influences the operational dynamics of a gas turbine and, by extension, the constraints imposed on materials and designs.

2.3. Gas turbine

The next component is the turbine (expander), which is the machine responsible for driving both the compressor and the generator by extracting enthalpy from the stream and converting it into effective work. In addition to the values calculated for the previous components, the model of the turbine requires determining the following quantities:

- 1) p_4 – outlet pressure,
- 2) $\dot{m}_g$ – flue gas mass flow,
- 3) $\eta_{s,T}$ – isentropic efficiency of the turbine,
- 4) η_m – mechanical efficiency of the turbine.

According to the principle of conservation of mass, the flow rate through the turbine remains constant:

$$\dot{m}_g = \text{const}. \quad (11)$$

Assuming adiabatic expansion, the energy balance of the turbine itself is as follows:

$$\dot{m}_g h_3 = \dot{m}_g h_4 + P_T, \quad (12)$$

where P_T is the theoretical (effective) power of the turbine, and h_4 is the specific enthalpy of the exhaust gas, given by the formula:

$$h_4 = h_3 - \Delta h_{34}, \quad (13)$$

where Δh_{34} is the actual enthalpy drop, taking into account isentropic efficiency, determined from the formula:

$$\Delta h_{34} = \frac{h_3 - h_{4s}}{\eta_{s,T}}, \quad (14)$$

where $h_{4s} = f(s_3, p_4)$ is the enthalpy behind the turbine, which assumes isentropic expansion. Knowing the actual specific enthalpy, similarly to the case of a compressor, it is possible to determine the temperature of the stream (exhaust gases) behind the turbine $t_4 = f(h_4, p_4)$ and the theoretical power of this ma-

chine from the energy balance, Eq. (12). After taking into account the mechanical efficiency of the turbine, the value of the mechanical power output to the shaft and directed to the compressor is obtained as:

$$P_{m,T} = P_T \eta_m. \quad (15)$$

2.4. Electric generator

In a gas turbine, the mechanical power obtained during expansion is utilised for two primary functions: to drive the compressor and to power the electric generator. Therefore, given the specified generator efficiency η_G , the electrical power is calculated using the following formula:

$$P_{el} = P_{m,GT} \eta_G, \quad (16)$$

where $P_{m,GT}$ is the net mechanical power of the gas turbine unit given by the formula:

$$P_{m,GT} = P_{m,T} - P_{m,C}. \quad (17)$$

The total efficiency of the gas turbine can ultimately be expressed as:

$$\eta_{GT} = \frac{P_{el}}{LCV \cdot \dot{m}_f}. \quad (18)$$

2.5. Model validation

In order to verify the mathematical model of the gas turbine, it is necessary to refer to real data. In this regard, actual data from a GE 9HA.01 turbine, referring to the ISO-2314 standard [26], were implemented. The input data for the model are presented in Table 1.

Table 1. Operational parameters of the gas turbine 9HA.01 representing input variables for the model validation.

Parameter	Symbol	Unit	Value
Temperature of the air at the inlet to the compressor	t_1	°C	15
Pressure of the air at the inlet to the compressor	p_1	bar	1.014
Relative humidity ratio	φ	%	60
Pressure of the air at the outlet of the compressor	p_2	bar	23.81
Pressure at the outlet of the turbine	p_4	bar	1.027
Compressor polytropic efficiency	$\eta_{n,C}$	%	93.8
Turbine isentropic efficiency	$\eta_{s,T}$	%	90
Electric generator efficiency	η_G	%	98.9
Mechanical efficiency of the compressor and the turbine	η_m	%	99
Pressure drop in combustion chamber	Δp_{23}	bar	1.19
Temperature of the fuel	t_f	°C	25
Pressure of the fuel	p_f	bar	36.3
Fuel mass flow	$\dot{m}_f$	kg/s	20.9
Temperature of the gas at the outlet of turbine	t_4	°C	628.9
Air mass flow	$\dot{m}_a$	kg/s	832.7

In order to compare the data declared by the producer with the theoretical results of the model, the values of the achieved electrical power of the gas turbine and the turbine inlet temper-

ature are compared in Table 2. The calculated results for the implemented actual parameters of the selected turbine are nearly identical to the data declared by the producer, with a relative error of approximately 0.1%. In consideration of the aforementioned factors, it can be concluded that, although certain assumptions and simplifications have been made, the model accurately reflects the actual process occurring within the machine.

Table 2. Comparison between the specifications of the gas turbine 9HA.01 and the model results.

Parameter	Symbol	Unit	Specifications	Model result
Electrical power	P_{el}	MW	446.0	445.6
Turbine inlet temperature	t_3	°C	1412.2	1410.3

3. Results and discussion

The mathematical model of a gas turbine presented in Section 2 was utilised to conduct a series of simulations of the operation of the aforementioned power system. The simulations analysed the impact of variables such as the compression ratio, exhaust gas temperature at the combustion chamber outlet, ambient pressure and relative humidity of the ambient air inlet to the compressor on the obtained operating parameters.

In the course of this study, the so-called base parameters (references) of the system were initially established, thereby serving as a reference point for further analysis and comparison. The values in question were derived from empirical data concerning actual turbines, including the 9HA.01 model from General Electric, and were also assumed on an intuitive basis. Subsequently, to execute fundamental simulations and ascertain the impact of selected variables, one or two parameters were modified while the values of the remaining parameters were maintained constant.

The input parameters at the base point are presented in Table 3, while the calculation results for the adopted parameters and assumptions of the modelled gas turbine are presented in Table 4.

Table 3. Input variables for the model reference point.

Parameter	Symbol	Unit	Value
Temperature of the air at the inlet to the compressor	t_1	°C	15
Pressure of the air at the inlet to the compressor	p_1	bar	1
Relative humidity ratio	φ	%	60
Turbine inlet temperature (TIT)	t_3	°C	1400
Pressure at the outlet of the turbine	p_4	bar	$1.06p_1$
Compressor polytropic efficiency	$\eta_{n,c}$	%	92
Turbine isentropic efficiency	$\eta_{s,T}$	%	90
Electric generator efficiency	η_G	%	98.6
Mechanical efficiency of the compressor and the turbine	η_m	%	99
Relative pressure drop in combustion chamber	ε	%	3
Temperature of the fuel	t_f	°C	15
Pressure of the fuel	p_f	bar	20
Fuel mass flow	$\dot{m}_f$	kg/s	16.038

Table 4. Calculated parameters for the model reference point.

Parameter	Symbol	Unit	Value
Specific enthalpy of the air at the inlet of the compressor	h_1	kJ/kg	14.9
Air mass flow	$\dot{m}_a$	kg/s	630
Specific enthalpy of the air at the outlet of the compressor	h_2	kJ/kg	459.0
Temperature of the air at the outlet of the compressor	t_2	°C	440.7
Flue gas mass flow	$\dot{m}_g$	kg/s	646
Pressure of the flue gas at the inlet of the turbine	p_3	bar	19.4
Specific enthalpy of the flue gas at the inlet of the turbine	h_3	kJ/kg	1689.4
Temperature at the outlet of the turbine	t_4	°C	664.8
Specific enthalpy at the outlet of the turbine	h_4	kJ/kg	747.0
Electrical power of the gas turbine	P_{el}	MW	315.7
Total efficiency of the gas turbine	η_{GT}	%	39.35

In the base model, the gas turbine attained an electrical power of 315.7 MW with an electrical efficiency of 39.35%, for a compression ratio of 20 and a turbine inlet temperature of 1400°C. The subsequent subsections delineate the impact of these final two parameters on the performance of the gas turbine.

3.1. The influence of compression ratio on TIT and efficiency

In order to examine the impact of compression ratio and turbine inlet temperature on the efficiency of a gas turbine, simulations were carried out for TIT values ranging from 1200°C to 1600°C with increments of 100°C, and for various compression ratios ranging from 10 to 80. The results of this study are presented in the graph shown in Fig. 3.

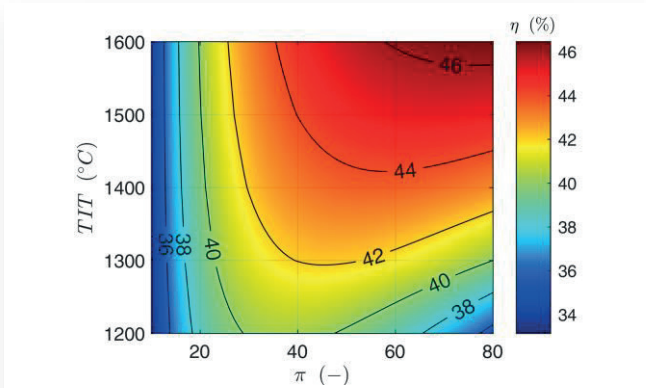


Fig. 3. Dependence of gas turbine efficiency on compression ratio and turbine inlet temperature (TIT).

As illustrated in Fig. 3, the efficiency of the gas turbine cycle is shown to vary depending on the compression ratio and the turbine inlet temperature (TIT). Consequently, it is feasible to ascertain the maximum efficiency of the gas turbine unit for the specified parameters. It is evident that efficiency increases with the temperature at the turbine inlet, an effect that arises from raising the temperature of the heat source. This increase in efficiency can reach up to 46% for a temperature of 1600°C. Fur-

thermore, for each TIT, there is a specified compression ratio that guarantees maximum efficiency at the given operating parameters. It is observed that, at lower temperatures, the turbine attains maximum efficiency at a compression ratio of 40–50, a range that is consistent with axial compressors comprising multiple subsonic stages. Additionally, the TIT has been observed to directly correlate with the optimal compression ratio, with values approaching 80 at temperatures exceeding 1500°C. It should be noted that the current gas turbines are incompatible with such high compression ratios, as depicted in Fig. 3. In practice, values below 50–60 or even lower for heavy gas turbines are assumed. This is a consequence of the relatively flat characteristic curve depicted in Fig. 3, or the complexity involved in operating or constructing multi-stage compressors (particularly in the context of aviation). It is also difficult to maintain such a high temperature without a sophisticated cooling system for the turbine blades.

In order to comprehensively assess the performance of a model gas turbine system, it is advantageous to incorporate an additional parameter: unit power, defined as the ratio of the obtained electrical power to the air mass flow:

$$P_u = \frac{P_{el}}{\dot{m}_a}. \quad (19)$$

As illustrated in Fig. 4, the power characteristics exhibit a noticeable divergence from the efficiency characteristics. The unit power also increases with temperature, reaching values of over 600 kJ/kg in extreme cases. However, for the highest temperature, the maximum unit power is achieved for much lower compression ratios – in the range of 15–25. The optimal compression ratio is consistent across the entire range of assumed temperatures. This phenomenon has been extensively studied in the context of aircraft turbine engines, where the optimal compression ratio, which maximises specific thrust and consequently minimises unit air mass input per thrust output in the engine, differs considerably from the economic compression ratio, which enables the lowest possible specific fuel consumption, defined as the fuel consumed per unit of thrust generated.

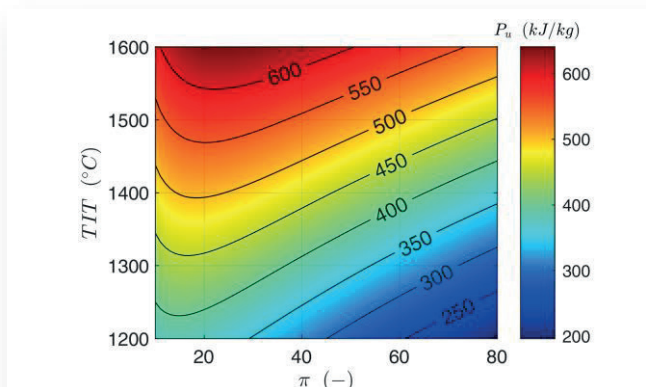


Fig. 4. Dependence of gas turbine unit power on compression ratio and turbine inlet temperature (TIT).

Considering the variations in power and efficiency characteristics, the subsequent step entailed the generation of a set of summary characteristic curves, incorporating the key parameters that characterise the operation of a gas turbine. These curves

are presented in the form of a graph in Fig. 5. The method of presenting the collective impact of the aforementioned parameters is frequently employed in the context of gas turbines. It facilitates the selection of operating parameters for the gas turbine based on a systematic, integrated approach.

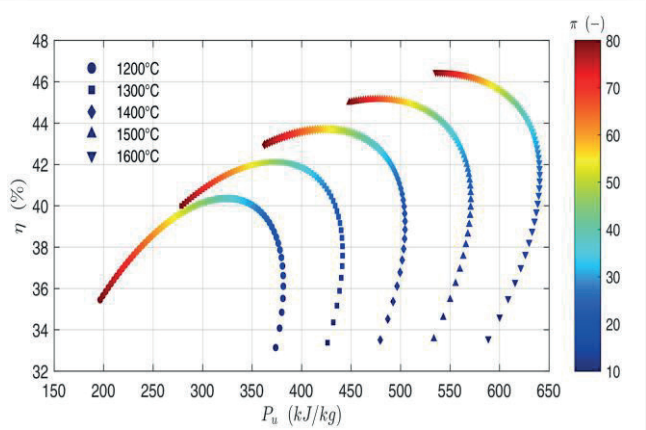


Fig. 5. Dependence of gas turbine efficiency on unit power for different values of compression ratio and turbine inlet temperature (TIT).

The graph presented in Fig. 5 shows the relationship between the efficiency of the gas turbine and the unit power in the form of curves determined for specific temperatures. The temperatures are marked with different shaped points, while the compression ratio scale is marked with a colour map. The graph shows the discrepancy between the maximum thermal efficiency and unit power, which are achieved for significantly different compression ratios.

Usually, the aim is to achieve the highest possible unit power, which, however, is far from the maximum efficiency of the gas turbine. Therefore, when designing such a system, a compromise must be found between maximum efficiency and maximum unit power. Using model characteristics may be helpful in this matter.

3.2. The influence of inlet air parameters on turbine performance

The gas turbine operating parameters presented in Section 3.1 and their dependence on design solutions can be controlled and influenced to a certain extent during the design or operation stage. In contrast, the parameters of the air at the gas turbine inlet (i.e. temperature, pressure and relative humidity) significantly affect operation and often result directly from atmospheric conditions (partially regulated from the air intake level). These parameters also clearly impact the operating parameters of single-shaft aircraft turbines, which are susceptible to changes within the so-called climatic characteristics.

Ambient air parameters may change in an uncontrolled manner depending on the season or the geographical location (climate zone) of the gas turbine unit. Therefore, it is important to understand how it behaves in different atmospheric conditions, which can be simulated using the aforementioned model.

To assess the impact of the environmental parameters, simulations were conducted for TIT = 1400°C and $\pi = 20$, as they

are considered representative within the scope of the analyses undertaken. The other input parameters, enumerated in Table 3, remained constant during the course of these simulations, with the exception of the fuel mass flux, which was constant in one case and variable in the other.

The results obtained, presented in Fig. 6–11, demonstrate the impact of successive atmospheric air parameters (static temperature at the compressor inlet, static pressure at the compressor inlet, and relative humidity at the compressor inlet) on the most important parameters characterising the operation of the gas turbine. In accordance with the established mathematical model, there are two distinct methodologies for ensuring a constant temperature at the turbine inlet. The primary strategy involves the regulation of the air mass flow supplied to the compressor, while maintaining a constant fuel mass flow rate into the combustion chamber. In the second, the fuel stream is selected automatically, and the air mass flux is maintained at a constant level. The second manner is more aligned with the established practices in real installations. However, a theoretical comparison of these two possibilities is worthwhile in the context of the consequences of different process control.

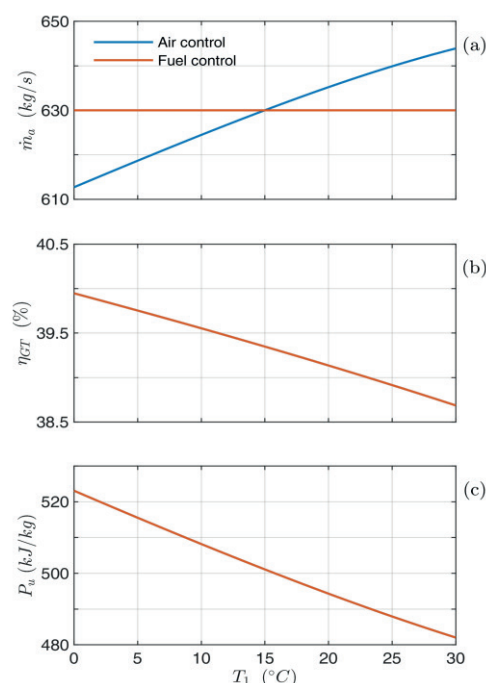


Fig. 6. Influence of the air temperature T_1 at the compressor inlet on: (a) mass flow of the inlet air $\dot{m}_a$, (b) total efficiency η_{GT} , (c) unit power P_u of the gas turbine.

Figure 6 illustrates the influence of the compressor inlet air temperature T_1 on the air mass flow $\dot{m}_a$, gas turbine total efficiency η_{GT} , and unit power P_u . A change in temperature from 0°C to 30°C was found to be accompanied by an almost linear decrease in efficiency, resulting in a reduction of nearly 1.5 percentage points (Fig. 6b), and a linear decrease in unit power of approximately 40 kJ/kg within the considered range of changes (Fig. 6c). Irrespective of the method selected for maintaining a constant TIT, the efficiency and unit power undergo analogous alterations. Consequently, differences are apparent in the indi-

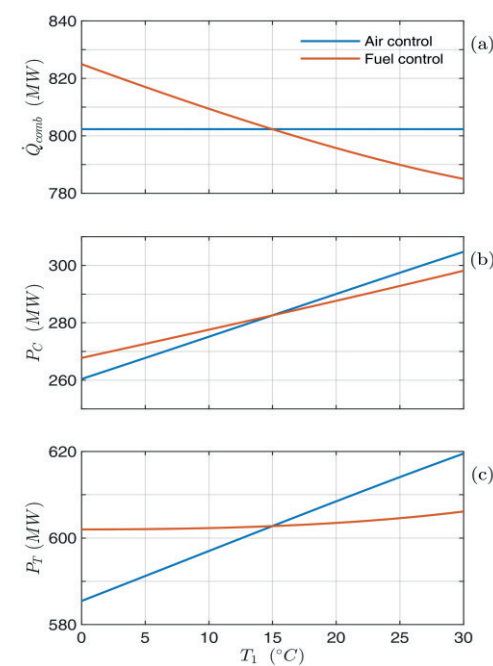


Fig. 7. Influence of the air temperature T_1 at the compressor inlet on: (a) heat flux supplied in the combustion chamber $\dot{Q}_{comb}$, (b) internal power of the compressor P_C , (c) internal power of the turbine P_T .

vidual powers of the combustion chamber, compressor and turbine (Fig. 7).

In the case of air control, and thus the constant fuel flow, the change of efficiency, according to Eq. (11), depends only on the change of the total electrical power of the gas turbine. The decrease in the total power can be explained by the decrease of the air density (due to the temperature change) and the increase of the mass flow of the inlet air to the compressor (Fig. 6a).

Although the decrease in density with temperature is self-evident, an explanation is necessary to understand why an increase in temperature at the compressor inlet leads to an increase in air mass flow. This is due to the assumption of a constant TIT and a constant fuel mass flow $\dot{m}_f$. The inlet air temperature t_1 , with the condition of fixed compression ratio ($\pi = 20$), causes an increase in temperature at the compressor outlet t_2 . This results in higher temperatures of the air supplied to the combustion chamber. Since the same amount of heat is obtained from fuel combustion, the air flow must be increased so that the flue gas temperature remains at the same assumed level.

The aforementioned factors result in the compressor requiring greater drive power (see Fig. 7b), with an increase of 17% for a 30-degree temperature growth. Concurrently, there is a slight (6%) increase in turbine power (see Fig. 7c), leading to an elevated back-work ratio and a reduction in the net power of the gas turbine.

In the second case, which pertains to fuel control, the air mass flow rate remains constant (see Fig. 6a). Consequently, an increase in air temperature leads to a reduction in the necessary heat input to the combustion chamber to maintain the desired TIT (see Fig. 7a). The increase in compressor power is analogous to the initial case, with the distinction that the air mass flow

rate remains constant. Given the air-fuel ratio of approximately 40, fluctuations in fuel mass flux result in a negligible alteration in exhaust gas production. Maintaining constant thermal conditions at the turbine inlet consequently yields a negligible alteration in turbine power. In conclusion, the discrepancy in gas turbine net power in comparison to the initial case is counterbalanced by maintaining a constant air mass flux. This results in the same characteristics of the unit power.

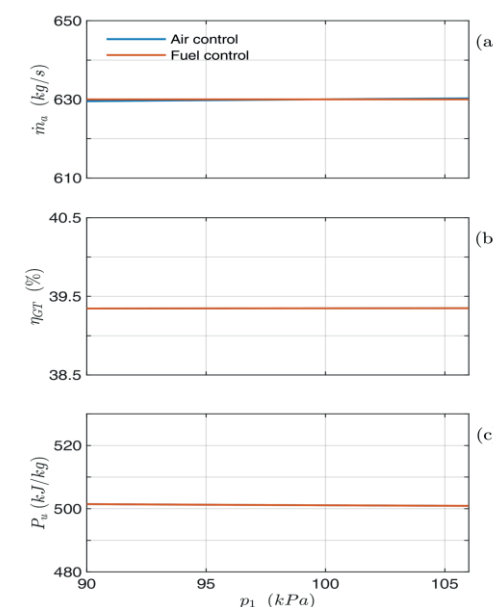


Fig. 8. Influence of the air pressure p_1 at the compressor inlet on: (a) mass flow of the inlet air $\dot{m}_a$, (b) total efficiency η_{GT} , (c) unit power of the gas turbine P_u .

As demonstrated in Fig. 8, the previously specified operational parameters exhibit a minor sensitivity to variations in air pressure at the compressor inlet. As illustrated in Fig. 8a, an increase in pressure p_1 at a constant assumed temperature t_1 results in a decrease in the air mass flow, which remains constant. Maintaining a constant compression ratio leads to a negligible change in temperature behind the compressor. The minor discrepancies can be explained by the variation in absolute pressure values, which consequently impacts the specific enthalpy. According to the implemented model, the specific enthalpy is a function of both temperature and pressure. However, for low pressures, this does not significantly affect enthalpy. Consequently, its value at characteristic points (h_2 behind the compressor and h_3 before the turbine) varies only slightly for different values of the inlet pressure. Therefore, the influence of the inlet pressure (which varies within the range of atmospheric air) on gas turbine efficiency and unit power is hardly noticeable.

As with variations in temperature, controlling the air or fuel mass flow rate so as to maintain a constant turbine inlet temperature (TIT) results in precisely the same characteristics of efficiency and the unit power of a gas turbine. A marginal discrepancy can be discerned in net power and, consequently, in the heat supply, compressor and turbine power (Fig. 9). Nevertheless, these deviations are primarily attributable to minor variations in air mass flux incorporated into the cycle.

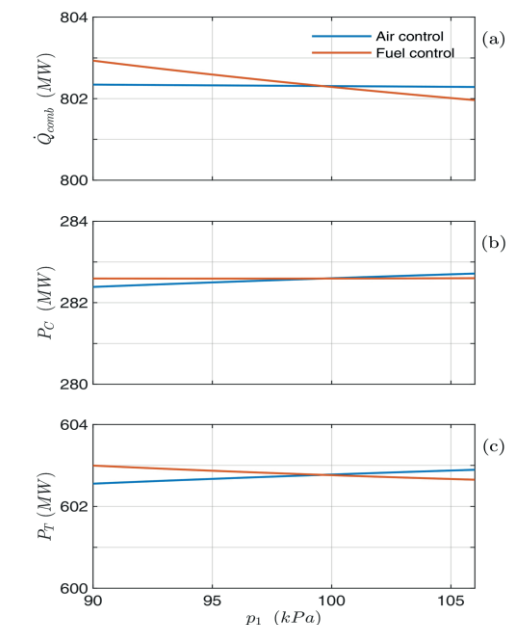


Fig. 9. Influence of the air pressure p_1 at the compressor inlet on: (a) heat flux supplied in the combustion chamber $\dot{Q}_{comb}$, (b) internal power of the compressor P_C , (c) internal power of the turbine P_T .

Another parameter analysed in terms of turbine operation is the relative humidity ϕ of the air at the compressor inlet. Its impact on the discussed parameters is shown in Figs. 10 and 11. A growth of the relative humidity slightly increases density, resulting in a decrease in air mass flow (Fig. 10a) in the case of air control. Conversely, the effect on enthalpy is more pronounced than the effect on pressure change. This results in a relatively modest yet substantial impact on gas turbine efficiency and unit power. The decline in efficiency is only marginally perceptible (Fig. 10b), and a slight rise in unit power (Fig. 10c) is evident.

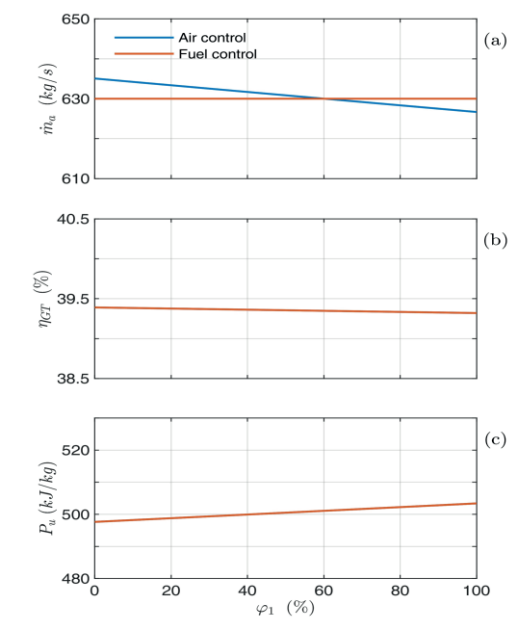


Fig. 10. Influence of the air relative humidity ϕ_1 at the compressor inlet on: (a) mass flow of the inlet air $\dot{m}_a$, (b) total efficiency η_{GT} , (c) unit power P_u of the gas turbine.

The discrepancies in the air control and fuel control cases are illustrated in Fig. 11. Given the identical thermal parameters at characteristic points, it is the air mass flux that is responsible for the observed discrepancies in compressor and turbine power.

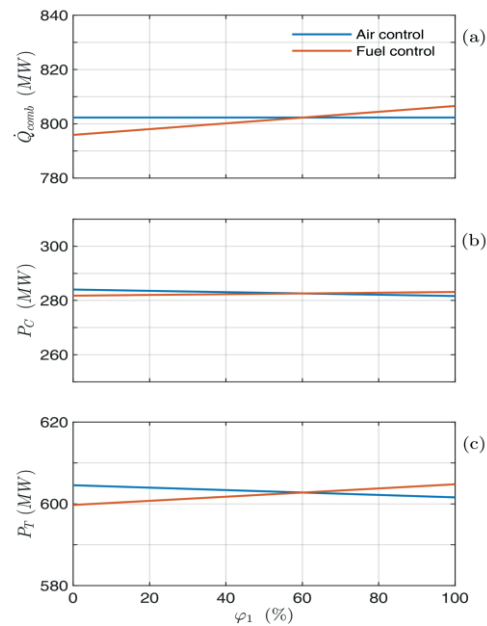


Fig. 11. Influence of the air relative humidity φ_1 at the compressor inlet on: (a) heat flux supplied in the combustion chamber $\dot{Q}_{comb}$, (b) internal power of the compressor P_C , (c) internal power of the turbine P_T .

The findings presented above indicate that, among the examined air parameters, the ambient temperature exerts the most significant influence on the turbine efficiency and unit power. The ambient pressure exerts a comparatively minor influence on the unit power fluctuations and the gas turbine efficiency. It is important to note, however, that the ambient pressure does not undergo as rapid changes during the day or over the course of several hours as the temperature does. Consequently, its impact may be discernible in extreme situations when the pressure deviates from its standard value. It has also been demonstrated that the relative humidity exerts a small but noticeable influence on the efficiency and unit power of a gas turbine.

3.3. The influence of hydrogen addition to the fuel on turbine performance

The created model was utilised to execute a preliminary investigation into the impact of enriching conventional fuel (methane) with hydrogen at varying mass percentages ranging from 0% to 100%. The simulation was performed with a constant air supply to the system, as illustrated in Table 3.

As demonstrated by the analyses, the rising proportion of hydrogen (Fig. 12a) in the fuel utilised in the modelled installation is accompanied by an increase in both the thermal power of the combustion chamber (heat supply) and the power of the turbine. This rise in power, coupled with the constant compressor power, results in an increase in the power output of the entire unit.

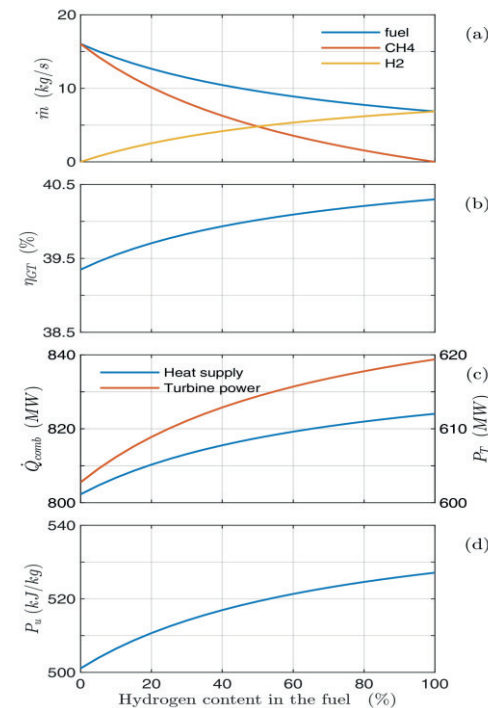


Fig. 12. Influence of the hydrogen content in fuel on: (a) mass flow of the fuel and its components, (b) total efficiency η_{GT} , (c) heat flux supplied to the combustion chamber and turbine power, (d) unit power of the gas turbine P_u .

As illustrated in Fig. 12c, the second increase exceeds the first in percentage terms. Consequently, the total efficiency of the gas turbine increases by less than 1 percentage point (see Fig. 12b). Additionally, the unit power increases by nearly 30 kJ/kg, due to its reference to a constant air flow supplied to the compressor.

It should be noted that the obtained results are strictly dependent on the simulation settings and model parameterisation. For instance, the aforementioned increase in thermal power of the combustion chamber stems from two factors: the higher calorific value of hydrogen compared to methane, and the increased proportion of water vapour in the exhaust gas stream, which raises its specific enthalpy.

An increase in the exhaust gas stream's specific enthalpy is necessary to reach the desired temperature at the turbine inlet. Consequently, more heat must be supplied, introducing more chemical energy into the cycle with the fuel. While the transition from pure methane to pure hydrogen combustion decreases the stream leaving the chamber by 9.182 kg/s (less than 1.5% of the initial mass flux), the increase in the exhaust gas stream's specific enthalpy is 58.54 kJ/kg (nearly 3.5% more). A similar situation occurs in the turbine. The increase in power is solely the result of the greater difference in exhaust gas enthalpy between the two isobars (assuming isentropic turbine efficiency). This difference is a direct result of the thermochemical properties of the gas mixtures formed in the combustion chamber. In the considered model, the pressure ratio, turbine inlet temperature, and the efficiency of the aforementioned machine are constant.

To enhance the representativeness of the results with respect to real objects, it would be necessary to consider factors such as

the combustion chamber efficiency (losses due to incomplete or partial combustion, pressure losses) or turbine efficiency as a function of the hydrogen content in the fuel. While these elements were not incorporated in the present simulations, the utilised tool, EBSILON Professional, enables the implementation of more advanced characteristics of the aforementioned machines and devices. Consequently, they remain viable options for zero-dimensional analyses of gas blocks, including those burning or co-burning alternative fuels.

4. Conclusions

The article presents a numerical model of a gas turbine, prepared using the EBSILON Professional software. This program was used to examine the impact of individual parameters (compression ratio, turbine inlet temperature and ambient air parameters) on the efficiency and the unit power of a gas turbine.

The EBSILON Professional program facilitates the modelling of real objects, including gas turbines, ensuring satisfactory accuracy and convergence with actual operating indicators. This is exemplified by the case of GE's 9HA.01 turbine. The software's functionality encompasses the optimisation of selected parameters, facilitating the observation of their impact on turbine performance. An illustration of this optimisation is provided in Fig. 5, a diagram that can be utilised to identify compression ratio values for which the gas turbine achieves a sufficiently high efficiency and the highest possible unit power.

The operation of a gas turbine is dependent on atmospheric conditions, including the temperature, pressure and relative humidity of the surrounding air. The atmospheric parameter that exerts the most influence on its operation is temperature. As demonstrated in Figs. 6–11, the efficiency and the unit power of a gas turbine are significantly affected by the air temperature, while changes in the pressure and the relative humidity are of lesser importance. Changes in ambient temperature are observed throughout the year, depending on seasonal conditions. Consequently, gas turbines demonstrate optimal efficiency during winter months, when ambient temperatures are sufficiently low. Conversely, gas turbines exhibit minimal efficiency during summer months, when temperatures frequently exceed 30°C, dependent upon the specific climate zone. However, it is important to note that the temperature of the inlet air should not become excessively low during operation, as this can lead to operational problems associated with condensation that can crystallise in the air intake. The presence of ice crystals on the compressor blades can damage this component of the gas turbine, leading to a forced shutdown. Therefore, in case of low inlet temperature (i.e. between –10°C and 10°C), the air supplied to the compressor is pre-heated to prevent this from happening. The impact of adding hydrogen to fuel on the operating parameters of a gas turbine was subsequently determined.

In the context of simulating actual gas turbine cycles, it is necessary to consider the material limitations of the components utilised and the atmospheric conditions characteristic of the climate zone in which the turbine is located. These factors impose greater restrictions that must be taken into account during the calculations.

It should be emphasised that the results obtained and presented in this paper are a direct consequence of the parameterisation of the numerical model described in Chapter 2 and referenced in Subsection 3.2. Although these results should be considered approximations of the actual state of the power system, the software allows for the inclusion of higher-order characteristics of individual machines and devices in the simulation, as well as more complex control systems.

References

- [1] Wasik, P., & Krzyślak, P. (2017). Method of modelling gas turbine performance. *Journal of Mechanical and Transport Engineering*, 69(1), 61–79. doi: 10.21008/j.2449-920X.2017.69.1.06
- [2] Ministry of Climate and Environment, Republic of Poland (2021). *Polityka energetyczna Polski do 2040 r.* <https://www.gov.pl/attachment/3209a8bb-d621-4d41-9140-53c4692e9ed8> [accessed 04 Nov. 2025] (in Polish).
- [3] Badyda, K., Harutyunyan, A., & Wołowicz, M. (2024). Repowering strategies: Analysis of gas turbine integration with steam cycle for enhanced power plant performance. *Archives of Thermodynamics*, 45(2), 99–106. doi: 10.24425/ather.2024.150856
- [4] Bartela, L., & Kotowicz, J. (2013). Analysis of operation of the gas turbine in a poligeneration combined cycle. *Archives of Thermodynamics*, 34(4), 137–159. doi: 10.2478/aoter-2013-0034
- [5] Cao, Y., & Dai, Y. (2017). Comparative analysis on off-design performance of a gas turbine and ORC combined cycle under different operation approaches. *Energy Conversion and Management*, 135, 84–100. doi: 10.1016/j.enconman.2016.12.072
- [6] Fernandez, D.A.P., Foliaco, B., Padilla, R.V., Bula, A., & Gonzalez-Quiroga, A. (2021). High ambient temperature effects on the performance of a gas turbine-based cogeneration system with supplementary fire in a tropical climate. *Case Studies in Thermal Engineering*, 26, 101206. doi: 10.1016/j.csite.2021.101206
- [7] Baakeem, S.S., Orfi, J., Alaqel, S., & Al-Ansary, H. (2017). Impact of Ambient Conditions of Arab Gulf Countries on the Performance of Gas Turbines Using Energy and Exergy Analysis. *Entropy*, 19(1), 32. doi: 10.3390/e19010032
- [8] De Sa, A., Al Zubaidy, S. (2011). Gas turbine performance at varying ambient temperature. *Applied Thermal Engineering*, 31(14–15), 2735–2739. doi: 10.1016/j.applthermaleng.2011.04.045
- [9] Cholewiński, M. (2024). Wstępne obliczenia termogazodynamiczne silników turbowentylatorowych z wykorzystaniem modeli zerowymiarowych. *XXI Konferencja Mechanika w Lotnictwie (ML-XXI 2024)*, 20–23 May, Kazimierz Dolny, Poland (in Polish).
- [10] Giampaolo, A. (2006). *Gas Turbine Handbook: Principles and Practices*. 3rd Edition. The Fairmont Press.
- [11] Cholewiński, M. (2023). Analiza wpływu stopnia dwuprzepływowości na wybrane parametry eksploatacyjne silników turbowentylatorowych. *XV Konferencja "Młodzi w Energetyce"*, 24–26 May, Wrocław, Poland (in Polish).
- [12] Chapman, J.W. *Practical Techniques for Modeling Gas Turbine Engine Performance*. NASA Glenn Research Center. <https://ntrs.nasa.gov/api/citations/20170000884/downloads/20170000884.pdf> [accessed 04 Nov. 2025].
- [13] Benson, T.J (1995). *An Interactive Educational Tool for Turbojet Engines*. NASA Lewis Research Center. <https://www.grc.nasa.gov/www/k-12/airplane/EngineTheory.pdf> [accessed 04 Nov. 2025].
- [14] Kuzmichev, V.S., Ostapyuk, Y., Tkachenko, A.Y., Krupenich, I.N., & Filinov, E.P. (2017). Comparative Analysis of the Com-

- puter-Aided Systems of Gas Turbine Engine Designing. *International Journal of Mechanical Engineering and Robotics Research*, 6(1), 28–35. doi: 10.18178/ijmerr.6.1.28-35
- [15] Gao, J.-y., & Huang, Y.-y. (2011). Modeling and Simulation of an Aero Turbojet Engine with GasTurb. *2011 International Conference on Intelligence Science and Information Engineering*, 20–21 August, Wuhan, China. doi: 10.1109/ISIE.2011.149
- [16] Visser, W.P.J., & Broomhead, M.J. (2000). *GSP. A generic object-oriented gas turbine simulation environment*. National Aerospace Laboratory NLR. <https://files.core.ac.uk/download/pdf/53034661.pdf> [accessed 04 Nov. 2025].
- [17] Tomków, Ł., & Cholewiński, M. (2015). Improvement of the LNG (liquid natural gas) regasification efficiency by utilizing the cold exergy with a coupled absorption – ORC (organic Rankine cycle). *Energy*, 87(C), 645–653. doi: 10.1016/j.energy.2015.05.041
- [18] Trawiński, P. (2020). Development of real gas model operating in gas turbine system in Python programming environment. *Archives of Thermodynamics*, 41(4), 23–61. doi: 10.24425/ather.2020.135853
- [19] Kotowicz, J., Job, M., Brzeczek, M., Nawrat, K., & Mędrych, J. (2016). The methodology of the gas turbine efficiency calculation. *Archives of Thermodynamics*, 37(4), 19–35. doi: 10.1515/aoter-2016-0025
- [20] Goucem, M. (2024). Influence of the Ambient Temperature on the Efficiency of Gas Turbines. *Fluid Dynamics and Materials Processing*, 20(20), 2265–2279. doi: 10.32604/fdmp.2024.052365
- [21] Razak, A.M.Y. (2013). Gas turbine performance modelling, analysis and optimisation. In *Modern Gas Turbine Systems. High Efficiency, Low Emission, Fuel Flexible Power Generation* (pp. 423–514). Woodhead Publishing Series in Energy.
- [22] Sammour, A.A., Komarov, O.V., Qasim, M.A., Almalghouj, S., Al Dakkak, A.M., & Du, Y. (2023). Ambient conditions impact on combined cycle gas turbine power plant performance. *Energy Sources, Part A: Recovery, Utilization, and Environmental Effects*, 45(1), 557–574. doi: 10.1080/15567036.2023.2172100
- [23] Swat, S. (2024). *Introduction to the simulation of power plants for EBSILON®Professional Version 16*. doi: 10.13140/RG.2.2.17864.33284
- [24] Madejski, P., & Żymelka, P. (2020). *Wprowadzenie do komputerowych obliczeń symulacji pracy systemów energetycznych w programie STEAG Ebsilon®Professional*. Wydawnictwa AGH (in Polish).
- [25] Bujalski, M., & Madejski, P. (2021). Modelowanie pracy układu turbiny gazowej z kotłem odzyskowym w programie Ebsilon Professional. *XIII Konferencja „Młodzi w Energetyce”*, 5–7 July, Wrocław, Poland (in Polish).
- [26] Gülen S.C. (2019). *Gas Turbine Combined Cycle Power Plants* (1st ed.). CRC Press.



Co-published by
Institute of Fluid-Flow Machinery
Polish Academy of Sciences
Committee on Thermodynamics and Combustion
Polish Academy of Sciences

Copyright©2026 by the Authors under licence CC BY-NC-ND 4.0

<http://www.imp.gda.pl/archives-of-thermodynamics/>



Non-Similar Analysis of MHD Ferro-Nanofluid Flow over an Exponential Stretching/Shrinking Sheet with Particle Effects

Md Sayeed Iftekhar Yousuf^a, Jahangir Alam^a, M.G. Murtaza^{b*}, Md Touhidul Islam^c,
Md Jashim Uddin^d, Tasnim Ul Islam^a, Md Fazley Elahi^a

^aDepartment of Computer Science and Mathematics, Bangladesh Agricultural University, Mymensingh-2202, Bangladesh

^bDepartment of Mathematics, Comilla University, Cumilla-3506, Bangladesh

^cDepartment of Irrigation and Water Management, Bangladesh Agricultural University, Mymensingh-2202, Bangladesh

^dDepartment of Applied Mathematics, Noakhali Science and Technology University, Noakhali-3814, Bangladesh

*Corresponding author email: murtaza@cou.ac.bd

Received: 22.12.2025; revised: 09.03.2026; accepted: 15.03.2026

Abstract

Biomagnetic nanofluids have attracted considerable attention because of their potential applications in biomedical engineering, particularly in drug delivery, cancer therapy, and thermal management systems. In the present study, a nonsimilar analysis of biomagnetic blood-based nanofluid containing multiwalled carbon nanotubes over an exponential stretching/shrinking sheet is investigated. The model incorporates several important physical effects, including a transverse magnetic field, viscous dissipation, suction/injection, particle volume fraction, particle radius, and particle spacing during nanoparticle–blood interaction. The governing partial differential equations are transformed using nonsimilarity transformations and solved numerically through the local nonsimilarity technique with the bvp4c solver in MATLAB. The effects of the physical parameters on velocity, temperature distribution, skin friction coefficient and local Nusselt number are analysed in detail. The results indicate that increasing the magnetic field parameter, particle radius and nanoparticle volume fraction significantly reduces the velocity profile while enhancing the temperature distribution. Moreover, an increase in particle spacing improves both velocity and temperature profiles. Quantitatively, the heat transfer rate decreases by approximately 23.46% and 8.83% in the stretching and shrinking cases, respectively, when the nanoparticle volume fraction increases from 0 to 0.1. Moreover, the skin friction coefficient increases by approximately 24.15% and 23.9% with the increasing magnetic field strength, while increasing the particle radius from 1 to 2 reduces the skin friction coefficient by approximately 16.52% and 33.18% in the stretching and shrinking cases, respectively. These findings provide useful insights for the design and optimisation of biomagnetic nanofluid systems in biomedical and thermal engineering applications.

Keywords: Biomagnetic fluid dynamics; Magnetohydrodynamics; Nanoparticles; Nonsimilarity; Particle radius

Vol. 47(2026), No. 2, 131–140; doi: 10.24425/ather.2026.158679

Cite this manuscript as: Yousuf, S.I.M., Alam, J., Murtaza, M.G., Islam, T.M., Uddin, J.M., Islam, T.U., & Elahi, F.M. (2026). Non-Similar Analysis of MHD Ferro-Nanofluid Flow over an Exponential Stretching/Shrinking Sheet with Particle Effects. *Archives of Thermodynamics*, 47(2), 131–140.

1. Introduction

The suspension of nanoparticles (usually ranging in size from 1 nm to 100 nm) in conventional fluids such as water, oils, blood or ethylene glycol is known as nanofluid, and it is a crucial field of research in fluid dynamics. These nanoparticles may significantly improve the fluid’s thermal conductivity and heat transfer

characteristics, making nanofluids ideal for a range of technical uses. In this context, a promising nexus between nanotechnology and biomedical research is blood-based nanofluid flow. Researchers want to improve drug delivery methods, provide novel diagnostic tools, and increase the usefulness of blood for therapeutic purposes by utilising the special qualities of nanoparticles. The integration of blood-based nanofluids in the biomed-

Nomenclature

B_0 – magnetic field strength, T
 C_p – heat capacity, J/(kg K)
 Ec – Eckert number
 h – particle interspacing, m
 M – magnetic field parameter
 Nu – Nusselt number
 Pr – Prandtl number
 R_p – particle radius, m
 Re – Reynolds number
 S – suction/injection parameter
 T – temperature, K
 u, v – velocity components along axial and normal directions, m/s

Greek symbols

η – pseudosimilarity variable
 θ – dimensionless temperature
 κ – thermal conductivity, W/(m K)

λ – stretching/shrinking parameter
 μ – dynamic viscosity, kg/(m s)
 ξ – nonsimilarity variable
 ρ – blood density, kg/m³
 σ – electrical conductivity, S/m
 ϕ – particle volume fraction

Subscripts and Superscripts

ηf – nanofluid
 w – wall
 ∞ – free-stream

Abbreviations and Acronyms

CNT – carbon nanotube
 HAM – homotopy analysis method
 MHD – magnetohydrodynamic
 MWCNT – multiwalled carbon nanotubes
 SWCNT – single-wall carbon nanotube

cal and engineering domains has intriguing prospects for improving performance and efficiency and tailoring therapeutics as research progresses.

Single-wall carbon nanotubes (SWCNTs) and multiwall carbon nanotubes (MWCNTs) are important components in the mathematical framework of blood-based hybrid nanofluid flow presented by Saeed et al. [1], which takes into account the electromagnetic field and coupling stress. The hybrid nanofluid model increased the heat transfer rate by 11.95%, according to previous studies. TiO₂+Ag/blood flow was numerically analysed by Alanazi et al. [2]. They discovered through computational use of the Runge–Kutta technique that the particle volume fraction significantly contributes to improved heat transmission. A blood-based Casson nanofluid flow that permits SWCNT and MWCNT particles over an extended sheet was covered by Mohamed et al. [3]. According to previous studies, compared with SWCNTs, MWCNTs significantly affect blood flow. The hybrid nanofluid stagnation fluid flow exposed to an expanding surface was investigated by Samat et al. [4] using carbon nanotubes. According to the study, magnetic fields are crucial for fluid transport behaviour, and whenever the magnetic field parameter is smaller than 1, several solutions to the problem may be found. Kerosene-based carbon nanotubes (CNTs) exhibit greater skin friction than do engine oil- and water-based CNTs, according to the mathematical model of homogeneous nanofluid flow of Khan et al. [5]. A MATLAB-based bvp4c-technique stagnation point flow of a CNT nanofluid along a curved surface was quantitatively examined by Khan et al. [6]. According to the study, when comparing SWCNTs to MWCNTs, the rate of heat transmission increased noticeably. Using water as the base fluid, Das et al. [7] investigated the effects of radiation and an external magnetic field on the unsteady flow of a three-dimensional hybrid nanofluid exposed to upward and downward rotating disks in the cases of SWCNT+Fe₃O₄ and MWCNT+Fe₃O₄. The numerical analysis reveals that, in comparison with that of SWCNT+Fe₃O₄, the rate of heat transmission for MWCNT+Fe₃O₄ is less than 0.35%.

A Levenberg–Marquardt multilayer artificial neural network was introduced by Priyadharshini and Archana [8] to investigate

SWCNT-MWCNT/H₂O flows. The authors discovered that larger values of the volume fraction result in an increase in the thermal profile of the fluid behaviour system as a whole. Rehman et al. [9] investigated the effects of slip circumstances and magnetic fields on blood/MWCNT behaviour while taking various fluid characteristics into account. Using the homotopy approach (homotopy analysis method – HAM), they reported that the Nusselt number increases in tandem with the volume friction, Eckert number and dynamic viscosity of the nanoparticles. Ramanuja et al. [10] investigated the effects of heat sources and thermal radiation on magnetohydrodynamic (MHD) blood-SWCNT flow and discovered that resistive impedance in the flow direction validates better findings in SWCNTs than in pure blood. A mathematical framework of TiO₂+Fe₃O₄+SiO₂/blood-based hybrid nanofluid flow subject to an extended curved surface was proposed by Nagaraja et al. [11]. The investigation revealed that the Prandtl number increases the system's total heat transfer rate. A thorough examination of SWCNTs/blood and MWCNTs/blood subjected to unstable Marangoni convection flow on a stretched surface was performed by Rehman et al. [12]. Increasing the stretching parameter values reduces fluid skin friction with the use of a computational process. A mathematical model of blood based CNT flows under Joule dissipation and viscous heating was developed by Nasir et al. [13]. They discovered that higher solid volume fractions for both types of CNTs increase the heat transfer rate to the blood vessel wall. El Ghil and Driouch [14] investigated the principles of electrically conducting water-based carbon nanotubes on an expanding sheet and discovered that MWCNTs had greater normal and tangential velocities than those of SWCNTs; the temperature distribution showed the opposite trend. In their discussion of MHD Casson ternary hybrid nanofluid slip flow over a surface expanded with convective heat, Mahanta and Sharma [15] reported that when the suction increases, the thickness of the boundary layer decreases.

Ahmad et al. [16] examined transient electroviscous flow between squeezing plates via a ternary hybrid nanofluid composed of silicon carbide, aluminium oxide, and multiwall carbon nanotubes dispersed in water. Their study demonstrated that the in-

corporation of multiple nanoparticles significantly enhances thermal transport compared with that of conventional nanofluids, particularly in the presence of thermal radiation and heat generation effects. Yasir et al. [17] investigated the radiative stagnation-point flow of cross fluid with gyrotactic microorganisms and highlighted the important role of the activation energy and heat source/sink mechanisms in modifying the thermal and concentration fields. Ahsan et al. [18] analysed stratified bioconvective Williamson nanofluid flow over a curved stretching surface under the influence of Joule heating, a magnetic field and nonlinear thermal radiation, and reported that the magnetic and Williamson parameters tend to suppress the velocity field, whereas the thermal parameters significantly alter the temperature distribution. Furthermore, Yasir et al. [19] conducted a comparative analysis of radiative magnetised flows of micropolar, Casson, and Maxwell fluids with melting heat transfer and demonstrated that thermal radiation markedly intensifies the temperature field, particularly in the Casson fluid. Kallem et al. [20] investigated magnetohydrodynamic hyperbolic tangent fluid flow over a nonlinear stretching sheet in a porous medium, highlighting the significant influence of magnetic field strength, chemical reactions and heat sources on flow and heat transfer characteristics. Biswas et al. [21] analysed thermomagnetic convection and entropy generation in a wavy porous cavity filled with a Cu-Al₂O₃/water hybrid nanofluid and reported that magnetic field intensity and wall undulations strongly affect hydrothermal transport behaviour. M and De [22] examined electroosmotic Maxwell nanofluid flow in a Darcy-Forchheimer porous medium considering the Soret and Dufour effects, and demonstrated that electromagnetic forces and cross-diffusion mechanisms play roles in controlling velocity, temperature and concentration fields.

Motivated by the above studies, the main objective of the present work is to investigate the nonsimilar flow and heat transfer characteristics of a biomagnetic blood-based nanofluid containing multiwalled carbon nanotubes (MWCNTs) over an exponentially stretching/shrinking sheet. The analysis considers the combined influence of the magnetic field parameters, nanoparticle volume fraction, particle radius, and particle spacing during nanoparticle–blood interactions. The novelty of this study lies in incorporating the effects of the particle radius and interparticle spacing within a nonsimilar biomagnetic nanofluid framework over an exponential stretching/shrinking surface. The governing nonlinear partial differential equations are solved via a local nonsimilarity technique with the MATLAB bvp4c solver. The effects of the governing parameters on the velocity, temperature, skin friction coefficient and local Nusselt number are examined in detail, providing useful insight into biomagnetic nanofluid transport relevant to biomedical and microscale thermal applications.

2. Mathematical formulation

A two-dimensional steady electrically conducting biomagnetic fluid with nanoparticles is considered along the exponentially stretching/shrinking sheet, as depicted in Fig. 1. For the current study, the following assumptions are taken into consideration:

- laminar flow,

- incompressible flow,
- MHD flow,
- blood is considered the base fluid,
- multiwalled carbon nanotubes (MWCNTs) are considered to have a spherical shape,
- slip conditions,
- nonsimilar flow,
- particle radius and interspacing is assumed, while interacting with blood.

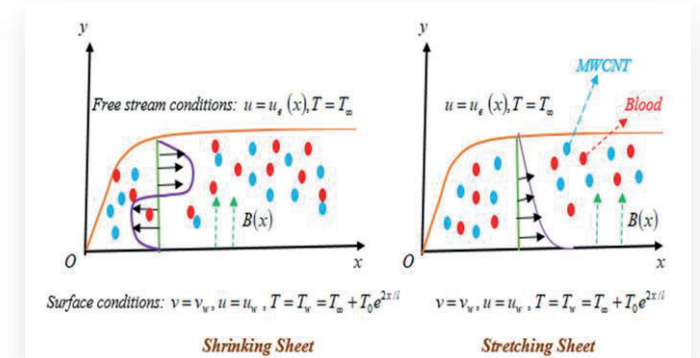


Fig. 1. Physical view of blood/MWCNT flow.

These assumptions define the physical framework of the present model and determine the range of applicability of the obtained results. The laminar and incompressible flow assumptions are appropriate for small-scale biomedical flow conditions where inertial and density variation effects are negligible. Therefore, the results obtained in this study apply to biomagnetic nanofluid transport systems, particularly in biomedical and microscale heat transfer applications where similar physical conditions exist.

Additionally, the sheet is considered to be stretched exponentially with velocity $u_w(x) = ce^{x/l}$ along the axial direction x , whereas the radial direction is denoted by y , where the stretching and shrinking cases are indicated by $c > 0$ and $c < 0$, respectively. The free stream velocity is represented by $u_e(x) = be^{x/l}$, where $b > 0$. Furthermore, the wall mass transmission is shown by $v_w(x) = v_0 e^{x/2l}$, in which mass suction and injection are indicated by $v_0 > 0$ and $v_0 < 0$, respectively. A strongly applied magnetic field in an exponential form is given by $B(x) = B_0 e^{x/2l}$, which acts perpendicularly to the surface. Considering all these assumptions, we extend the work of [15,23,24], and the proposed mathematical model is given by:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0, \quad (1)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = u_e \frac{du_e}{dx} + \frac{\mu_{nf}}{\rho_{nf}} \left(\frac{\partial^2 u}{\partial y^2} \right) - \frac{\sigma_{nf}}{\rho_{nf}} B_0^2(x)(u - u_e), \quad (2)$$

$$(\rho C_p)_{nf} \left(u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} \right) = \kappa_{nf} \left(\frac{\partial^2 T}{\partial y^2} \right) + \mu_{nf} \left(\frac{\partial u}{\partial y} \right)^2, \quad (3)$$

with boundary conditions [23,24]:

- at $y = 0$:

$$v = v_w, \quad u = u_w, \quad T = T_w = T_\infty + T_0 e^{2x/l}, \quad (4a)$$

- as $y \rightarrow \infty$:

$$u = u_e(x), \quad T = T_\infty, \quad (4b)$$

where the velocity components along the axial and normal directions are u and v , respectively. Additionally, blood density, viscosity, electrical conductivity and thermal conductivity are denoted by ρ , μ , σ , κ , respectively. The subscript symbol $(\dots)_{nf}$ indicates a nanofluid, which refers to a base fluid with nanoparticles. The proposed model is shown in Fig. 2.

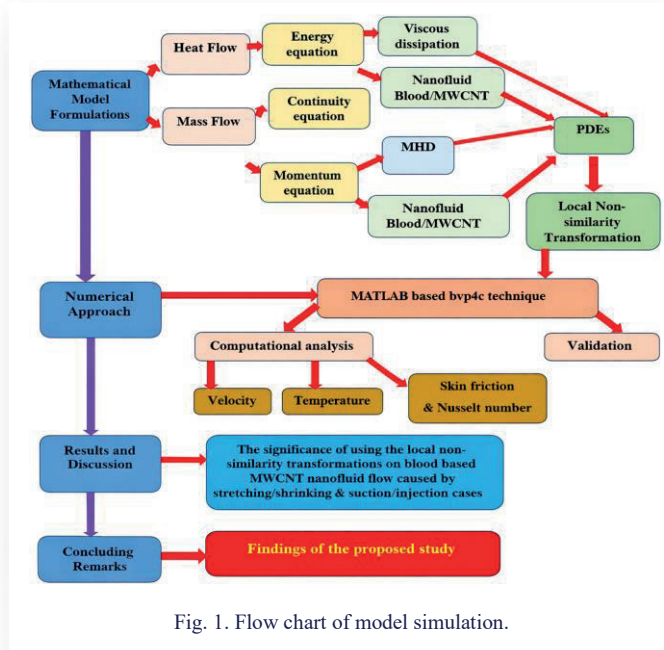


Fig. 1. Flow chart of model simulation.

The effective thermo-physical characteristics of the applied nanofluid model equations are given in [25–28], where ϕ is the particle volume fraction. Additionally, the symbols R_p , h denote the particle radius and interspacing, respectively.

$$\frac{\mu_{nf}}{\mu_{blood}} = 1 + 2.5\phi + 4.5 \left[\frac{1}{\frac{h}{R_p} \left(2 + \frac{h}{R_p} \right) \left(1 + \frac{h}{R_p} \right)^2} \right],$$

$$\rho_{nf} = (1 - \phi)\rho_{blood} + \phi\rho_{MWCNT},$$

$$(\rho C_p)_{nf} = (1 - \phi)(\rho C_p)_{blood} + \phi(\rho C_p)_{MWCNT}, \quad (5)$$

$$\frac{\sigma_{nf}}{\sigma_{blood}} = (1 - \phi)\sigma_{blood} + \phi\sigma_{MWCNT},$$

$$\frac{\kappa_{nf}}{\kappa_{blood}} = \frac{(\kappa_{MWCNT} + 2\kappa_{blood}) - 2\phi(\kappa_{blood} - \kappa_{MWCNT})}{(\kappa_{MWCNT} + 2\kappa_{blood}) + \phi(\kappa_{blood} - \kappa_{MWCNT})}.$$

To mathematically handle this proposed model, the following nonsimilarity flow is introduced [29,30]:

$$\xi = e^{x/l}, \quad \eta = \sqrt{\frac{b}{2v_f l}} y e^{x/2l},$$

$$\theta(\xi, \eta) = \frac{T - T_\infty}{T_w - T_\infty}, \quad u = b e^{x/l} f'(\xi, \eta), \quad (6)$$

$$v = -\sqrt{\frac{b v_f}{2l}} e^{x/2l} \left(\eta f'(\xi, \eta) + f(\xi, \eta) + 2\xi \frac{\partial f}{\partial \xi} \right),$$

where ξ is known as a nonsimilarity variable, while η is known as a pseudosimilarity variable.

Equations (2) and (3) are reduced into the following nonsimilar system through the use of Eqs. (5) and (6) as follows:

$$A_1 f''' + f f'' - 2(f'^2 - 1) - 2A_2 M(f' - 1) - 2\xi \left[f' \frac{\partial^2 f}{\partial \xi \partial \eta} - f'' \frac{\partial f}{\partial \xi} \right] = 0 \quad (7)$$

$$A_3 A_4 \theta'' + Pr \left[A_4 A_5 Ec f'^2 + f \theta' - 4\theta f' \right] - 2Pr\xi \left[f' \frac{\partial \theta}{\partial \xi} - \theta' \frac{\partial f}{\partial \xi} \right] = 0, \quad (8)$$

along with:

$$f(\xi, 0) + 2\xi \frac{\partial f}{\partial \xi} = S, \quad f'(\xi, 0) = \lambda, \quad \theta(\xi, 0) = 1,$$

$$f'(\xi, \infty) \rightarrow 0, \quad \theta(\xi, \infty) \rightarrow 0, \quad (9)$$

where prime $(\dots)'$ denotes the derivatives concerning η .

Additionally, the physical nondimensional parameters are given in Table 1.

Table 1. Physical parameters.

Parameter	Expression
Stretching/shrinking parameter	$\lambda = \frac{b}{c}$ Stretching whenever: $\lambda > 0$ Shrinking whenever: $\lambda < 0$ Static whenever: $\lambda = 0$
Suction/injection parameter	$S = -\sqrt{\frac{2l}{b v_f}} v_0$ Suction whenever: $S > 0$ Injection whenever: $S < 0$
Magnetic field parameter	$M = \frac{B_0^2 \sigma_f l}{b \rho_f}$
Prandtl number	$Pr = \frac{v_f}{\alpha_f}$
Eckert number	$Ec = \frac{T_0 (\rho C_p)_f}{b^2}$
Local Reynolds number	$Re = \frac{b l}{v_f}$

Additionally, we have:

$$A_1 = \frac{1 + 2.5\phi + 4.5 \left[\frac{1}{\frac{h}{R_p} \left(2 + \frac{h}{R_p} \right) \left(1 + \frac{h}{R_p} \right)^2} \right]}{\left(1 - \phi + \phi \frac{\rho_{MWCNT}}{\rho_{blood}} \right)}, \quad A_2 = \frac{\left(1 - \phi + \phi \frac{\sigma_{MWCNT}}{\sigma_{blood}} \right)}{\left(1 - \phi + \phi \frac{\rho_{MWCNT}}{\rho_{blood}} \right)},$$

$$A_3 = \frac{\kappa_{nf}}{\kappa_{blood}}, \quad A_4 = \frac{1}{\left(1 - \phi + \phi \frac{(\rho C_p)_{MWCNT}}{(\rho C_p)_{blood}} \right)},$$

$$A_5 = 1 + 2.5\phi + 4.5 \left[\frac{1}{\frac{h}{R_p} \left(2 + \frac{h}{R_p} \right) \left(1 + \frac{h}{R_p} \right)^2} \right].$$

Two important physical quantities under the proposed model are the skin friction parameter and the local Nusselt number:

$$(2Re)^{-\frac{1}{2}} C_f = \frac{A_1}{\xi} f'' f(\xi, 0),$$

$$(2Re)^{-\frac{1}{2}} Nu = -\xi \theta'(\xi, 0). \quad (10)$$

3. Numerical procedure and validation

3.1. Local similarity technique

It was suggested that the nonsimilar boundary layers may be solved by applying a local nonsimilarity approach, which was initially presented by Minkowycz and Sparrow [28] and then developed by [29–32]. The local similarity form of Eqs. (7) and (8) associated with boundary conditions is obtained as follows by removing the terms associated with $\partial f / \partial \xi$ and $\partial \theta / \partial \xi$:

$$A_1 f''' + f f'' - 2(f'^2 - 1) - 2A_2 M(f' - 1) = 0, \quad (11)$$

$$A_3 A_4 \theta'' + Pr \left[A_4 A_5 Ec f'^2 + f \theta' - 4\theta f' \right] = 0, \quad (12)$$

along with:

$$f(\xi, 0) + 2\xi \frac{\partial f}{\partial \xi} = S, \quad f'(\xi, 0) = \lambda, \quad \theta(\xi, 0) = 1,$$

$$f'(\xi, \infty) \rightarrow 0, \quad \theta(\xi, \infty) \rightarrow 0 \quad (12)$$

3.2. Local nonsimilarity technique

3.2.1. First truncation level

To achieve first-level truncation, Eqs. (7)–(8) are kept without approximation and substituted:

$$\begin{cases} g = \frac{\partial f}{\partial \xi} \\ m = \frac{\partial \theta}{\partial \xi} \end{cases} \quad (14)$$

Therefore, one can obtain:

$$A_1 f''' + f f'' - 2(f'^2 - 1) - 2A_2 M(f' - 1) - 2\xi [f' g' - g f''] = 0 \quad (15)$$

$$A_3 A_4 \theta'' + Pr \left[A_4 A_5 Ec f'^2 + f \theta' - 4\theta f' \right] - 2Pr\xi [f' m - \theta' g] = 0, \quad (16)$$

along with:

$$f(\xi, 0) + 2\xi g = S, \quad f'(\xi, 0) = \lambda, \quad \theta(\xi, 0) = 1$$

$$f'(\xi, \infty) \rightarrow 0, \quad \theta(\xi, \infty) \rightarrow 0 \quad (17)$$

3.2.2. Second truncation level

Now, we differentiate with respect to ξ and remove all the terms $\frac{\partial g(\eta, \xi)}{\partial \xi}$, $\frac{\partial m(\eta, \xi)}{\partial \xi}$ and their derivatives. In light of the above, the system of the 2nd truncation level contains Eq. (18)–(20):

$$A_1 g''' + f g'' + 3g f'' - 6f' g' - 2A_2 M g' - 2\xi [g'^2 - g g''] = 0, \quad (18)$$

$$Pr[2A_4 A_5 Ec f'' g'' + f m' - 6m f' + 3\theta' g - 4\theta g'] + 2Pr\xi [g' m - m' g] + A_3 A_4 m'' = 0 \quad (19)$$

with the following BCs:

$$g(\xi, 0) = 0, \quad g'(\xi, 0) = 0, \quad m(\xi, 0) = 0$$

$$g'(\xi, \infty) \rightarrow 0, \quad m(\xi, \infty) \rightarrow 0. \quad (20)$$

4. Results and discussion

This section presents a visual representation of the computational results of the physical parameters that affect the fluid velocity, temperature, skin friction and Nusselt number. We used the bvp4c approach, which is based on MATLAB, to handle this computationally. Table 2 also presents a comparison with existing research to determine the accuracy of the numerical approach used, which was determined to be within acceptable bounds. Additionally, this study used MWCNT particles to simulate blood numerically. Consequently, we conducted further research [34–37] and considered the information shown in Table 3 to produce realistic simulations.

Table 2. Comparison of values of $f''(0)$ when $M = \phi = \zeta = Ec = 0$, $\lambda = 1$, $Pr = 7$ for various values of S .

S	Present outcomes (local nonsimilarity solutions)	Gorla and Sidawi [33]
−0.7	−0.72250	−0.72252
−0.4	−0.83418	−0.83420
0.0	−1.01441	−1.01435
0.4	−.2313	−1.23180
0.7	−1.41879	−1.41884

Table 3. Physical properties of blood and MWCNTs.

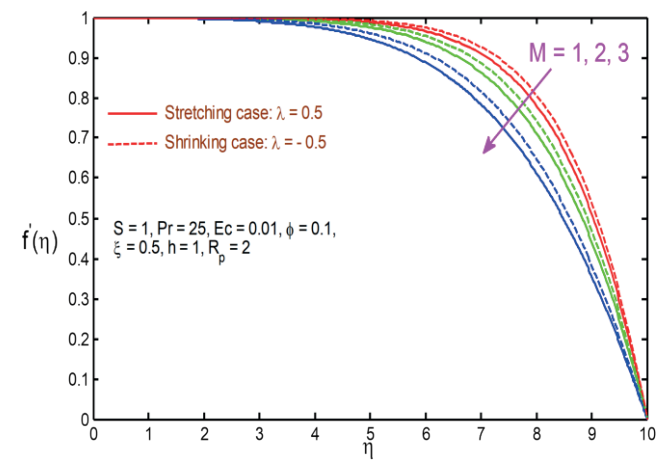
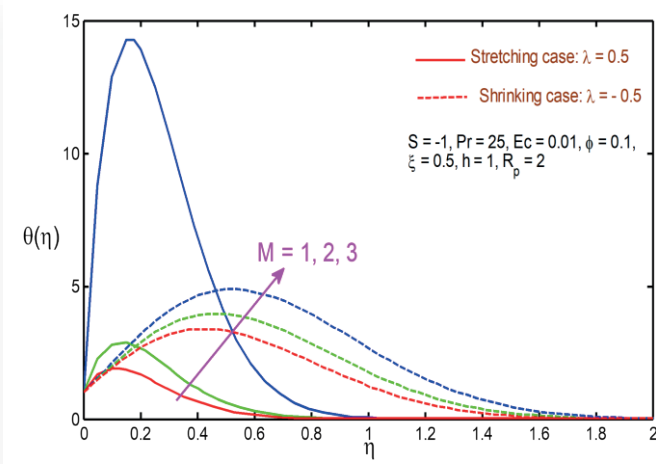
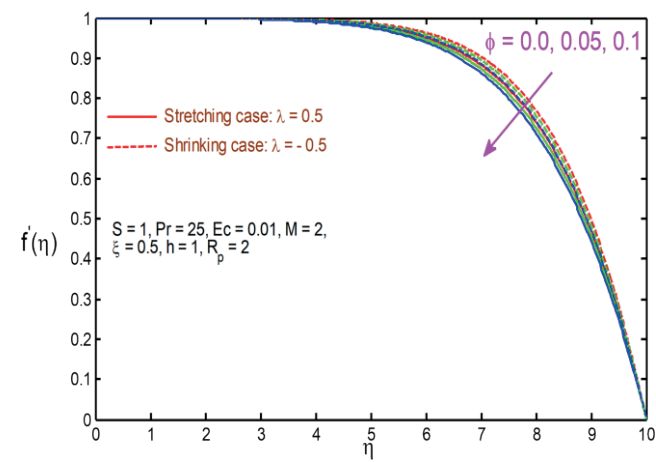
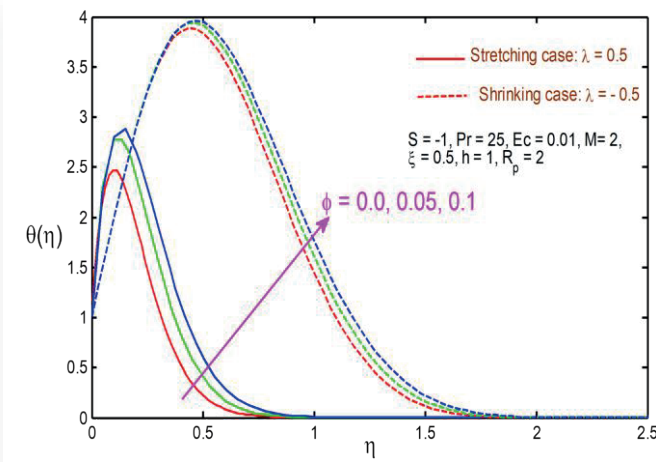
Properties	Base fluid: blood	Nanoparticles MWCNT
C_p (J kg ^{−1} K ^{−1})	3.9×10^3	796
ρ (kg m ^{−3})	1050	1600
σ (s m ^{−1})	0.8	1.9×10^{-4}
κ (W m ^{−1} K ^{−1})	0.5	3.7

In this study, we present a comprehensive comparison of the following:

- blood/MWCNT behaviour in the stretched case,
- blood/MWCNT behaviour in the shrinking case.

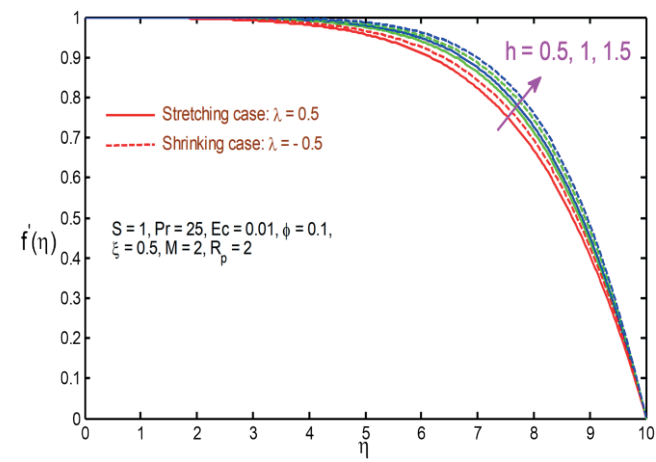
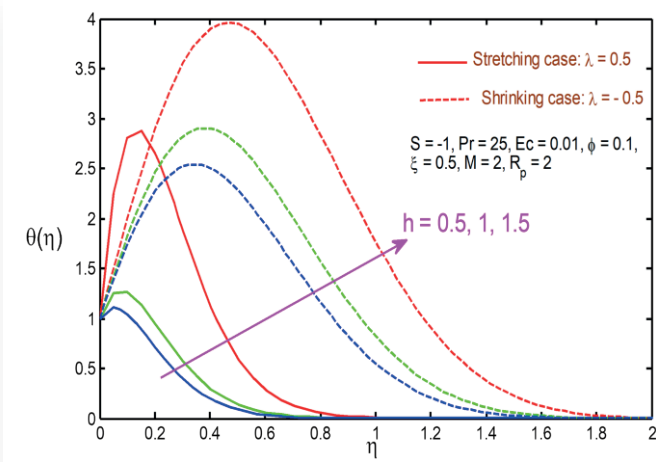
Figures 3 and 4 illustrate the influence of the magnetic field parameter on the velocity and temperature profiles for both the stretching and shrinking cases. Increasing the magnetic field parameter reduces the fluid velocity while increasing the temperature distribution. This behaviour is attributed to the Lorentz force generated by the interaction between the magnetic field and the electrically conducting nanofluid, which acts as a resistive force opposing the fluid motion and suppressing the velocity field. The reduced fluid motion weakens convective heat transport and promotes thermal energy accumulation within the boundary layer, resulting in an increase in the temperature profile.

Figures 5 and 6 illustrate the effects of nanoparticle volume fraction on the velocity and temperature profiles. Increasing the volume fraction reduces the fluid velocity while enhancing the

Fig. 3. Influence of M on $f'(\eta)$.Fig. 4. Influence of M on $\theta(\eta)$.Fig. 5. Influence of ϕ on $f'(\eta)$.Fig. 6. Influence of ϕ on $\theta(\eta)$.

temperature distribution. The reduction in velocity occurs because the addition of nanoparticles increases the effective viscosity of the nanofluid, which strengthens the momentum resistance within the boundary layer. Conversely, the temperature increases since a higher nanoparticle concentration improves the effective thermal conductivity of the nanofluid, thereby increasing heat transport within the fluid.

Figures 7 and 8 present the influence of the particle spacing on the velocity and temperature distributions. Increasing the particle spacing enhances both the velocity and temperature profiles. Physically, a larger interparticle spacing reduces nanoparticle–nanoparticle interactions and agglomeration effects, which promotes more uniform dispersion of nanoparticles within the base fluid. This improved dispersion facilitates more efficient mo-

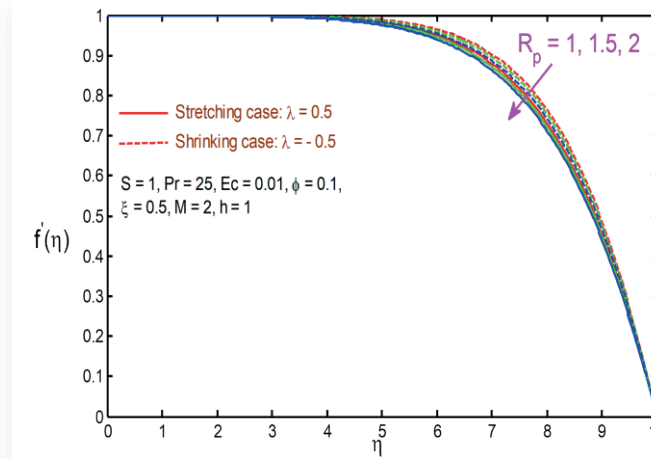
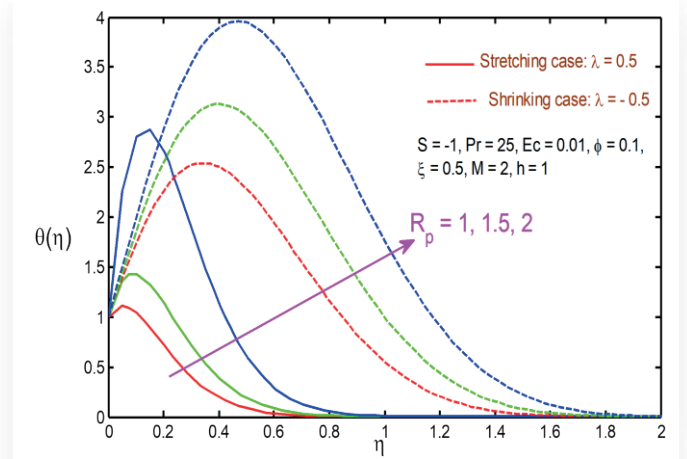
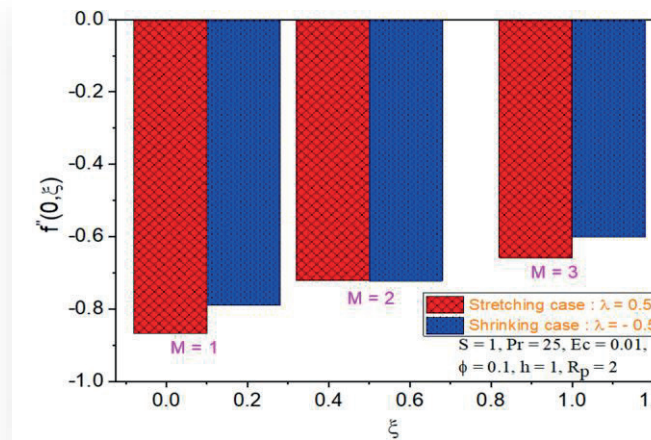
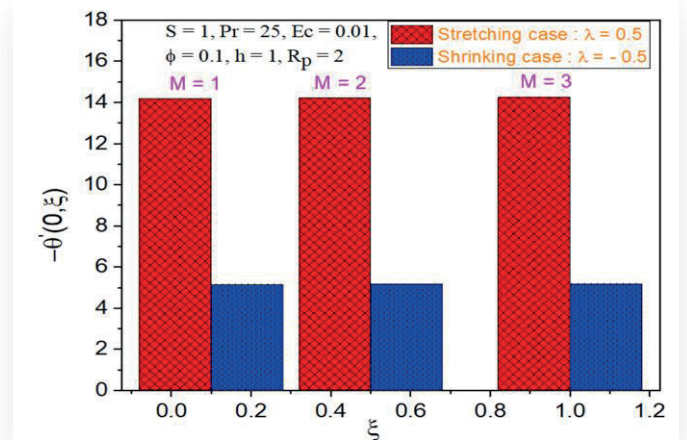
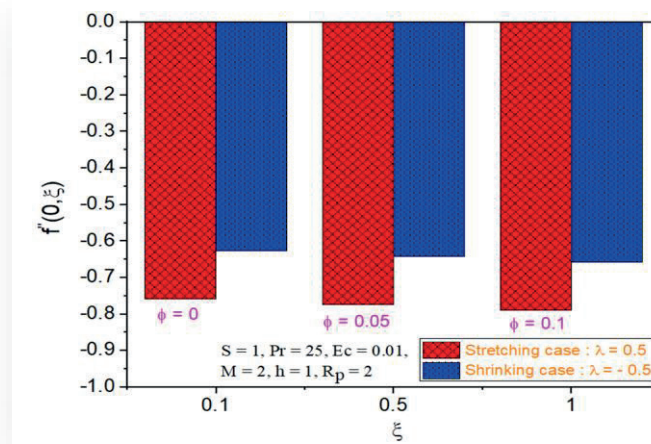
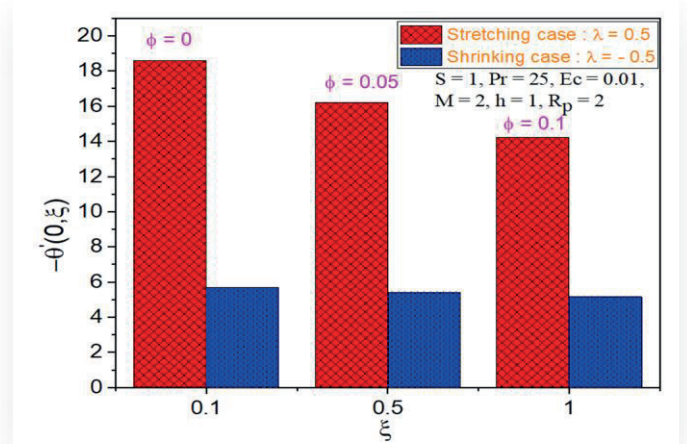
Fig. 7. Influence of h on $f'(\eta)$.Fig. 8. Influence of h on $\theta(\eta)$.

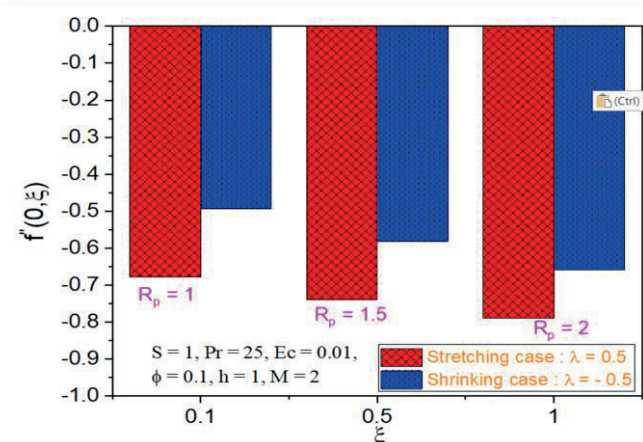
mentum and heat transport within the boundary layer, resulting in enhanced velocity and temperature distributions.

Figures 9 and 10 illustrate the influence of the particle radius on the velocity and temperature distributions. Increasing the particle radius decreases the velocity profile while enhancing the temperature distribution. The reduction in velocity is attributed to the increase in effective viscosity and particle inertia associated with larger nanoparticles, which strengthens the momentum

resistance within the boundary layer. Conversely, the temperature increases because larger particles increase the effective thermal conductivity of the nanofluid, promoting improved heat transport within the fluid.

Finally, Figs. 11 to 16 examine variations of the skin friction and local Nusselt number with physical parameters such as the particle radius, particle volume fraction and magnetic field.

Fig. 9. Influence of R_p on $f'(\eta)$.Fig. 10. Influence of R_p on $\theta(\eta)$.Fig. 11. Influence of M on $f''(0, \zeta)$ against ζ .Fig. 12. Influence of M on $-\theta'(0, \zeta)$ against ζ .Fig. 13. Influence of ϕ on $f''(0, \zeta)$ against ζ .Fig. 14. Influence of ϕ on $-\theta'(0, \zeta)$ against ζ .

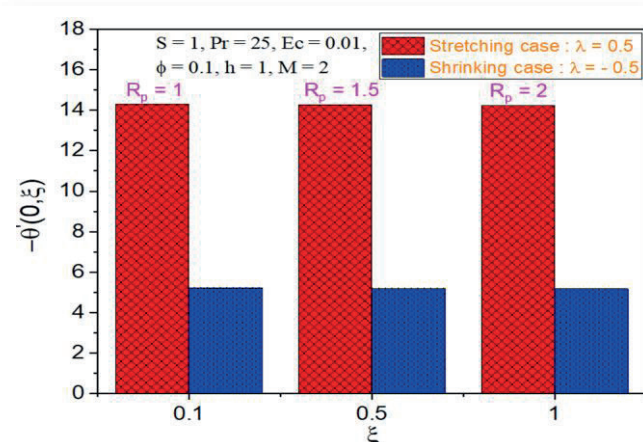
Fig. 15. Influence of R_p on $f''(0, \xi)$ against ξ .

According to the study, for both the stretching and shrinking scenarios, the magnetic field parameter improves both the skin friction and the Nusselt number. The skin friction is shown to improve by 24.15% in the stretching case and by 23.9% in the shrinking case over a range of magnetic field parameter values from 1 to 3, whereas the Nusselt number improves by 0.36% and 0.68% for the stretching and shrinking situations, respectively. Both the skin friction and the Nusselt number decrease as the particle radius increases. The skin friction decreases by 16.52% and 33.18% for the stretching and shrinking scenarios, respectively, for particle radius values ranging from 1 to 2. The Nusselt number values are decreased by 0.53% and 0.96%, respectively, for the stretching and shrinking scenarios. The skin friction and Nusselt number decrease as the particle volume fraction increases from 0 to 0.1. According to the study, the skin friction decreases by approximately 4.2% in the stretching case and by approximately 4.97% in the shrinking case. The Nusselt number likewise shows a noticeable shift, decreasing by approximately 23.46% and 8.83%, respectively, in the stretching and shrinking scenarios.

5. Conclusions

The present study investigated the nonsimilar flow and heat transfer characteristics of a biomagnetic blood-based MWCNT nanofluid over an exponentially stretched/shrinking sheet under the influence of magnetic field, particle radius, particle volume fraction and particle spacing. The governing equations were solved via the local nonsimilarity technique with the MATLAB bvp4c solver. The key findings are summarised as follows:

- The velocity profile decreases significantly with increasing magnetic field parameters, particle radius and particle volume fraction, whereas the temperature profile increases under the same conditions for both the stretching and shrinking cases.
- Increasing the particle spacing enhances both the velocity and temperature distributions, indicating improved nanoparticle dispersion within the base fluid.
- The skin friction coefficient increases by approximately 24.15% in the stretching case and 23.9% in the shrinking case when the magnetic field parameter increases within the considered range.

Fig. 16. Influence of R_p on $-\theta'(0, \xi)$ against ξ .

- The heat transfer rate (Nusselt number) decreases by approximately 23.46% and 8.83% for the stretching and shrinking cases, respectively, when the nanoparticle volume fraction increases from 0 to 0.1.
- Increasing the particle radius from 1 to 2 reduces the skin friction coefficient by approximately 16.52% in the stretching case and 33.18% in the shrinking case, while also slightly decreasing the Nusselt number.

From a practical perspective, the results indicate that magnetic control and nanoparticle characteristics can significantly influence heat transfer behaviour in biomagnetic nanofluid systems, which may be useful in biomedical engineering applications such as magnetic drug delivery, targeted thermal therapy and microscale cooling technologies.

Although the present analysis provides useful insight into biomagnetic nanofluid transport, the model is developed under simplified assumptions such as constant thermophysical properties, Newtonian fluid behaviour, and steady laminar flow conditions. In real biomedical environments, additional phenomena such as ferrohydrodynamic (FHD) interactions, variable viscosity and thermal conductivity, non-Newtonian blood rheology, and radiative heat transfer may significantly affect the flow and heat transfer characteristics. The incorporation of these effects in future studies would provide a more realistic description of biomagnetic nanofluid transport in biomedical and microscale thermal systems.

References

- [1] Saeed, A., Alsubie, A., Kumam, P., Nasir, S., Gul, T., & Kumam, W. (2021). Blood based hybrid nanofluid flow together with electromagnetic field and couple stresses. *Scientific Reports*, 11, 12865. doi: 10.1038/s41598-021-92186-z
- [2] Alanazi, M.M., Hendi, A.A., Raza, Q., Rehman, M.A., Qureshi, M.Z.A., Ali, B., & Shah, N.A. (2023). Numerical computation of hybrid morphologies of nanoparticles on the dynamic of nanofluid: The case of blood-based fluid. *Axioms*, 12, 163. doi: 10.3390/axioms12020163
- [3] Mohamed, M.K.A., Yasin, S.H.M., Salleh, M.Z., & Alkasasbeh, H.T. (2021). MHD stagnation point flow and heat transfer over a stretching sheet in a blood-based Casson ferrofluid with Newtonian heating. *Journal of Advanced Research in Fluid*

Mechanics and Thermal Sciences, 82, 1–11. doi: 10.37934/arfmts.82.1.111

- [4] Samat, N.A.A., Bachok, N., & Arifin, N.M. (2024). Boundary layer stagnation point flow and heat transfer over a nonlinear stretching/shrinking sheet in hybrid carbon nanotubes under magnetohydrodynamics. *Computation*, 12, 46. doi: 10.3390/computation12030046
- [5] Khan, W.A., Khan, Z.H., & Rahi, M. (2014). Fluid flow and heat transfer of carbon nanotubes along a flat plate with Navier slip boundary. *Applied Nanoscience*, 4, 633–641. doi: 10.1007/s13204-013-0242-9
- [6] Khan, M.R., Pan, K., Khan, A.U., & Ullah, N. (2020). Comparative study on heat transfer in CNTs–water nanofluid over a curved surface. *International Communications in Heat and Mass Transfer*, 116, 104707. doi: 10.1016/j.icheatmasstransfer.2020.104707
- [7] Das, K., Sutradhar, B., & Kundu, P.K. (2024). Impact of nonlinear radiation on an unsteady magneto hybrid nanofluid flow over an upward/downward rotating disk. *Numerical Heat Transfer, Part A*, 85, 2664–2681. doi: 10.1080/10407782.2023.2228477
- [8] Priyadharshini, P., & Archana, M.V. (2025). Artificial neural networks computing system for Ree–Eyring hybrid nanofluid flow over a solar panel sheet: Hamilton–Crosser model. *International Journal of Ambient Energy*, 46, 2451212. doi: 10.1080/01430750.2025.2451212
- [9] Rehman, A., Khan, I.A., Abualnaja, K.M., Ali, H.E., Jan, R., & Mahmoud, E.E. (2025). Analytical simulation of Darcy–Forchheimer MHD flow of CNT nanofluids over a stretching/shrinking surface with variable viscosity and slip conditions. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, 47, 184. doi: 10.1007/s40430-025-05428-w
- [10] Ramanuja, M., Kavitha, J., Sudhakar, A., & Radhika, N. (2023). Study of MHD SWCNT–blood nanofluid flow with viscous dissipation and radiation effects through porous medium. *Journal of the Nigerian Society of Physical Sciences*, 5, 1054–1054. doi: 10.46481/jnsps.2023.1054
- [11] Nagaraja, B., Vidhya, K.G., Almeida, F., & Kumar, P. (2025). Numerical illustration of diffusive flow of blood-based tri-hybrid nanofluid generated by a curved stretching sheet using law of porosity. *Numerical Heat Transfer, Part A: Applications*, 86(16), 5626–5647. doi: 10.1080/10407782.2024.2333042
- [12] Rehman, A., Khan, D., Jan, R., Aloqaily, A., & Mlaiki, N. (2023). Marangoni convection in stagnation point flow of blood-based CNT nanofluid over an unsteady stretching surface. *International Journal of Thermofluids*, 20, 100470. doi: 10.1016/j.ijft.2023.100470
- [13] Nasir, M., Bég, O.A., Al-Dossari, M., Sarhan, N.M., Kuharat, S., Waqas, M., & Daminova, G. (2024). SWCNT/MWCNT-doped thermomagnetic nanoblood boundary layer flow with non-Darcy, chemical reaction, viscous heating and Joule dissipation. *Diamond and Related Materials*, 148, 111447. doi: 10.1016/j.diamond.2024.111447
- [14] El Ghil, I., & Driouch, M. (2024). Impact of inclined magnetic field on nonorthogonal stagnation point flow of CNT–water through stretching surface in a porous medium. *Journal of Thermal Engineering*, 10(1), 115–129. doi: 10.18186/thermal.1429409
- [15] Mahanta, C., & Sharma, R.P. (2024). Collective impact of convective boundary and slip conditions on Casson ternary nanofluid flow over a stretching sheet. *Journal of Thermal Analysis and Calorimetry*, 149, 8391–8408. doi: 10.1007/s10973-024-13146-z
- [16] Ahmad, S., Khan, A.A., Khan, M.N., Knani, S., Ahammad, N.A., & Eleesy, I.E. (2025). Energy Transport in Ternary Hybrid

Nanofluid with Variable Thermal Conductivity under the Combined Influence of Electroviscous Effect, Electric Potential, and Heat Generation. *Dynamics of Atmospheres and Oceans*, 112, 101619. doi: 10.1016/j.dynatmoce.2025.101619

- [17] Yasir, M., Khan, M.N., Ahammad, N.A., Alshaban, E., Hassaballa, A.A., & Elseesy, I.E. (2025). Rheology of thermally nonlinear radiated flow of cross-material induced by shrinking surface: Heat radiation applications. *Journal of Radiation Research and Applied Sciences*, 18(4), 101956. doi: 10.1016/j.jrras.2025.101956
- [18] Ahsan, N., Khan, M.N., Aslam, M.N., Saidani, T., Alnfai, M.M., & Gamaoun, F. (2025). Bio-convective flow of non-Newtonian nanofluid with triple stratification effect over a curved stretching surface. *ZAMM-Journal of Applied Mathematics and Mechanics/Zeitschrift für Angewandte Mathematik und Mechanik*, 105(11), e70280. doi: 10.1002/zamm.70280
- [19] Yasir, M., Choubani, K., Khan, M.N., Almeshaal, M.A., & Husien, M. (2025). Influence of Cattaneo–Christov radiative heat flux in magnetized flow performance of three different materials with variable constraints. *Dynamics of Atmospheres and Oceans*, 112, 101608. doi: 10.1016/j.dynatmoce.2025.101608
- [20] Kallem, S.R., Sheri, S.R., Prathiba, A., & Shankar, G. (2025). Numerical analysis of the effect of chemical reaction and heat source on MHD hyperbolic tangent fluid flow across a non-linear stretching sheet in a porous medium. *Archives of Thermodynamics*, 46(2), 57–67. doi: 10.24425/ather.2025.154906
- [21] Biswas, N., Mandal, D.K., Manna, N.K., & Benim, A.C. (2025). Thermomagnetic convection and entropy generation in a hybrid nanofluid filled wavy-walled cavity heated non-uniformly. *Archives of Thermodynamics*, 46(1), 61–81. doi: 10.24425/ather.2025.154182
- [22] M, A., & De, P. (2025). Electro-osmotic and magnetohydrodynamic flow of Maxwell nanofluid over Darcy–Forchheimer porous medium with Soret–Dufour effects. *Archives of Thermodynamics*, 46(2), 69–81. doi: 10.24425/ather.2025.154907
- [23] Magyari, E., & Keller, B. (1999). Heat and mass transfer in boundary layers on an exponentially stretching continuous surface. *Journal of Physics D: Applied Physics*, 32(5), 577. doi: 10.1088/0022-3727/32/5/012
- [24] Prakash, O., Rao, P.S., Sharma, R.P., & Mishra, S.R. (2023). Hybrid nanofluid MHD motion towards an exponentially stretching/shrinking sheet with the effect of thermal radiation, heat source and viscous dissipation. *Pramana*, 97(2), 64. doi: 10.1007/s12043-023-02533-0
- [25] Ferdows, M., Alam, J., Murtaza, M.G., & Tzirtzilakis, E.E. (2023). Effects of magnetic particle diameter and spacing on biomagnetic flow and heat transfer over a stretched cylinder in presence of magnetic dipole. *Journal of Mechanics in Medicine and Biology*, 23, 2350036. doi: 10.1142/S0219519423500367
- [26] Alam, J., & Murtaza, M.G. (2024). Significance of biomagnetic fluid flow with Ni–ZnFe₂O₄ particle over a wedge in the presence of a magnetic dipole. *Multiscale and Multidisciplinary Modeling, Experiments and Design*, 7, 5673–5684. doi: 10.1007/s41939-024-00526-z
- [27] Alam, J., Murtaza, M.G., Petropoulou, E.N., Tzirtzilakis, E.E., & Ferdows, M. (2023). A parametric simulation of MHD flow and heat transfer of blood–Fe₃O₄ nanofluid over an exponentially stretching cylinder. *BioNanoScience*, 13, 891–899. doi: 10.1007/s12668-023-01141-x
- [28] Minkowycz, W.J., & Sparrow, E.M. (1974). Local nonsimilar solutions for natural convection on a vertical cylinder. *Journal of Heat Transfer*, 96(2), 178–183. doi: 10.1115/1.3450161
- [29] Mansoor, M., Nawaz, Y., & Ul-Hassan, Q.M. (2022). Nonsimilar method for heat and mass transfer of MHD radiative flow over an

- exponentially stretching sheet. *Thermal Science*, 26(5A), 3703–3714. doi: 10.2298/TSCI210505309M
- [30] Cui, J., Razzaq, R., Farooq, U., Khan, W.A., Farooq, F.B., & Muhammad, T. (2022). Forced convection analysis of nanofluid flow over a stretching sheet with chemical reaction and heat generation using nonsimilar modeling. *Alexandria Engineering Journal*, 61, 4253–4261. doi: 10.1016/j.aej.2021.09.045
- [31] Ferdows, M., Barmon, A., Bég, O.A., Shamshuddin, M.D., & Sun, S. (2023). Nonisothermal electroconductive radiative stretching boundary layer heat transfer with aligned magnetic field: A local nonsimilar solution. *Applied Sciences*, 13(7), 4592. doi: 10.3390/app13074592
- [32] Razzaq, R., & Farooq, U. (2021). Nonsimilar forced convection analysis of Oldroyd-B fluid flow over an exponentially stretching surface. *Advances in Mechanical Engineering*, 13(7). doi: 10.1177/168781402111034604
- [33] Gorla, R.S.R., & Sidawi, I. (1994). Free convection on a vertical stretching surface with suction and blowing. *Applied Scientific Research*, 52, 247–257. doi: 10.1007/BF00853952
- [34] Tzirtzilakis, E.E. (2005). A mathematical model for blood flow in magnetic field. *Physics of Fluids*, 17, 077103. doi: 10.1063/1.1978807
- [35] Tzirtzilakis, E.E., & Tanoudis, G.B. (2003). Numerical study of biomagnetic fluid flow over a stretching sheet with heat transfer. *International Journal of Numerical Methods for Heat and Fluid Flow*, 13(7), 830–848. doi: 10.1108/09615530310502055
- [36] Alam, J., Murtaza, M.G., Petropoulou, E.N., Tzirtzilakis, E.E., & Ferdows, M. (2022). Group-theoretical simulation of biomagnetic fluid flow and heat transfer with Fe_3O_4 particles under thermal radiation and magnetic dipole. *Mathematics*, 10(19), 3520. doi: 10.3390/math10193520
- [37] Alkasasbeh, H.T., Swalmeh, M.Z., Bani Saeed, H.G., Al Faqih, F.M., & Talafha, A.G. (2020). Investigation on CNTs–water and human blood-based Casson nanofluid flow over a stretching sheet under impact of magnetic field. *Frontiers in Heat and Mass Transfer*, 14, 1–7. doi:10.5098/hmt.14.15



Co-published by
Institute of Fluid-Flow Machinery
Polish Academy of Sciences
Committee on Thermodynamics and Combustion
Polish Academy of Sciences

Copyright©2026 by the Authors under licence CC BY-NC-ND 4.0

<http://www.imp.gda.pl/archives-of-thermodynamics/>



Convective Transfer Analysis of Magnetised Ternary Hybrid Nanofluid over a Linear Stretching Sheet: Non-Similar Solutions

Ahmed Jan^{a*}, Farida Aslam^b, Rubina Adnan^c

^aCollege of Mechanical and Electronic Engineering, Shandong University of Science and Technology, Qingdao, Shandong Province, 266590, PR China

^bDepartment of Mathematical Science, BUITEMS, Quetta 87300, Pakistan

^cDepartment of Computer Science, COMSATS University Islamabad, Pakistan

*Corresponding author email: ahmedjan400@gmail.com; jan_ahmed@sdust.edu.cn

Received: 21.11.2025; revised: 13.04.2026; accepted: 22.04.2026

Abstract

Use of nanofluids to improve heat transfer properties has drawn considerable concern due to their wide use in various applications such as electronic appliances, fuel cell batteries and hybrid-powered engines. These are only a few examples of the many applications of heat transfer that can greatly utilise the distinctive properties of nanofluids. A magnetohydrodynamic nanofluid convective study is carried out for a sheet stretched under a non-uniform heat source/sink in a permeable medium. The heat transfer analysis simulations entail dissipation of porosity and the presence of Ohmic heating. Non-similarity transformations are then applied to the equations that have been formulated to derive dimensionless partial differential equations. Local non-similarity approach has been used to transform partial differential equations into an equation where partial differential equations can be solved as ordinary differential equations, using the bvp4c MATLAB tool. Every physical parameter is represented in a graphical representation. Moreover, the tables that present the summary of the skin friction coefficient and Nusselt number are evaluated in detail. The main findings of this study include the fact that an increase in the strength of the magnetic field decreases the fluid velocity but improves the thermal distribution. It has been observed that the higher the porosity, the lower the skin friction. In addition, high radiation levels also help in having a more favourable temperature field.

Keywords: Non-similar analysis; Non-uniform heat source/sink; Magnetohydrodynamic; Thermal radiation; MATLAB bvp4c.

Vol. 47(2026), No. 2, 141–150; doi: 10.24425/ather.2026.158680

Cite this manuscript as: Jan, A., Aslam, F., & Adnan, R. (2026). Convective Transfer Analysis of Magnetised Ternary Hybrid Nanofluid over a Linear Stretching Sheet: Non-Similar Solutions. *Archives of Thermodynamics*, 47(2), 141–150.

1. Introduction

There have already been extensive studies designed to analyse the thermo-physical properties of traditional heat transfer fluids, but these fluids are not sufficient to handle the increased challenges that are presented by contemporary technology. As a result, the introduction of nanoparticles to these fluids has been suggested as one of the solutions. Engineered composites (nanofluids) created by the mixing of nanoparticles with a base fluid

have emerged as a result. Nanoparticles, which are significantly smaller than the base fluid, are added to the base fluid. Recent experimental studies show that thermal conductivity is increased significantly when nanoparticles are incorporated into conventional fluids. The outstanding natural properties of nanofluids form a bridge between the microscopic and macroscopic molecular structures. Nanofluids cannot be compared with others in terms of heat transmission. It is through their good properties that they have become a hot commodity in various industries

Nomenclature

a – time constant, 1/s
 B_o – applied magnetic field, kg/(A·s²)
 C_f – skin friction
 c_p – specific heat, m²/(s²·K)
 Ec – Eckert number, $Ec = (al)^2 / (c_p)_f (T_w - T_\infty)$
 E^* – internal heat production
 f, g, h – dimensionless dependent variables
 F^* – internal heat absorption
 g – gravitational acceleration, m²/s
 Gr – Grashof number, $Gr = \beta_f g (T_w - T_\infty) l^3 / \nu_f^2$
 k – thermal conductivity, W/(m·K)
 k_p – permeability of free space, m²
 k^* – radiation absorption coefficient, 1/m
 M – magnetic number, $M = \sigma_f B^2 / a \rho_f$
 Nu – Nusselt number
 Pr – Prandtl number, $Pr = \alpha_f / \nu_f$
 q''' – heat source, W/m³
 q_r – thermal radiation, W/m²
 R_d – radiation parameter, $R_d = 4\sigma^* T_\infty^3 / K_f k^*$
 Re – Reynolds number, $Re = al^2 / \nu_f$
 T – temperature, K
 T_w – wall temperature, K

T_∞ – ambient temperature, K
 u, v – velocities, m/s
 x, y – independent variables, m

Greek symbols

α – mixed convection parameter, $\alpha = Gr/Re$
 α_f – thermal diffusivity, m²/s
 β_f – thermal expansion, 1/K
 θ – dimensionless temperature
 λ – porosity parameter, $\lambda = \nu_f / k_p a$
 ξ, η – non-similarity and pseudo variable
 μ_f – dynamic viscosity, kg/(m·s)
 ν_f – kinematic viscosity, m²/s
 ρ_f – density, kg/m³
 σ^* – Stefan-Boltzmann constant, W/(m²·K)

Subscripts and Superscripts

f – base fluid
 $thnf$ – ternary hybrid nanofluid

Abbreviations and Acronyms

MHD – magnetohydrodynamics
 ODE – ordinary differential equation
 PDE – partial differential equation

such as fuel cells, electronics and hybrid engines. Moreover, nanofluids are also used in industries due to their high levels of production of quality oils and lubricants. Choi and Eastman [1] first suggested the notion of nanoparticles while conducting classic research on nanofluids, which they characterised as tailored nanoparticle suspensions in standard heat transfer fluids. It has been demonstrated that the efficiency of traditional fluids in heat transfer can be significantly enhanced by the introduction of nanoparticles. Nanofluid thermal properties were widely researched by Buongiorno [2], who prepared a whole theoretical basis to accomplish the aim. The detailed mathematical study by Mabood et al. [3] on heat transfer in the nanofluid movement in a nonlinear sheet stretching in the boundary layer is very comprehensive. The experiment was carried out considering the influence of an external magnetic field. Saddiqui et al. [4] investigated the characteristics of thermal transport under the circumstances of constant movements of the boundary layer over a stretched melting boundary. The dissipation effects in the viscous viscosity were considered in the effort. Masood et al. [5] explored stagnation point flow patterns in a hybrid nanofluid that comprised polystyrene and titanium dioxide particles suspended in water. Heat production and uptake were evaluated in an extensive manner. Hussain et al. [6] studied combined heat and mass transport via the Cross fluid framework. The probe was used to light up the input of the magnetic dipole to the motion of ferrofluid on a hard surface.

Mixed convection is prevalent across a broad spectrum of engineering, industrial and natural systems where forced and free convection processes operate concurrently. These low-velocity flow scenarios are encountered in a variety of practical applications, such as heat exchangers, marine environments, atmospheric currents, solar receivers exposed to wind, fan-cooled

electronic components, and nuclear reactors during emergency shutdown scenarios. Chen et al. [7] conducted an investigation into the characteristics of laminar forced convective flow and heat transfer over a horizontal flat plate, utilising analytical methods rooted in local similarity and local non-similarity solutions to assess the impacts of streamwise pressure gradients induced by buoyancy. Chamkha et al. [8] studied coupled heat and mass transfer boundary-layer free convection over a vertical flat plate in a fluid-saturated porous medium, considering thermophoretic particle deposition and heat generation/absorption. Ahmad and Pop [9] analysed the boundary layer flow of nanofluids over a vertical flat plate embedded within a porous medium, focusing specifically on the performance of different nanoparticle varieties, including copper, alumina and titanium oxide. Mahanthesh et al. [10] explored how viscous dissipation and nonlinear thermal radiation affect the magnetohydrodynamic boundary layer flows of specific nanofluids. Cao et al. [11] studied the combined influences of nonlinear thermal radiation and partial slip conditions on the mixed convective boundary layer flow involving ternary hybrid nanoparticles of different sizes dispersed in nanofluids. Additionally, Jakeer et al. [12] carried out experiments to examine the effect of radiation on a flat sheet under nonlinear Darcy-Forchheimer flow conditions, with their research encompassing ternary hybrid nanofluids, water-based nanofluids and hybrid nanofluids.

Flow and heat transport in porous media have wide applications in various areas of science and engineering. Applications of porous media have been widely utilised in many industries, such as soil salts leaching, chemical-based catalytic reactors, grain preservation, heat exchange between soil and the atmosphere, solar energy harvest, electrochemical processes, thermal insulation of nuclear reactors, moisture diffusion in fibrous in-

ulating media, regenerative heat exchangers, geothermal systems, and others. A large amount of literature is available on these scenarios, mostly in the classical Darcy flow situation. However, at high flow rates or in porous media with high permeability, a non-linear Darcy law effect occurs, in which inertial effects are now of interest. The effect of departure is attributed to the fact that flow separation in the medium occurs and is observable when the Reynolds number exceeds 1. It has been found that the effect of non-Darcy phenomena on natural convection in porous materials is of interest, and experimental studies have investigated a wide range of governing parameters and used various combinations of fluids and solids. Laurait and Prasad [13] examined the free convection influenced by the boundary conditions in a vertical enclosure. Recently, Beckermann and co-authors [14] carried out a numerical study in which the Forchheimer-Brinkmann extended Darcy equation of motion was used. Cortell [15] was able to explore the dynamics of fluid flow when the surfaces were impermeable, with an internal heat source or absorption and surface distension. Salahuddin et al. [16] investigated the effect of a porous medium on nanofluid flow mixed convection near an exponentially contoured surface. Two base fluids, such as water and ethylene glycol, and alumina nanoparticles were considered. Hussain et al. [17] performed an analysis of the influence of nanofluid using the Darcy-Forchheimer Casson fluid model of a stretching sheet. The aim of a numerical analysis that was carried out by Keshavarzian and Sayehvand [18] was to assess the precision of the local thermal non-equilibrium concept and the Forchheimer-Brinkman extended Darcy model.

The study of magnetohydrodynamic (MHD) flow and heat transfer across a fixed or movable planar surface has evolved as a popular research topic with several applications in a variety of technical areas, including the petroleum industry, plasma studies, geothermal energy extraction and aerodynamics. A number of artificial solutions aimed at achieving this objective have been developed since effective control of boundary layer behaviour is a crucial component. Sparrow and Cess [19] investigated the influence of an induced magnetic field on free convection heat transfer from an isothermal plate. Ellahi et al. [20] examined entropy generation and slip effects in the context of MHD heat transfer over a flat plate. Jan et al. [21] assessed heat and flow transfer analysis of magnetised nanofluid flow over a curved geometry. Iqbal and Abbasi [23] evaluated entropy generation impacts on multi-layered peristaltic flow of a curved nanofluid. The evaluation incorporated temperature-sensitive thermal conductivity traits from a Carreau-Yasuda nanofluid. Ghasemi et al. [23] examined nanofluid dynamics over an extending surface amid radiation and magnetic influences. The examination applied a distinct spectral relaxation method. Iqbal et al. [24] determined that MHD, alongside a revised Darcy's law, alters Powell-Eyring nanofluid conveyance inside a bent channel under a radial magnetic field.

This paper concentrates on the thermal characteristics of a ternary hybrid nanofluid, which is a novel arrangement that exceeds the conventional mono and hybrid nanofluids with the application of highly improved thermal conductivities and viscosities. This is attributed to synergistic blending of three kinds

of nanoparticles, such as aluminium oxide Al₂O₃, titanium dioxide TiO₂ and magnesium oxide MgO, which are suspended in a liquid water base of ethylene glycol, and hence enhances the overall thermo-physical properties of the mixture, such as the density, specific heat and rheology.

The heat transfer enhancement rates in this ternary hybrid nanofluid, particularly its behaviour to nonuniform heat source/sink effects in a magnetised mixed convection boundary layer flow over an extending surface, are analysed. The greatest novelty of the study is using a nonuniform heat source, in the form of a spatially varying thermal generation term (e.g. quadratic or exponential distribution), which makes the heat flux distribution conditions much more realistic in the advanced engineering of the real world (solar collectors, nuclear reactors, electronic cooling systems, etc.). In comparison with the earlier study in which most of the literature either considered a uniform source of heat or an isolated binary nanofluid to steady thermal loading, this study is the first study to couple the nonuniform sources of heat with the complex interaction of thermo-physical processes in a ternary hybrid nanofluid to uncover the effects of perturbations due to nanoparticles on local temperature gradients and Lorentz force interactions in the magnetic forcing of the fluid. This interaction not only improves convective heat dissipation, but it also gets rid of possible hotspots that provide us with new information on how to make things more stable and efficient in mixed thermal settings. A thorough treatment of such phenomena must employ non-similarity transformations [25–29] to reduce the governing partial differential equations to a system of dimensionless ordinary differential equations that are solved numerically with the bvp4c solver of MATLAB to solve high-fidelity problems in the standard form of a boundary value problem. The physical observations, including velocity profiles, temperature distributions, skin friction coefficients and change in Nusselt number (Nu), are explained through detailed graphical plots and table representation, which elucidate the transformational quality of this hybrid solution considering the next-generation thermal management technology.

2. Problem formulation

This study examines two-dimensional incompressible boundary layer flow of a ternary hybrid nanofluid under nonuniform heat source/sink effects near an extending sheet. The flow domain is limited to $x \geq 0$ and $y \geq 0$, while the sheet resides at $y = 0$. The x -coordinate follows the sheet extension path. The y -coordinate runs perpendicular to the surface. Equal opposing forces along the x -axis extend the sheet and preserve the origin. Boundary layer approximations balance normal stress against shear stress in magnitude. A transverse magnetic field acts on the fluid. The energy equation integrates linear thermal radiation via the Rosseland diffusion model. The fluid's magnetic Reynolds number (Re) stays sufficiently low to neglect the Hall effect and induced magnetic field. The momentum equation accounts for both electric and magnetic field influences. Equations for motion and energy conservation explain mixed convection inside the boundary layer using the Boussinesq approximation, which is consistent with the specified physical framework.

A schematic flow diagram is shown in Fig. 1.

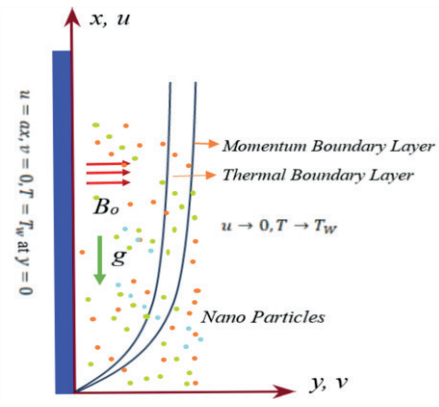


Fig. 1. Schematic flow diagram.

The set of governing equations is written as:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0, \quad (1)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = \frac{\mu_{thnf}}{\rho_{thnf}} \frac{\partial^2 u}{\partial y^2} + \frac{g(\rho\beta)_{thnf}}{\rho_{thnf}} (T - T_\infty) + \frac{\mu_{thnf}}{\rho_{thnf}} \frac{u}{k_p} - \frac{\sigma_{thnf} B_0^2}{\rho_{thnf}} u, \quad (2)$$

$$u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} = \frac{k_{thnf}}{(\rho c_p)_{thnf}} \frac{\partial^2 T}{\partial y^2} - \frac{1}{(\rho c_p)_{thnf}} \frac{\partial q_r}{\partial y} + \frac{\mu_{thnf}}{(\rho c_p)_{thnf}} \left(\frac{\partial u}{\partial y} \right)^2 + \frac{1}{(\rho c_p)_{thnf}} \left(\sigma_{thnf} B_0^2 + \frac{\mu_{thnf}}{k_p} \right) u^2 + \frac{q_r'''}{(\rho c_p)_{thnf}}, \quad (3)$$

with boundary conditions [28]:

$$\text{– at } y \rightarrow 0: \quad u = u(w) = ax, \quad v = 0, \quad T = T_w, \quad (4a)$$

$$\text{– as } y \rightarrow \infty: \quad u \rightarrow 0, \quad v \rightarrow 0, \quad T \rightarrow T_\infty. \quad (4b)$$

In this model, the dynamic viscosity is μ_f , ρ_f is the fluid density, heat capacitance is $(\rho c_p)_f$; T_w , T_∞ are the free stream and ambient temperature, respectively; T is the temperature, and

$$Q''' = \frac{k_{thnf}}{xv_{thnf}} [E^*(T_w - T_\infty) + u(w)F^*(T - T_\infty)],$$

$$q_r = \frac{4\sigma^*}{3k^*} \frac{\partial T^4}{\partial y},$$

where E^* and F^* are characteristics of internal heat production and absorption that vary with space and temperature [30]. It is observed that when $E^* > 0$ and $F^* > 0$, internal heat is being produced.

3. Coordinate transformation

The coordinate transformation is used to transform the equations of fluid flow and heat transfer between the physical domain and a computational or dimensionless domain. The method enables the equations to be reformulated into a more straightforward and tractable form to improve numerical accuracy and stability. The transformation characterises a correlation between the physical coordinates and a fresh group of coordinates which are more illustrative of the geometry or flow attributes of the issue. It allows scaling of variables such as length, velocity and tempera-

ture to relevant reference quantities, thereby assisting in the generalisation of the results to varying physical conditions. It is especially applicable in curved geometries or complicated configurations of boundaries, where it is hard to formulate these problems in physical coordinates. The main characteristics of coordinate transformation are that it can lessen the number of variables, remove any dependence on the dimensions, and that it can be used to apply the boundary conditions on irregular surfaces easily. It also enables a better understanding of the governing parameters. This method, as opposed to similarity transformations, does not lose the complete dependence of the variables on both spatial coordinates, which gives a more detailed description of the flow field without any restrictive assumptions. The following transformation is introduced:

$$\eta = y \sqrt{\frac{a}{v_f}}, \quad \xi = \frac{x}{l}, \quad u = ax \frac{\partial f}{\partial \eta}, \quad v = -\sqrt{av_f} \left(f + \xi \frac{\partial f}{\partial \xi} \right), \quad (5)$$

$$\theta(\xi, \eta) = \frac{T - T_\infty}{T_w - T_\infty}.$$

Using Eq. (5), Eq. (1) is identically satisfied. Eqs. (2)–(4) in a dimensionless non-similar system are given below:

$$\frac{\partial^3 f}{\partial \eta^3} - \left(\lambda + M \frac{\mu_f}{\mu_{thnf}} \frac{\sigma_{thnf}}{\sigma_f} \right) \frac{\partial f}{\partial \eta} + \frac{\mu_f}{\mu_{thnf}} \frac{(\rho\beta)_{thnf}}{(\rho\beta)_f} \xi^{-1} \alpha \theta + \frac{\rho_{thnf}}{\rho_f} \frac{\mu_f}{\mu_{thnf}} f \frac{\partial^2 f}{\partial \eta^2} - \frac{\rho_{thnf}}{\rho_f} \frac{\mu_f}{\mu_{thnf}} \left(\frac{\partial f}{\partial \eta} \right)^2 = \frac{\rho_{thnf}}{\rho_f} \xi \left(\frac{\partial f}{\partial \eta} \frac{\partial^2 f}{\partial \xi \partial \eta} - \frac{\partial f}{\partial \xi} \frac{\partial^2 f}{\partial \eta^2} \right), \quad (6)$$

$$\frac{1}{\text{Pr}} \left(\frac{k_{thnf}}{k_f} + R_d \right) \frac{\partial^2 \theta}{\partial \eta^2} + \text{Ec} \xi^2 \frac{\mu_{thnf}}{\mu_f} \left(\frac{\partial^2 f}{\partial \eta^2} \right)^2 + \text{Ec} \xi^2 \left(\lambda \frac{\mu_{thnf}}{\mu_f} + M \frac{\sigma_{thnf}}{\sigma_f} \right) \left(\frac{\partial f}{\partial \eta} \right)^2 + \frac{(\rho c_p)_{thnf}}{(\rho c_p)_f} f \frac{\partial \theta}{\partial \eta} + \frac{k_{thnf}}{k_f} \frac{\rho_{thnf}}{\rho_f} \frac{\mu_f}{\mu_{thnf}} \frac{\text{Ec}}{\text{Pr}} \left[E^* \frac{\partial f}{\partial \eta} + F^* \theta \right] = \frac{(\rho c_p)_{thnf}}{(\rho c_p)_f} \xi \left[\frac{\partial f}{\partial \eta} \frac{\partial \theta}{\partial \xi} - \frac{\partial f}{\partial \xi} \frac{\partial \theta}{\partial \eta} \right]. \quad (7)$$

Non-similar boundary conditions are:

$$\text{– at } \eta = 0: \quad \frac{\partial f}{\partial \eta} = 1, \quad f = 0, \quad \theta = 1, \quad (8a)$$

$$\text{– as } \eta \rightarrow \infty: \quad \frac{\partial f}{\partial \eta} \rightarrow 0, \quad \theta \rightarrow 0. \quad (8b)$$

4. Physical quantities

The surface friction coefficient C_f and the local Nusselt number Nu are defined as:

$$C_f = \left(\frac{\tau_w}{\rho_f (u_w)^2} \right)_{y=0}, \quad \text{Nu} = \left(\frac{x q_w}{k_f (T_w - T_\infty)} \right)_{y=0}, \quad (9)$$

where τ_w is the surface shear stress, q_w is the surface heat flux:

$$\tau_w = \left(\mu_{thnf} \frac{\partial u}{\partial y} \right)_{y=0}, \quad q_w = - \left(\frac{k_{thnf}}{k_f} + q_r \right) \left(\frac{\partial T}{\partial y} \right)_{y=0}. \quad (10)$$

The dimensionless forms of C_f and Nu_x are:

$$\left. \begin{aligned} \text{Re}^{\frac{1}{2}} C_f &= \xi^{-1} \frac{\mu_{thnf}}{\mu_f} \frac{\partial^2 f}{\partial \eta^2} (\xi, 0) \\ \text{Re}^{-\frac{1}{2}} \text{Nu} &= -\xi \left(\frac{k_{thnf}}{k_f} + R_d \right) \frac{\partial \theta}{\partial \eta} (\xi, 0) \end{aligned} \right\} \quad (11)$$

5. Local non-similarity method

To avoid complications caused by the introduction of spatial coordinates into the physical controlling parameters, a stream-wise coordinate must be introduced. The addition of these coordinates will help eliminate unnecessary variables from the flow equations. Applying the strategy described in Sparrow et al. [25] and Sparrow and Yu [26] modifies the local flow problem of similarity into a non-similar flow problem. In addition, the authors formulated the two-equation truncation model in this notion while considering the impact of the stream-wise coordinate [31,32]. In certain research, including the stream-wise coordinate has been shown to significantly improve the flow and mass balance, whereas in others, including this coordinate has not yet been shown to make a substantial improvement. Upon applying the Taylor series expansion, all terms containing $\partial/\partial \xi$ in Eqs. (6)–(8) are eliminated. This enables the introduction of necessary variables and the formulation of first- and second-level truncation equations.

5.1. First-level truncation

Assuming that $\xi \ll 1$, the right-hand sides of Eqs. (6)–(8) are equal to zero. Equations (6)–(8) can be rewritten as:

$$f''' - \left(\lambda + M \frac{\mu_f}{\mu_{thnf}} \frac{\sigma_{thnf}}{\sigma_f} \right) f' + \frac{\mu_f}{\mu_{thnf}} \frac{(\rho\beta)_{thnf}}{(\rho\beta)_f} \xi^{-1} \alpha \theta + \frac{\rho_{thnf}}{\rho_f} \frac{\mu_f}{\mu_{thnf}} f f'' - \frac{\rho_{thnf}}{\rho_f} \frac{\mu_f}{\mu_{thnf}} f'^2 = 0, \quad (12)$$

$$\frac{1}{\text{Pr}} \left(\frac{k_{thnf}}{k_f} + R_d \right) \theta'' + \text{Ec} \xi^2 \frac{\mu_{thnf}}{\mu_f} f'^2 + \frac{k_{thnf}}{k_f} \frac{\rho_{thnf}}{\rho_f} \frac{\mu_f}{\mu_{thnf}} \frac{\text{Ec}}{\text{Pr}} [E^* f' + F^* \theta] + \text{Ec} \xi^2 \left(\lambda \frac{\mu_{thnf}}{\mu_f} + M \frac{\sigma_{thnf}}{\sigma_f} \right) f'^2 + \frac{(\rho c_p)_{thnf}}{(\rho c_p)_f} f \theta' = 0, \quad (13)$$

with related boundary conditions:

$$\text{– at } \eta = 0: \quad f' = 1, \quad f = 0, \quad \theta = 1, \quad (14a)$$

$$\text{– as } \eta \rightarrow \infty: \quad f' \rightarrow 0, \quad \theta \rightarrow 0. \quad (14b)$$

5.2. Second-level truncation

Introducing new variables $g = \partial f / \partial \xi$ and $h = \partial \theta / \partial \xi$, and differentiating Eqs. (6)–(8) over ξ , and assuming $\xi \ll 1$, yields:

$$g''' - \left(\lambda + M \frac{\mu_f}{\mu_{thnf}} \frac{\sigma_{thnf}}{\sigma_f} \right) g' + \frac{\mu_f}{\mu_{thnf}} \frac{(\rho\beta)_{thnf}}{(\rho\beta)_f} \xi^{-1} \alpha (h - \xi^{-1} \theta) + \frac{\rho_{thnf}}{\rho_f} \frac{\mu_f}{\mu_{thnf}} (2g f'' + f g'') - 3 \frac{\rho_{thnf}}{\rho_f} \frac{\mu_f}{\mu_{thnf}} f' g' + \frac{\mu_f}{\mu_{thnf}} \frac{\rho_{thnf}}{\rho_f} \xi (g'^2 - g g'') = 0, \quad (15)$$

$$\begin{aligned} & \frac{1}{\text{Pr}} \left(\frac{k_{thnf}}{k_f} + R_d \right) h'' + 2\text{Ec} \xi \frac{\mu_{thnf}}{\mu_f} f''^2 + 2\text{Ec} \xi^2 \frac{\mu_{thnf}}{\mu_f} f'' g'' + \\ & + \frac{k_{thnf}}{k_f} \frac{\rho_{thnf}}{\rho_f} \frac{\mu_f}{\mu_{thnf}} \frac{\text{Ec}}{\text{Pr}} [E^* g' + F^* h] + \\ & + 2\text{Ec} \xi \left(\lambda \frac{\mu_{thnf}}{\mu_f} + M \frac{\sigma_{thnf}}{\sigma_f} \right) f'^2 + \\ & + 2\text{Ec} \xi^2 \left(\lambda \frac{\mu_{thnf}}{\mu_f} + M \frac{\sigma_{thnf}}{\sigma_f} \right) f' g' + \\ & + \frac{(\rho c_p)_{thnf}}{(\rho c_p)_f} (f h' + 2g \theta' - f' h) - \frac{(\rho c_p)_{thnf}}{(\rho c_p)_f} \xi [g' h - g h'] = 0, \end{aligned} \quad (16)$$

with related boundary conditions:

$$\text{– at } \eta = 0: \quad g' = 0, \quad g = 0, \quad h = 0, \quad (17a)$$

$$\text{– as } \eta \rightarrow \infty: \quad g' \rightarrow 0, \quad h \rightarrow 0. \quad (17b)$$

6. Results and discussion

Numerical solutions for the dimensionless partial differential equations (Eqs. (6) and (7)) and the designated boundary conditions (Eqs. (8a), (8b)) are meticulously derived using the local non-similarity method, which is implemented up to a second truncation level with the bvp4c MATLAB toolkit. This study investigates the influence of the magnetic number and porosity parameter in a porous medium containing various working fluids, including Al_2O_3 + water, Al_2O_3 + TiO_2 + water, and Al_2O_3 + TiO_2 + MgO + water. The properties of water, aluminium oxide, titanium oxide and magnesium oxide, the models of thermo-physical properties, and computed values of $\text{Re}^{1/2} C_f$ and $\text{Re}^{-1/2} \text{Nu}$ are shown in Tables 1–4, respectively.

Table 1. Properties of the water, aluminium oxide (Al_2O_3), titanium oxide (TiO_2) and magnesium oxide (MgO) [30,33].

Materials	ρ , kg/m ³	K , W/(m·K)	σ , Ωm	β , 1/K	c_p , J/(kg·K)
Aluminium oxide (Al_2O_3)	3970	40	$35 \cdot 10^{-6}$	$0.85 \cdot 10^{-5}$	765
Titanium oxide (TiO_2)	4250	8.9538	$1 \cdot 10^{-12}$	$0.9 \cdot 10^{-5}$	686.6
Magnesium oxide (MgO)	3560	45	$2.6 \cdot 10^{-6}$	$1.05 \cdot 10^{-5}$	955
Water (H_2O)	997.1	0.613	$5.5 \cdot 10^{-6}$	$21 \cdot 10^{-5}$	4179

To evaluate the proposed methods' effectiveness and precision, an extensive comparison was made between the computational results and the findings of several esteemed authors, as shown in Table 5:

- Tsai et al. [35] utilised the Chebyshev finite difference method, whereas Giresha et al. [36] applied the Runge-Kutta-Fehlberg method for their solutions,
- the present research employs the local non-similarity method up to second-level truncation using the bvp4c MATLAB package.

Table 2. Models of thermo-physical properties [34].

Viscosity	$\mu_{(Al_2O_3+TiO_2+MgO)water} = \frac{(Al_2O_3 + TiO_2)water}{(1 - \phi_{Al_2O_3})^{2.5}(1 - \phi_{TiO_2})^{2.5}(1 - \phi_{SiO_2})^{2.5}}$
Density	$\rho_{(Al_2O_3+TiO_2+MgO)water} = (1 - \phi_{MgO})((1 - \phi_{TiO_2})(1 - \phi_{Al_2O_3})\rho_{water} + \phi_{Al_2O_3}\rho_{Al_2O_3} + \phi_{TiO_2}\rho_{TiO_2}) + \phi_{MgO}\rho_{MgO}$
Heat capacitance	$(\rho c_p)_{(Al_2O_3+TiO_2+MgO)water} = (1 - \phi_{MgO})\left((1 - \phi_{TiO_2})\left((1 - \phi_{Al_2O_3})(\rho c_p)_{water} + \phi_{Al_2O_3}(\rho c_p)_{Al_2O_3}\right) + \phi_{TiO_2}(\rho c_p)_{TiO_2}\right) + \phi_{MgO}(\rho c_p)_{MgO}$
Thermal expansion	$(\rho\beta)_{(Al_2O_3+TiO_2+MgO)water} = (1 - \phi_{MgO})\left((1 - \phi_{TiO_2})\left((1 - \phi_{Al_2O_3})(\rho\beta)_{water} + \phi_{Al_2O_3}(\rho\beta)_{Al_2O_3}\right) + \phi_{TiO_2}(\rho\beta)_{TiO_2}\right) + \phi_{MgO}(\rho\beta)_{MgO}$
Electric conductivity	$\frac{\sigma_{(Al_2O_3+TiO_2+MgO)water}}{\sigma_{(Al_2O_3+TiO_2)water}} = \frac{\sigma_{MgO} + 2\sigma_{(Al_2O_3+TiO_2)water} - 2\phi_{MgO}(\sigma_{(Al_2O_3+TiO_2)water} - \sigma_{MgO})}{\sigma_{MgO} + 2\sigma_{(Al_2O_3+TiO_2)water} + \phi_{MgO}(\sigma_{(Al_2O_3+TiO_2)water} - \sigma_{MgO})}$
Thermal conductivity	$\frac{k_{(Al_2O_3+TiO_2+SiO_2)water}}{k_{(Al_2O_3+TiO_2)water}} = \frac{k_{MgO} + 2k_{(Al_2O_3+TiO_2)water} - 2\phi_{MgO}(k_{(Al_2O_3+TiO_2)water} - k_{MgO})}{k_{MgO} + 2k_{(Al_2O_3+TiO_2)water} + \phi_{MgO}(k_{(Al_2O_3+TiO_2)water} - k_{MgO})}$

Table 3. Computed values of $Re^{1/2}C_f$ for various values of M , λ , α .

M	λ	α	$Re^{1/2}C_f$		
			Al_2O_3	$Al_2O_3 + TiO_2$	$Al_2O_3 + TiO_2 + MgO$
0.3	0.3	0.5	-1.5432014126	-1.2943728742	-1.0885117924
0.5			-1.9414080301	-1.6095879270	-1.3361500205
0.7			-2.2746152535	-1.8755502640	-1.5468988871
0.5	0.3	0.5	-1.9414080301	-1.6095879270	-1.3361500205
	0.6		-1.9727973884	-1.6471243162	-1.3809461220
	1.0		-2.0139344766	-1.6959626978	-1.4386415544
		0.5	-1.9414080301	-1.6095879270	-1.3361500205
		1.0	-1.8650650802	-1.5466559231	-1.2855071000
		1.5	-1.2312155521	-1.4788166666	-1.7825933276

Table 4. Computed values of $Re^{-1/2}Nu$ for various values of M , λ , α , R_d , Ec , E^* and F^* ; $\phi_{Al_2O_3} = \phi_{TiO_2} = \phi_{MgO} = 0.1$

M	λ	R_d	Ec	E^*	F^*	$Re^{-1/2}Nu$		
						$Al_2O_3 + water$	$Al_2O_3 + TiO_2 + water$	$Al_2O_3 + TiO_2 + MgO + water$
0.5	0.3	1.0	0.2	0.1	0.1	-0.6979895966	-0.6954793018	-0.7170713161
1.0						-0.7986160421	-0.7536400412	-0.7074828826
1.5						-0.8978679485	-0.8674086293	-0.8187923402
	0.3					-0.6979895966	-0.6954793018	-0.7170713161
	0.6					-0.5307828279	-0.6580774445	-0.6715155841
	1.0					-0.4997764931	-0.6145833652	-0.6192276302
		0.5				-0.4894079681	-0.4841726931	-0.5147548449
		1.0				-0.6979895966	-0.6954793018	-0.7170713161
		1.5				-0.8494338404	-0.8480060767	-0.8649297428
			0.2			-0.6979895966	-0.6954793018	-0.7170713161
			0.3			-0.6820049371	-0.6787217611	-0.6885904197
			0.4			-0.6661576762	-0.6620799098	-0.6601762966
				0.1		-0.6979895966	-0.6954793018	-0.7170713161
				0.3		-0.7154796303	-0.7148261966	-0.7170661331
				0.5		-0.7328774635	-0.7340835802	-0.7170609501
					0.1	-0.6979895966	-0.6954793018	-0.7170713161
					0.3	-0.7083172021	-0.7064924464	-0.7170685399
					0.5	-0.7185305620	-0.7173727386	-0.7170657638

Table 5. The comparison of dimensionless temperature gradient $\theta'(0)$ for the case of $M = \lambda = \alpha = Ec = R_d = E^* = 0$ and $\phi_{Al_2O_3} = \phi_{TiO_2} = \phi_{MgO} = 0$.

Pr	F^*	Tsai et al. [36], Chebyshev finite difference method	Giresha et al. [37], Runge-Kutta-Fehlberg method	Present study, local non-similarity method
2	-2	-2.4859	-2.4859	-2.4860961811
3	-3	-3.0281	-3.0281	-3.0730949222
4	-4	-3.5851	-3.5851	-3.5679155162

The influence of the buoyancy parameter α on the velocity, represented by $\frac{\partial f}{\partial \eta}(\xi, \eta)$, in the context of nanoparticle presence, is vividly illustrated in Fig. 2.

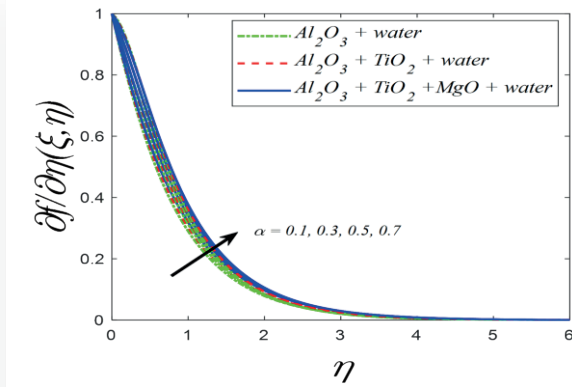


Fig. 2. The effect of buoyancy parameter α on velocity profile; $\lambda = 0.1$, $M = 1.0$, $E^* = F^* = 0.5$, $R_d = 1.3$ and $Ec = 0.1$.

Notably, the figure conspicuously elucidates that augmenting the value of α leads to a discernible escalation in the fluid velocity $\frac{\partial f}{\partial \eta}(\xi, \eta)$. Furthermore, the thickness of the momentum boundary layer was observed to increase as the values of the buoyancy parameter δ were enhanced. Variations in δ corre-

spond to variations in distance from the surface. The data indicate that the buoyancy force acting as an assistive factor exhibits characteristics like those of a favourable pressure gradient. The buoyancy force has a significant impact on the movement of liquid through a consolidated permeable medium. $Al_2O_3 + TiO_2 + MgO + water$, in comparison to $Al_2O_3 + TiO_2 + water$, is more sensitive to changes in the buoyancy parameter.

Figures 3a and 3b illustrate the impact of λ on the distribution of three types of nanofluids: $Al_2O_3 + TiO_2 + MgO + water$, $Al_2O_3 + TiO_2 + water$, and $Al_2O_3 + water$. As the value of λ is increased, there is a decrease in velocity $\frac{\partial f}{\partial \eta}(\xi, \eta)$ while the thermal profile $\theta(\xi, \eta)$ rises. From a physical standpoint, the presence of a permeable medium results in an increase in flow resistance, leading to a reduction in both flow velocity and flow rate. Consequently, the fluid's temperature rises because of the heating sheet's energy transfer to the medium. Therefore, the thickness of the boundary layer decreases as a function of the surface layer's velocity. The resultant layer gradually thins out. The velocity distribution is observed to be greater for $Al_2O_3 + water$ and lesser for $Al_2O_3 + TiO_2 + MgO + water$, whereas the temperature profile exhibits an opposite trend.

Figures 4a and 4b demonstrate the influence of the magnetic number (M) on the distributions of velocity $\frac{\partial f}{\partial \eta}(\xi, \eta)$ and temperature $\theta(\xi, \eta)$ in $Al_2O_3 + TiO_2 + MgO + water$, $Al_2O_3 + TiO_2 + water$, and $Al_2O_3 + water$.

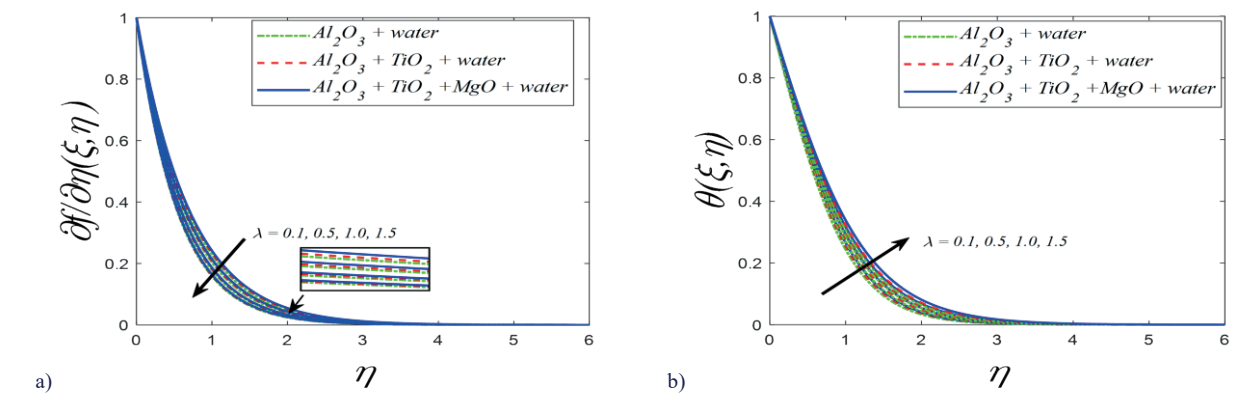


Fig. 3. The effect of porosity parameter λ on (a) velocity and (b) temperature profile; $\alpha = 0.1$, $M = 1.0$, $E^* = F^* = 0.5$, $R_d = 1.3$ and $Ec = 0.1$.

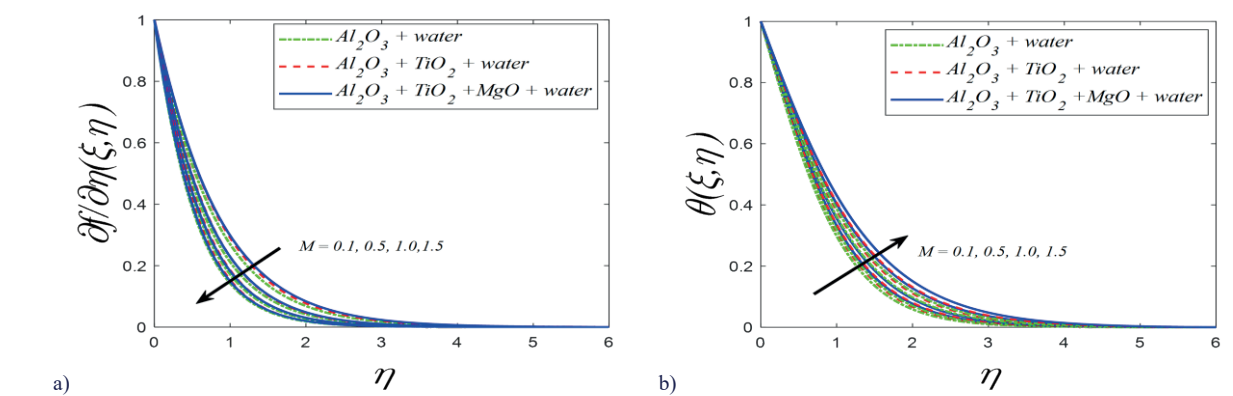


Fig. 4. The effect of magnetic number M on (a) velocity and (b) temperature profile; $\alpha = 0.1$, $\lambda = 0.1$, $E^* = F^* = 0.5$, $R_d = 1.3$ and $Ec = 0.1$.

As M increases with time, the temperature of the nanofluid also increases; on the other hand, the velocity distributions are found to decrease. The magnetic field is applied outside the fluid and perpendicular to the fluid flow, and its effect on the velocity and temperature of nanoparticles in the fluid is evaluated. The Lorentz force, which occurs due to the magnetic field, provides resistance which impedes and decelerates the movement of fluid particles and consequently decreases fluid velocity. The resistance force caused by the magnetic field reinforcement increases the thickness of the boundary layer and the density of nanoparticles. Consequently, the temperature distribution becomes more pronounced. When M increases, the nanofluid exhibits an increase in velocity. Both $\text{Al}_2\text{O}_3 + \text{TiO}_2/\text{water}$ and $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{MgO}/\text{water}$ exhibit an elevated temperature profile. Furthermore, the temperature distribution for the ternary $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{MgO}/\text{water}$ nanofluid increases more noticeably than that of $\text{Al}_2\text{O}_3 + \text{TiO}_2/\text{water}$ and $\text{Al}_2\text{O}_3/\text{water}$ nanofluids.

Figures 5 and 6 have been created to investigate the impact of heating sources (sinks) E^* and F^* on the temperature field $\theta(\xi, \eta)$.

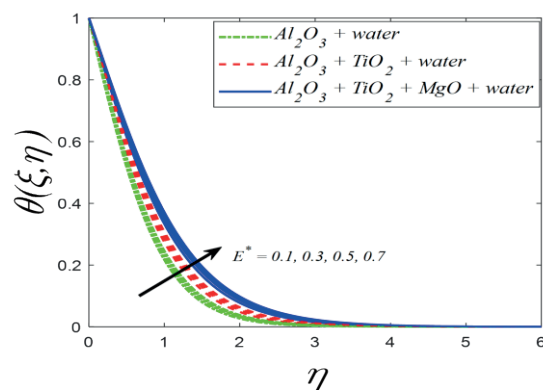


Fig. 5. The effect of E^* on the temperature profile; $\alpha = 0.1, \lambda = 0.1, M = 1.0, F^* = 0.5, R_d = 1.3$ and $Ec = 0.1$.

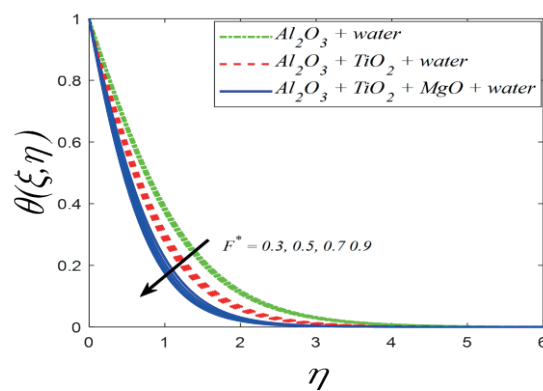


Fig. 6. The effect of F^* on the temperature profile; $\alpha = 0.1, \lambda = 0.1, M = 1.0, E^* = 0.5, R_d = 1.3$ and $Ec = 0.1$.

It is a well-established fact that, in the case of heat absorption E^* , liquid shifts away from the surface and moves towards the wall in a vertical direction in the case of heat generation F^* . The system absorbs heat in the state of E^* . Both E^* and F^* operate based on the premise of an endothermic or exothermic reaction,

where the scheme either absorbs or releases heat. The process of photosynthesis is a prime example of heat absorption, wherein plants absorb heat to synthesise their food. The nuclear fission reaction exemplifies a phenomenon of heat generation whereby heat is released through a specific process. The increase in volume fraction of nanoparticles leads to an increase in intermolecular collisions among fluid molecules, resulting in a corresponding change in fluid properties. The distribution of thermal conductivity and temperature is uniform throughout the fluid flowing in an upward direction. The upward change is more prominent in the case of $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{MgO} + \text{water}$ compared to $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{water}$ and $\text{Al}_2\text{O}_3 + \text{water}$, as depicted in Fig. 5. It has been observed that there is an inverse relationship between the thickness of a fluid and its temperature. It has been observed that an increase in fluid viscosity results in a decrease in fluid temperature, which causes more heat to be released. This ultimately leads to a reduction in fluid temperature in the event of an increase in F^* , as depicted in Fig. 6.

Figure 7 elucidates the significant effects of thermal radiation on the temperature distributions in $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{MgO} + \text{water}$, $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{water}$, and $\text{Al}_2\text{O}_3 + \text{water}$, where temperature increments scale linearly with the radiation parameter R_d , highlighting radiation as a dominant heat transfer pathway that extracts energy from the vibrational and translational kinetics of liquid particles to generate thermal energy through fluid agitation and consequent heating. This mechanism is expected to increase nanoparticle concentrations in the boundary layer according to radiation intensity, causing localised heat buildup.

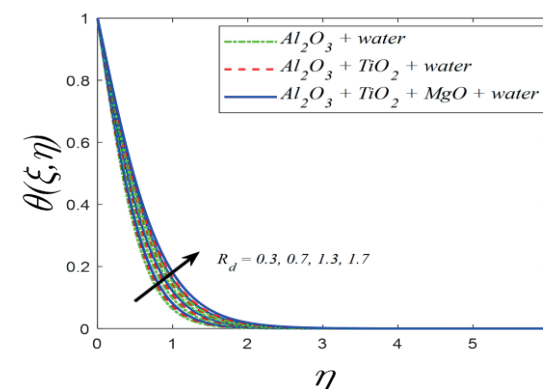


Fig. 7. The effect of R_d on the temperature profile; $\alpha = 0.1, \lambda = 0.1, M = 1.0, E^* = F^* = 0.5$ and $Ec = 0.1$.

Graphical data substantiate that nanofluids exhibit the greatest thermal affinity, exceeding $\text{Al}_2\text{O}_3 + \text{water}$ and $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{water}$ and $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{MgO} + \text{water}$ in heat retention, unlocking myriad applications across industries: in solar thermal systems, nanoparticle-water enhancements could propel collector efficiencies via superior radiative uptake in sun-rich environments, fostering sustainable energy transitions; electronics cooling benefits from hybrid nanofluid boundary layer synergies in microchannel designs, averting thermal failures in compact devices.

The impact of the Eckert number Ec on the temperature distribution is illustrated in Fig. 8.

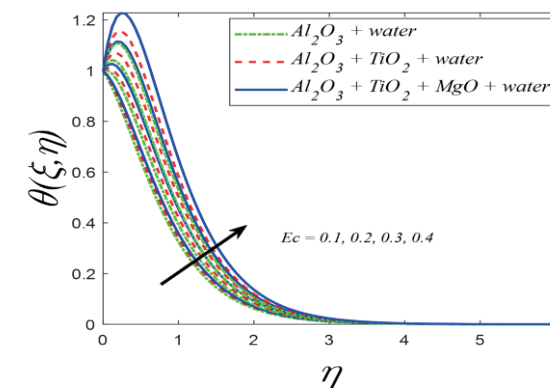


Fig. 8. The effect of Ec on the temperature profile; $\alpha = 0.1, \lambda = 0.1, M = 1.0, E^* = F^* = 0.5, R_d = 1.3$.

Empirical evidence clearly shows that better results occur as water temperatures rise. The generation of frictional heating has been attributed to the amplification in question, as per recent findings. The ratio of the difference in kinetic energy to the change in enthalpy between a solid boundary and a fluid is quantified by the Eckert number, which is a nondimensional quantity. Any kind of fluid may have its kinetic energy transformed into stored energy using a newly proposed process. This process is achieved through the performance of work against viscous strains incrementally. This phenomenon is observable. The figure illustrates that the impact of $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{MgO} + \text{water}$ on temperature distribution is more significant in comparison to that of $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{water}$ and $\text{Al}_2\text{O}_3 + \text{water}$.

6. Conclusions

This current study endeavours to assess the influence of thermal radiation and Ohmic heating on the mixed convection boundary-layer flow of a magnetised nanofluid over a horizontal surface. This surface is situated within a porous medium and is under steady-state conditions. The governing boundary layer equations undergo a process of non-dimensionalisation, leading to the emergence of a challenging system of nonlinear partial differential equations. The equations are then solved using numerical solution methods in MATLAB, which are based on the powerful nature of the bvp4c toolset and instead use the local non-similarity technique to do so. The parameters which influence the flow and heat transfer characteristics have been explained in the study, including λ , α , M and R_d . According to the results of the present research, it is possible to draw the following conclusions:

1. The precision and verification of the present study have been provided in tabular format, as compared to the previous literature published, which used different numerical techniques and methods.
2. The graphs indicate the influence of M on the velocity and temperature profile. As the magnetic number increases, the velocity profile decreases and the heat profile increases. The velocity profile also decreases more with $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{MgO} + \text{water}$ compared to $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{water}$ and $\text{Al}_2\text{O}_3 + \text{water}$, and more control is exerted by $\text{Al}_2\text{O}_3 + \text{TiO}_2$

+ $\text{MgO} + \text{water}$. Similarly, it is observed that the heat transfer rate is higher in both $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{water}$ and in $\text{Al}_2\text{O}_3 + \text{water}$, but the drag force is lower in $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{MgO} + \text{water}$.

3. The porosity parameter affects the thermal profile more for $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{MgO} + \text{water}$ as compared to $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{water}$ and $\text{Al}_2\text{O}_3 + \text{water}$, which affects the velocity profile less. During an increase, the rate of heat transfer decreases, whereas the drag coefficient decreases more slowly in $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{MgO} + \text{water}$ than in $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{water}$.
4. As the values of R_d and Ec increase, the thermal profile and the rate of heat transfer become increased, but the influence of $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{MgO} + \text{water}$ is much stronger than that of $\text{Al}_2\text{O}_3 + \text{TiO}_2 + \text{water}$ and $\text{Al}_2\text{O}_3 + \text{water}$.

References

- [1] Choi, S.U., & Eastman, J.A. (1995). Enhancing thermal conductivity of fluids with nanoparticles. *ASME International Mechanical Engineering Congress & Exposition, November 12–17, San Francisco, USA*. Rep. No. ANL/MSD/CP-84938; CONF-951135-29. doi: 10.1115/IMECE1995-0926
- [2] Buongiorno, J. (2006). Convective transport in nanofluids. *Journal of Heat and Mass Transfer*, 128(3), 240–250. doi: 10.1115/1.2150834
- [3] Mabood, F., Khan, W., & Ismail, A. (2015). MHD boundary layer flow and heat transfer of nanofluids over a nonlinear stretching sheet: A numerical study. *Journal of Magnetism and Magnetic Materials*, 374, 569–576. doi: 10.1016/j.jmmm.2014.09.013
- [4] Siddiqui, B.K., Batool, S., ul Hassan, Q.M., & Malik, M.Y. (2021). Irreversibility analysis in the boundary layer MHD two dimensional flow of Maxwell nanofluid over a melting surface. *Ain Shams Engineering Journal*, 12(3), 3217–3227. doi: 10.1016/j.asej.2021.01.017
- [5] Masood, S., Farooq, M., & Anjum, A. (2021). Influence of heat generation/absorption and stagnation point on polystyrene– $\text{TiO}_2/\text{H}_2\text{O}$ hybrid nanofluid flow. *Scientific Reports*, 11(1), 22381. doi: 10.1038/s41598-021-01747-9
- [6] Hussain, I., Hobiny, A., Irfan, M., Tabrez, M., & Khan, W. (2023). Impact of magnetic dipole contribution on radiative ferromagnetic Cross nanofluid with viscous dissipation aspects. *Journal of Magnetism and Magnetic Materials*, 579, 170706. doi: 10.1016/j.jmmm.2023.170706
- [7] Chen, T.S., Sparrow, E.M., & Mucoglu, A. (1977). Mixed convection in boundary layer flow on a horizontal plate. *Journal of Heat and Mass Transfer*, 99(1), 66–71. doi: 10.1115/1.3450657
- [8] Chamkha, A.J., Al-Mudhaf, A.F., & Pop, I. (2006). Effect of heat generation or absorption on thermophoretic free convection boundary layer from a vertical flat plate embedded in a porous medium. *International Communications in Heat and Mass Transfer*, 33(9), 1096–1102. doi: 10.1016/j.icheatmasstransfer.2006.04.009
- [9] Ahmad, S., & Pop, I. (2010). Mixed convection boundary layer flow from a vertical flat plate embedded in a porous medium filled with nanofluids. *International Communications in Heat and Mass Transfer*, 37(8), 987–991. doi: 10.1016/j.icheatmasstransfer.2010.06.004v
- [10] Mahanthesh, B., Gireesha, B.J., & Animasaun, I.L. (2018). Exploration of non-linear thermal radiation and suspended nanoparticles effects on mixed convection boundary layer flow of nanoliquids on a melting vertical surface. *Journal of Nanofluids*, 7(5), 833–843. doi: 10.1166/jon.2018.1521v

- [11] Cao, W., Animasaun, I.L., Yook, S.J., Oladipupo, V.A., & Ji, X. (2022). Simulation of the dynamics of colloidal mixture of water with various nanoparticles at different levels of partial slip: Ternary-hybrid nanofluid. *International Communications in Heat and Mass Transfer*, 135, 106069. doi: 10.1016/j.icheatmasstransfer.2022.106069
- [12] Jakeer, S., Reddy, S.R.R., Rashad, A.M., Rupa, M.L., & Manjula, C. (2023). Nonlinear analysis of Darcy-Forchheimer flow in EMHD ternary hybrid nanofluid (Cu-CNT-Ti/water) with radiation effect. *Forces in Mechanics*, 10, 100177. doi: 10.1016/j.finmec.2023.100177
- [13] Lauriat, G., & Prasad, V. (1987). Natural convection in a vertical porous cavity: a numerical study for Brinkman-extended Darcy formulation. *Journal of Heat and Mass Transfer*, 109(3), 688–696. doi: 10.1115/1.3248143
- [14] Beckermann, C., Viskanta, R., & Ramadhyani, S. (1986). A numerical study of non-Darcian natural convection in a vertical enclosure filled with a porous medium. *Numerical Heat Transfer*, 10(6), 557–570. doi: 10.1080/10407788608913535
- [15] Cortell, R. (2005). Flow and heat transfer of a fluid through a porous medium over a stretching surface with internal heat generation/absorption and suction/blowing. *Fluid Dynamics Research*, 37(4), 231–245. doi: 10.1016/j.fluidyn.2005.05.001
- [16] Salahuddin, T., Akram, A., Khan, M., Siddique, N., Alaoui, M.K., & Aly, S. (2020). A comparative study of $\gamma\text{Al}_2\text{O}_3\text{--C}_2\text{H}_6\text{O}_2$ and $\gamma\text{Al}_2\text{O}_3\text{--H}_2\text{O}$ near a vertical curved surface having porous medium. *International Journal of Hydrogen Energy*, 45(46), 24275–24284. doi: 10.1016/j.ijhydene.2020.06.120
- [17] Hussain, M., Rahim, I., Jan, A., Ahmad, M., & Khan, M.R. (2025). Comparative analysis of the radiative stagnation point nanofluid flow with the considerations of inclined magnetic field and porous medium. *Journal of Thermal Analysis and Calorimetry*, 150, 22597–22605. doi: 10.1007/s10973-025-15219-z
- [18] Keshavarzian, B., & Sayehvand, H.O. (2023). Validation of the local thermal equilibrium assumption for free convection boundary layer flow over a horizontal cylinder embedded in an infinite saturated porous medium. *Results in Physics*, 44, 106112. doi: 10.1016/j.rinp.2022.106112
- [19] Sparrow E.M., & Cess, R.D. (1961). The effect of a magnetic field on free convection heat transfer. *International Journal of Heat and Mass Transfer*. 3(4), 267–274. doi: 10.1016/0017-9310(61)90042-4
- [20] Ellahi, R., Alamri, S.Z., Basit, A., & Majeed, A. (2018). Effects of MHD and slip on heat transfer boundary layer flow over a moving plate based on specific entropy generation. *Journal of Taibah University for Science*, 12(4), 476–482. doi: 10.1080/16583655.2018.1483795
- [21] Jan, A., Ahmad, A., & Aslam, F. (2025). Enhanced thermal performance of aggregated TiO_2 nanoparticles in non-Newtonian fluid flow over curved surfaces under magnetic and porous medium effects. *Journal of Thermal Analysis and Calorimetry*, 150(18), 14517–14529. doi: 10.1007/s10973-025-14516-x
- [22] Iqbal, J., & Abbasi, F.M. (2026). Analysis of entropy generation for Magnetohydrodynamics peristaltic motion of Carreau-Yasuda nanofluid through a curved channel with variable thermal conductivity and Joule heating. *Waves in Random and Complex Media*, 35(7), 12563–12582. doi: 10.1080/17455030.2022.2134603
- [23] Ghasemi, S.E., Mohsenian, S., Gouran, S., & Zolfagharian, A. (2022). A novel spectral relaxation approach for nanofluid flow past a stretching surface in presence of magnetic field and nonlinear radiation. *Results in Physics*, 32, 105141. doi: 10.1016/j.rinp.2021.105141
- [24] Iqbal, J., Abbasi, F.M., Alkinidri, M., & Alahmadi, H. (2023). Heat and mass transfer analysis for MHD bioconvection peristaltic motion of Powell-Eyring nanofluid with variable thermal characteristics. *Case Studies in Thermal Engineering*, 43, 102692. doi: 10.1016/j.csite.2022.102692
- [25] Sparrow, E.M., Quack, H., & Boerner, C.J. (1970). Local non-similarity boundary-layer solutions. *AIAA Journal*, 8(11), 1936–1942. doi: 10.2514/3.6029
- [26] Sparrow, E.M., & Yu, H.S. (1971). Local non-similarity thermal boundary-layer solutions. *ASME. Journal Heat and Mass Transfer*, 93(4), 328–334. doi: 10.1115/1.3449827
- [27] Jan, A., Farooq, U., Khan, M.I., & Aslam, F. (2025). Applications of neural networking analysis on drag reduction and heat transfer rate of non-Newtonian fluid flows in energy storage systems. *International Communications in Heat and Mass Transfer*, 169, 109569. doi: 10.1016/j.icheatmasstransfer.2025.109569
- [28] Riaz, S., Naheed, N., Farooq, U., Lu, D., & Hussain, M. (2023). Non-similar investigation of magnetized boundary layer flow of nanofluid with the effects of Joule heating, viscous dissipation and heat source/sink. *Journal of Magnetism and Magnetic Materials*, 574, 170707. doi: 10.1016/j.jmmm.2023.170707
- [29] Anuar, N.S., Bachok, N., & Pop, I. (2021). Influence of buoyancy force on Ag-MgO/water hybrid nanofluid flow in an inclined permeable stretching/shrinking sheet. *International Communications in Heat and Mass Transfer*, 123, 105236. doi: 10.1016/j.icheatmasstransfer.2021.105236
- [30] Iqbal, Z., & Mehmood, Z. (2017). Elastic deformation analysis on MHD viscous dissipative flow of viscoelastic fluid: An exact approach. *Communications in Theoretical Physics*, 67(5), 561. doi: 10.1088/0253-6102/67/5/561
- [31] Chamkha, A.J., Rashad, M., & Subba Reddy Gorla, R. (2014). Non-similar solutions for mixed convection along a wedge embedded in a porous medium saturated by a non-Newtonian nanofluid: natural convection dominated regime. *International Journal of Numerical Methods for Heat & Fluid Flow*, 24(7), 1471–1486. doi: 10.1108/HFF-07-2012-0169
- [32] Jan, A., Khan, M.I., Farooq, U., Aslam, F., & Adnan, R. (2026). PINN-based numerical modeling of MHD flows in porous media over linear stretching boundaries. *International Communications in Heat and Mass Transfer*, 172, 110592. doi: 10.1016/j.icheatmasstransfer.2026.110592
- [33] Rajesh, V., Mallesh, M.P., & Bég, O.A. (2015). Transient MHD free convection flow and heat transfer of nanofluid past an impulsively started vertical porous plate in the presence of viscous dissipation. *Procedia Materials Science*, 10, 80–89. doi: 10.1016/j.mspro.2015.06.028v
- [34] Adnan, & Ashraf, W. (2022). Thermal efficiency in hybrid ($\text{Al}_2\text{O}_3\text{--CuO/H}_2\text{O}$) and ternary hybrid nanofluids ($\text{Al}_2\text{O}_3\text{--CuO--Cu/H}_2\text{O}$) by considering the novel effects of imposed magnetic field and convective heat condition. *Waves in Random and Complex Media*, 35(4), 8029–8044. doi: 10.1080/17455030.2022.2092233
- [35] Tsai, R., Huang, K.H., & Huang, J.S. (2008). Flow and heat transfer over an unsteady stretching surface with non-uniform heat source. *International Communications in Heat and Mass Transfer*, 35(10), 1340–1343. doi: 10.1016/j.icheatmasstransfer.2008.07.001
- [36] Gireesha, B.J., Ramesh, G.K., Abel, M.S., & Bagewadi, C.S. (2011). Boundary layer flow and heat transfer of a dusty fluid flow over a stretching sheet with non-uniform heat source/sink. *International Journal of Multiphase Flow*, 37(8), 977–982. doi: 10.1016/j.ijmultiphaseflow.2011.03.014



Co-published by
Institute of Fluid-Flow Machinery
 Polish Academy of Sciences
Committee on Thermodynamics and Combustion
 Polish Academy of Sciences

Copyright©2026 by the Authors under licence CC BY-NC-ND 4.0

<http://www.imp.gda.pl/archives-of-thermodynamics/>



Exergy-Based Performance Assessment of a Solar Air Heater Equipped with Zig-Zag Structured Baffles

Jaya Bohra^a, Vijay Singh Bisht^a, Prabhakar Bhandari^{b,c*}, Kshitij Pandey^d, Sandeep Singh^e, Ankur Singh Bist^f, Lalit Ranakoti^g, Shivasheesh Kaushik^h, Kapil Kumar Sharma^g

^aDepartment of Thermal Engineering, Faculty of Technology, Veer Madho Singh Bhandari Uttarakhand Technical University, Dehradun 248007, Uttarakhand, India

^bDepartment of Mechanical Engineering, K.R. Mangalam University, Gurugram 122103, Haryana, India

^cCentre of Excellence-Cyber Security, School of Engineering and Technology, K.R. Mangalam University, Gurugram 122103, Haryana, India

^dDepartment of Mechanical Engineering, Uttaranchal University, Dehradun 248007, Uttarakhand, India

^eO.P. Jindal Global University, Sonapat, 131001, Haryana, India

^fCSE, Graphic Era Hill University, Bhimtal Campus 263132, Uttarakhand, India

^gDepartment of Mechanical Engineering, Graphic Era Deemed to be University, Dehradun 248002, Uttarakhand, India

^hDepartment of Mechanical Engineering, Shivalik College of Engineering Dehradun 248197, Uttarakhand, India

*Corresponding author email: prabhakar.bhandari40@gmail.com

Received: 22.12.2025; revised: 19.03.2026; accepted: 30.03.2026

Abstract

By giving the absorber plate artificial roughness, the efficacy of a solar air heater may be increased. A mathematical study has been performed on solar air heaters roughened with Z-shaped baffles. With the help of MATLAB code, an exergy analysis, estimation of exergetic losses and efficiency have been made. The effect of relative pitch ratio for 45° zig-zag structured baffles with a fixed relative blockage height of 0.3 on various exergy parameters has also been studied. The current study has been done for Reynolds numbers ranging from 3000 to 14 000 and for 1100 W/m² of solar radiation. It is deduced that with the increase in Reynolds number, the effective efficiency enhances up to a certain point, beyond which it begins to decline. Moreover, the pitch ratio of 1.5 has shown the best performance up to the Reynolds number of 5000, while at higher Reynolds numbers, the pitch ratio of 3 has produced better results.

Keywords: Zig-zag structured baffles; Solar air heater; Absorber plate; Thermal attributes; Exergy investigation

Vol. 47(2026), No. 2, 151–157; doi: 10.24425/ather.2026.158674

Cite this manuscript as: Bohra, J., Bisht, V.S., Bhandari, P. Pandey, K., Singh, S., Bist, A.S., Ranakoti, L., Kaushik, S., & Sharma, K.K. (2026). Exergy-Based Performance Assessment of a Solar Air Heater Equipped with Zig-Zag Structured Baffles. *Archives of Thermodynamics*, 47(2), 151–157.

1. Introduction

Energy plays a significant part in each one's life and is available in numerous forms. As the population around the world and their energy requirements increase, the dependence on conventional fuel also increases. To cope with the demand, the world has started focusing more on renewable or non-conventional resources of energy. Among all types of energy, solar energy has shown the highest potential. A solar air-heater (SAH) is one of

the devices that uses sun powered energy. SAH has applications in very vast fields, like in air conditioning, crop drying, etc. [1–3]. As solar air heaters may have low heat transfer rates, researchers try to improve their performance using turbulators. There are several types of turbulators introduced, including inserts/ribs, baffles, fins, delta winglets, vortex generators, etc., that generate turbulence or vortices at walls, resulting in a favourable heat transfer rate with negligible pressure penalty [4,5]. Recent studies on nanoparticle-induced heating demonstrate that

Nomenclature

A	– surface area, m ²
C	– conversion factor
C_d	– coefficient of discharge
C_p	– specific heat of air, J/(kg·K)
D	– hydraulic diameter, m
e	– rib height, m
E_{LA}	– exergy losses by absorption of irradiation by the absorber, W
E_{Le}	– exergy losses by both radiative and convective heat transfer from the absorber, W
E_{LAP}	– exergy losses by friction, W
E_{LAT}	– exergy losses by the heat transfer to the working fluid, W
f	– friction factor for roughened duct
F_o	– heat removal factor
h	– heat transfer coefficient, W/(m ² ·K)
H	– height of duct, m
I	– solar radiation intensity, W/m ²
k	– thermal conductivity, W/(m·K)
L	– length of test section in duct, m
M	– mass flow rate of air, kg/s
N	– number of glass covers
Nu	– Nusselt number
P	– pitch, m
P_m	– pumping power, W
Pr	– Prandtl number
PR	– relative roughness pitch
Q_u	– useful heat gain, W
Re	– Reynolds number
t	– thickness, m
T	– temperature, K
U_L	– head loss coefficients, W/(m ² ·K)
V	– mean flow velocity, m/s
W	– width of duct, m

Greek symbols

α	– flow angle of attack, °
β	– collector inclination to horizontal, °
δ_i	– thickness of insulation, m
ΔP	– pressure drop, N/m ²
ΔT	– temperature drop, K
ε	– emissivity
η_c	– Carnot efficiency
η_{exe}	– exergetic efficiency
η_{th}	– thermal efficiency of solar collector
λ	– wavelength of incident ray, μm
μ	– dynamic viscosity of air, N s/m ²
ν	– kinematic viscosity, m ² /s
ρ_a	– density of air at mean bulk temperature, kg/m ³
σ	– Stefan–Boltzmann constant, W/(m ² ·K ⁴)
$(\tau\alpha)$	– transmittance–absorptance product

Subscripts and Superscripts

a	– ambient
b	– tail
e	– collector edge
f	– fluid
g	– glass cover
i	– inlet
m	– mean
o	– outlet
p	– absorber plate
r	– roughened
s	– smooth
t	– head
w	– wind

Abbreviations and Acronyms

SAH – solar air heater

optimised materials surface and geometry can significantly enhance heat generation, distribution and convective transport, offering useful design insights for improving thermal performance in solar air heater systems [6,7].

Motivated by the simple yet versatile geometrical characteristics of English alphabet shapes, numerous researchers have investigated the use of alphabet-patterned ribs as artificial roughness elements on the absorber plates of SAH. These rib configurations, modelled after different alphabetic profiles, are designed to disturb the laminar sublayer, enhance turbulence, and promote effective mixing of the air stream near the heated surface [8]. Lau et al. [9] studied V-shaped ribs at various attack angles and observed that 45° and 60° ribs (of relative pitch equal to 10) significantly enhanced heat transfer and flow characteristics compared to smooth and fully-angled ribs, whereas 120° and 135° ribs produced lower heat transfer rates and higher friction losses. They also reported that roughening both opposing walls does not improve overall duct performance. Singh et al. [10,11] studied perforated multi-V-profile ribs in solar air heaters for Reynolds numbers of 2000–18 000, roughness widths of 2–10, and open area ratios up to 0.31. The best performance was obtained at a roughness width of 6 and an open area ratio of 0.27,

and they proposed Nusselt and friction factor correlations with accuracies of 12% and 7.5%, respectively.

Lanjewar et al. [12] experimentally showed that W-shaped wire ribs provide better thermo-hydraulic performance than V-shaped ribs in solar air heaters. Ghildyal et al. [13] reported that U-shaped turbulators achieved the highest Nusselt number and overall performance, while D-shaped turbulators produced the greatest friction. In a later study, Ghildyal et al. [14] found that W-contoured turbulators delivered the highest thermal efficiency compared with tapered and reverse-tapered designs (Re = 4000–18 000). Chaudhari et al. [15] reported that inverted L-shaped ribs with a pitch of 17.86 achieved optimal exergy efficiency at 1000 W/m², while Kumar et al. [16] showed performance improvement using a hybrid S-shaped rib-protrusion design. Alhilali et al. [17] observed 2.5–3.0 times higher Nusselt numbers and 2.8–4.4 times greater friction factors with I-shaped ribs in a two-dimensional numerical study of solar air heaters.

Sriromreun et al. [18] experimentally investigated the thermo-hydraulic performance of a rectangular duct with 45° Z-shaped baffles, comparing in-phase and out-of-phase arrangements, as shown in Fig. 1. The in-phase configuration produced stronger turbulence and higher heat transfer rates, but also in-

creased friction losses. Based on the results, empirical correlations for the Nusselt number and friction factor were developed for similar roughened duct designs. Bohra et al. [19] investigated the influence of geometric parameters, i.e. blockage ratio on the thermal performance of SAH equipped with Z-shaped baffles mounted on the absorber plate. They reported the best thermal performance achieved at a blockage ratio of 0.3.

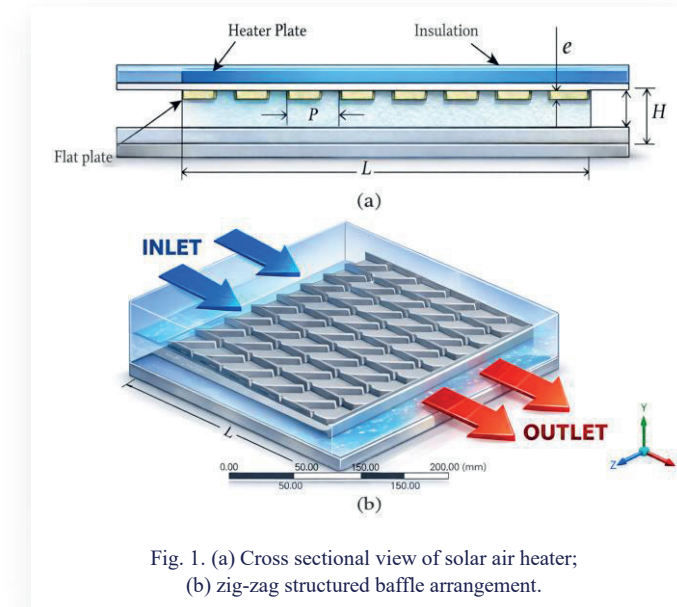


Fig. 1. (a) Cross sectional view of solar air heater; (b) zig-zag structured baffle arrangement.

The present research work introduces a pioneering investigation into the synergistic impact of exergy analysis and the implementation of zig-zag structured baffles in solar air heater systems. Unlike conventional studies that primarily rely on energy analysis, this research employs exergy analysis, providing a more comprehensive evaluation of energy utilisation and losses. The integration of zig-zag structured baffles represents a novel design modification aimed at optimising heat transfer within the solar air heater. The innovative zig-zag configuration promotes turbulent flow, thereby enhancing the interaction between the air and absorber plate. By combining these two innovative elements, the research aims to achieve a breakthrough in solar air heater efficiency, offering a holistic and advanced approach to the design and optimisation of sustainable solar heating technologies.

The primary objective of the research is to systematically investigate and quantify the impact of incorporating zig-zag structured baffles having different relative roughness pitch on the performance of solar air heater systems using exergy analysis. Through this research, the goal is to provide a comprehensive understanding of the flow of energy, losses and efficiencies within the solar air heater, surpassing traditional energy analyses. The specific focus on exergy analysis aims to identify and address inefficiencies and opportunities for improvement that may not be apparent in conventional energy assessments. Additionally, by introducing the novel element of zig-zag structured baffles, the research seeks to assess and optimise heat transfer mechanisms within the solar air heater, ultimately contributing valuable insights towards enhancing the overall efficiency and effectiveness of solar air heating technologies.

2. Materials and methods

In the current attempt, exergetic evaluation has been investigated for the solar heater having zig-zag structured baffles in the plate. Table 1 gathers the information on SAH characteristics and the operating constraints.

Table 1. Boundary conditions, variables and operational conditions employed in analytical computations.

Parameters	Range of values
Rectangular channel total length (L)	700 mm
Channel width (W)	300 mm
Channel height (H)	25 mm
Transmittance-absorptance ($\tau\alpha$)	0.8
Emissance of glass (e_g)	0.88
Emissance of plate (e_p)	0.9
Glass cover thickness (t_g)	4 mm
Number of glass covers (N)	1
Thickness of insulation (t_i)	50 mm
Thermal conductivity of insulation (K_i)	0.037 W/(m K)
Wind velocity (V_w)	1 m/s
Relative roughness pitch (PR)	1.5-3
Uniform heat flux or insolation (I)	1000 W/m ²
Inlet temperature of air (T_a)	300K

MATLAB program has been used to evaluate different exergetic parameters. The correlations for the Nusselt number (Nu) and friction factor (f) for the duct of rectangular shape having zig-zag structured baffles are shown in Eqs. (1) and (2), respectively [18]

$$Nu = 0.29023Re^{0.669891}Pr^{0.4}\left(\frac{e}{H} + 1\right)^{4.854282}Pr^{0.4}\left(\frac{P}{H} + 1\right)^{-0.47904}, \quad (1)$$

$$f = 0.237223Re^{-0.0726}\left(\frac{e}{H} + 1\right)^{9.859262}\left(\frac{P}{H} + 1\right)^{-0.86932}. \quad (2)$$

The total or overall loss coefficient is expressed as:

$$U_L = U_t + U_b + U_e, \quad (3)$$

where U_t , U_b and U_e are the head loss coefficient, tail loss coefficient and side loss coefficient, respectively.

The useful gain in energy is expressed as:

$$Q_{u1} = [I(\tau\alpha) - U_L(T_{ap} - T_s)]A_p. \quad (4)$$

where T_{ap} and T_s are the temperatures of the absorber plate and the surroundings.

Equation (5) expresses another method for estimating a usable energy gain:

$$Q_{u2} = F_o[I(\tau\alpha) - U_L(T_o - T_i)]A_p, \quad (5)$$

where F_o denotes the heat removal factor, T_o and T_i are the temperatures of the outflow and inlet air, respectively.

Ideally, the values Q_{u1} and Q_{u2} would be equal. Nonetheless, only when the variance between the two values is more than 0.1% of Q_{u1} , the absorber plate temperature has been altered. In the MATLAB code, a conditional loop has been employed till the condition is satisfied. A uniform heat flux of 1000 W/m² is

commonly applied in SAH numerical studies to represent a realistic and controlled solar irradiation condition on the absorber plate. This value corresponds to typical peak solar insolation under clear-sky conditions, making it a practical reference for performance analysis. Using a constant heat flux also simplifies the boundary conditions, allowing direct comparison between different rib configurations by ensuring that any change in thermal performance is solely due to geometric modifications rather than variations in heat input [15,20]. The tabular presentation in Table 2 outlines the thermo-physical characteristics of the absorber plate material, namely, aluminium, and the surrounding air.

Table 2. Boundary condition and thermo-physical properties of absorber plate material and air.

Properties	Air	Aluminium
Density	1.176 kg/m ³	2719 kg/m ³
Specific heat	1006.4 J/(kg K)	871 J/(kg K)
Thermal conductivity	0.02624 W/(m K)	202.4 W/(m K)
Viscosity	1.857e-05 kg/(m s)	---

Thereafter, key performance parameters such as exergetic losses and overall exergy efficiency were evaluated for different pitch ratios. The analysis considered four major sources of exergy destruction: (i) losses associated with solar irradiation absorption at the absorber plate, (ii) losses due to combined radiative and convective heat transfer from the absorber plate to the ambient environment, (iii) losses occurring during heat transfer from the absorber plate to the working fluid, and (iv) losses arising from fluid friction within the duct. This comprehensive evaluation enabled a clearer understanding of how pitch ratio influences the thermodynamic performance of the solar air heater.

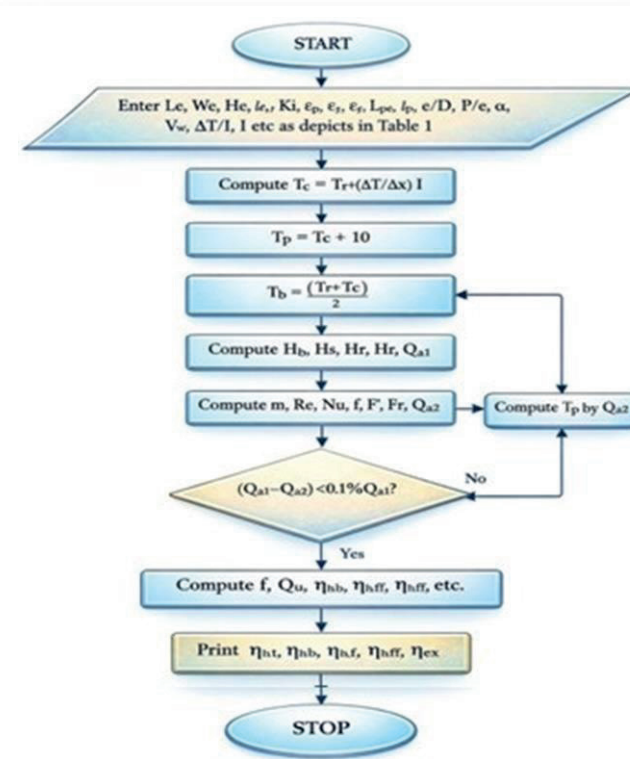


Fig. 2. Flowchart of the assumed mathematical model.

The flowchart in Fig. 2 depicts the step-by-step process followed in the current numerical work.

The expressions given in Eqs. (6)–(11) follow the approach presented by Bohra et al. [19]. Exergy losses by friction are determined by:

$$E_{L\Delta P} = \frac{m}{\rho} \Delta P \frac{T_a}{T_{fm}} \quad (6)$$

Exergy losses by heat transfer to the working fluid are calculated by:

$$E_{L\Delta T} = A_p \eta_{th} \left(\frac{T_a}{T_{fm}} - \frac{T_a}{T_{pm}} \right) \quad (7)$$

Exergy losses by both radiative and convective heat transfer from the absorber to the atmosphere are calculated by:

$$E_{Le} = U_L A_p (T_{pm} - T_a) \left(1 - \frac{T_a}{T_{pm}} \right) \quad (8)$$

Exergy losses by absorption of irradiation by the absorber are calculated as:

$$E_{LA} = I A_p (\tau \alpha) \eta_{exe} - \left(1 - \frac{T_a}{T_{pm}} \right) \quad (9)$$

The exergetic efficiency is calculated as:

$$\eta_{exe} = \frac{E_n}{E_s} \quad (10)$$

where E_n and E_s are the net exergy flow and exergy input, respectively. The effective efficiency (η_{eff}) is calculated as:

$$\eta_{eff} = \frac{Q_u - P_m/C}{I A_p} \quad (11)$$

3. Results and discussion

The addition of zig-zag structured baffles to a solar air heater can improve its thermal performance. However, this augmentation is always accompanied by increased friction losses. Due to that, the net energy gain will reduce, and it is always desirable to select the optimum design parameters. The influence of pitch ratio for Reynolds numbers ranging from 3000 to 14 000 has been studied. All configurations of pitch ratio were studied for a constant blockage height ratio of 0.3, insolation of 1100 W/m² and attack angle of 45°.

3.1. Exergy losses

Exergy losses due to friction are determined by the factor that produces increased friction losses. The lower the temperature rise parameter, the higher the mass flow rate, and thus the larger the friction losses. Figure 3 depicts the fluctuation of exergetic losses via friction with the temperature.

It was discovered that the most exergetic losses occur at lower pitch ratios. The value of exergetic losses via friction ($E_{L\Delta P}$) then increases as the temperature rise parameter decreases. It was observed that exergy losses due to friction are minimised after the temperature ascent criterion ($\Delta T/I$) of 0.004.

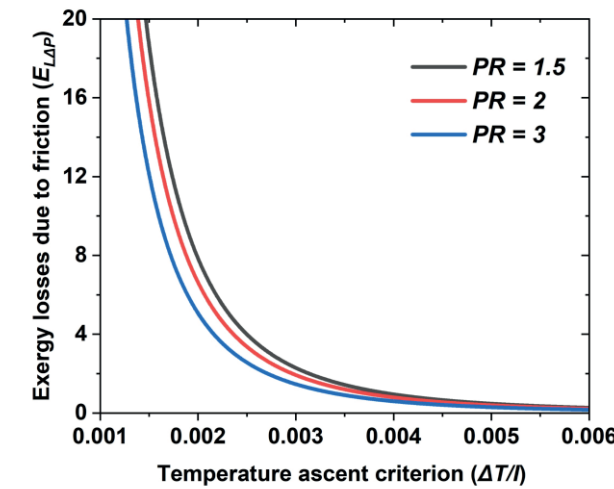


Fig. 3. Variation of exergy losses due to friction ($E_{L\Delta P}$) with temperature ascent criterion ($\Delta T/I$).

Exergy losses by absorption (E_{LA}) are due to irradiation absorbed by the plate of absorber at the mean temperature (T_{pm}), and as the temperature ($\Delta T/I$) increases, E_{LA} is found to decrease, as shown in Fig. 4. It was observed that the pitch ratio of 3 has shown the least value of E_{LA} . The irreversibility rises as the temperature rise parameter increases, resulting in higher exergy losses. The absorber plate's temperature rises in lockstep with the temperature rise parameter. For all pitch ratios, E_{LA} rises with the temperature ascent criterion, but eventually begins to decline.

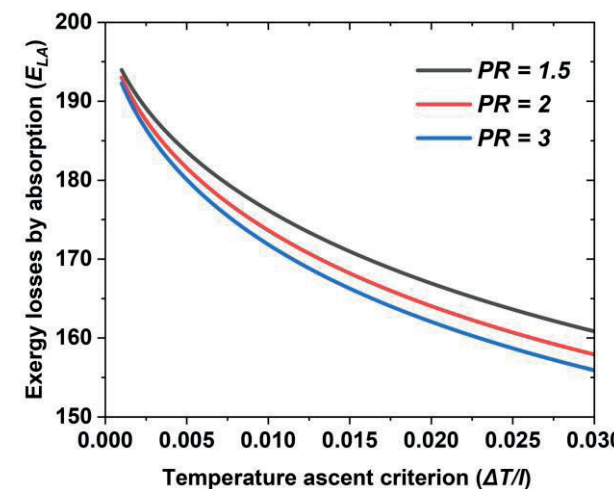


Fig. 4. Variation of exergy losses (E_{LA}) with temperature ascent criterion.

As the temperature rise parameter is increased, there is a corresponding increase in irreversibility, leading to higher exergy losses. The absorber plate temperature also elevates with the escalation of the temperature rise parameter. Consequently, the maximum exergetic losses occur when the temperature rise parameter has a higher value. The variation of exergy loss due to the working fluid with the temperature ascent criterion ($\Delta T/I$) shows a similar trend for all pitch ratios (PR) in Fig. 5. At low values of $\Delta T/I$, the exergy loss increases sharply, indicating

enhanced useful energy transfer to the fluid as the temperature difference grows. The curves reach a peak around $\Delta T/I \approx 0.01$ – 0.015 , where the exergy transfer is maximised. Beyond this point, a gradual decline is observed for all PR values, suggesting that a higher temperature ascent leads to increased irreversibilities and reduced effective exergy transfer. Among the cases, $PR = 1.5$ exhibits the highest peak exergy loss, followed by $PR = 3$ and $PR = 2$, while at larger $\Delta T/I$ values, $PR = 2$ maintains comparatively higher exergy loss than the others. Overall, the results indicate the existence of an optimum temperature ascent criterion for maximising exergy transfer to the working fluid, which slightly varies with the pitch ratio.

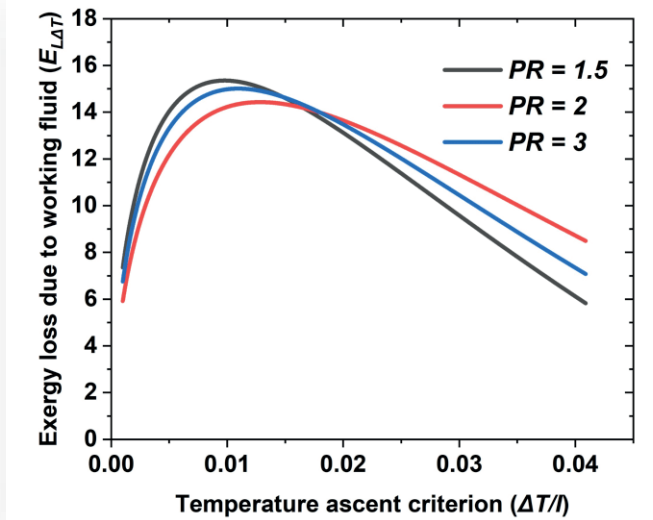


Fig. 5. Variation of exergy losses by working fluid ($E_{L\Delta T}$) with temperature ascent criterion.

Figure 6 shows the variation of exergy deficit to the surroundings (E_{Le}) with the temperature rise parameter. It has been acknowledged that with an increment in temperature rise parameter, exergy loss to the environment also rises because of the greater temperature differential between the absorber plate and the environment. Moreover, the pitch ratio of 3 has shown a maximum E_{Le} followed by the pitch ratios 2 and 1.5.

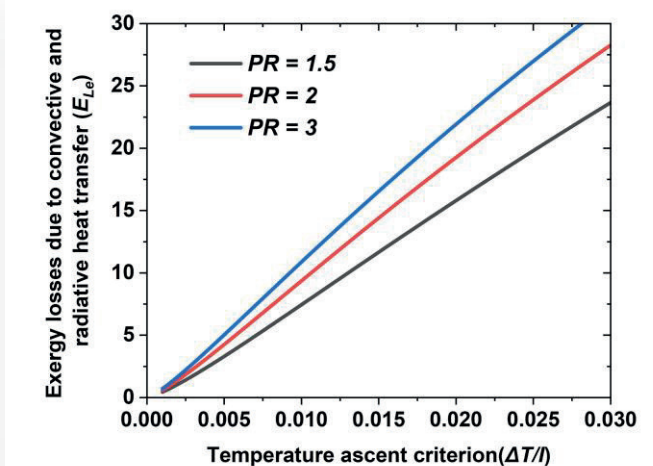


Fig. 6. Variation of exergy losses with temperature ascent criterion by convective and radiative heat transfer (E_{Le}).

3.2. Exergetic efficiency

Figure 7 shows how the exergetic efficiency (η_{exe}) varies with the temperature ascent criterion ($\Delta T/I$) for diverse pitch ratios. Due to the extremely high mass flow rate, a lower $\Delta T/I$ value concludes in a negative adverse value of exergetic efficiency. The exergetic efficiency improves as the value of $\Delta T/I$ increases. The pitch ratio 3 has the lowest exergetic efficiency, while the pitch ratio 1.5 has the highest exergetic efficiency. This is due to the supremacy of mechanical power, or the power necessary to overcome frictional forces.

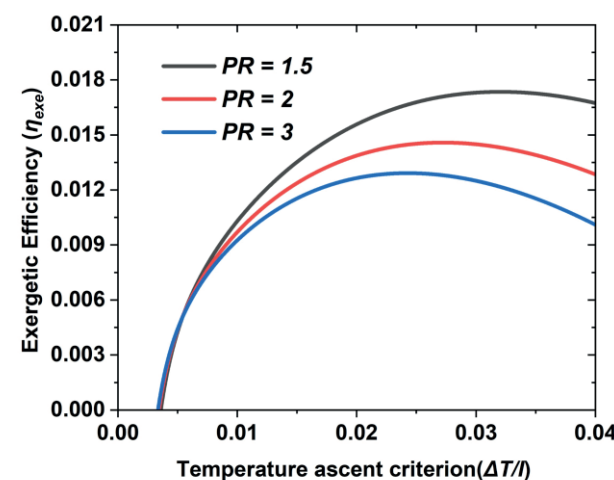


Fig. 7. Variation of exergetic efficiency (η_{exe}) with the temperature ascent criterion.

3.3. Effective efficiency

Figure 8 illustrates the variation of effective efficiency (η_{eff}) with Reynolds number for different pitch ratio (PR) configurations at a constant relative blockage ratio of 0.3, insolation of 1100 W/m^2 , and attack angle of 45° .

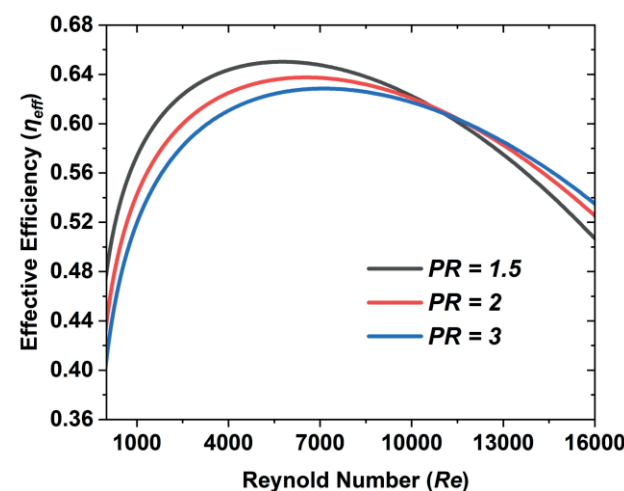


Fig. 8. Variation of η_{eff} with Re for various pitch proportions.

It is observed that the effective efficiency increases with the Reynolds number, reaches a peak, and then gradually declines

at higher flow rates. The initial rise in efficiency is due to improved convective heat transfer at higher Reynolds numbers, whereas the subsequent decline is caused by the increasing mechanical power required to overcome frictional losses.

At lower Reynolds numbers, the configuration with $PR = 1.5$ demonstrates the highest effective efficiency, attaining a maximum of about 64% at $Re \approx 5000$. This trend continues up to approximately $Re = 11\,000$, where $PR = 1.5$ maintains superior performance compared to other configurations. However, beyond this point, the efficiency for $PR = 1.5$ decreases more rapidly, and the configuration with $PR = 3$ exhibits comparatively better performance at higher Reynolds numbers. This indicates that lower pitch ratios are more effective at moderate flow conditions, while higher pitch ratios become advantageous at elevated Reynolds numbers due to reduced frictional penalties.

4. Conclusions

The numerical investigation demonstrates that the incorporation of zig-zag structured baffles significantly influences the exergetic and effective performance of the solar air heater over the Reynolds number range of 3000–14 000.

Exergy analysis reveals that frictional losses are the highest at lower pitch ratios, while exergy transfer to the working fluid reaches a maximum at a temperature ascent criterion of about $\Delta T/I \approx 0.01\text{--}0.015$.

Among the tested configurations, $PR = 1.5$ shows the highest exergetic efficiency, whereas $PR = 3$ results in the lowest efficiency due to reduced turbulence intensity.

In terms of effective efficiency, $PR = 1.5$ achieves the maximum value of approximately 64% at $Re \approx 5000$ and maintains superior performance up to about $Re \approx 11\,000$. Beyond this range, the efficiency of $PR = 1.5$ declines more rapidly, and $PR = 3$ performs better at higher Reynolds numbers due to comparatively lower frictional penalties.

Overall, the results indicate that a lower pitch ratio is optimal for moderate flow conditions, while a higher pitch ratio becomes advantageous at elevated Reynolds numbers.

References

- Kumar, A., Karn, A., McGregor, C., & Singh, V.P. (2025). A critical analysis of paradigm shifts from domestic solar cooking to institutional solar cooking technologies. *Renewable and Sustainable Energy Reviews*, 218, 115840. doi: 10.1016/j.rser.2025.115840
- Al-Yasiri, Q., Szabó, M., & Arıcı, M. (2022). A review on solar-powered cooling and air-conditioning systems for building applications. *Energy Reports*, 8, 2888–2907. doi: 10.1016/j.egy.2022.01.172
- Bisht, V.S., Patil, A.K., & Gupta, A. (2018). Review and performance evaluation of roughened solar air heaters. *Renewable and Sustainable Energy Reviews*, 81, 954–977. doi: 10.1016/j.rser.2017.08.036
- Saini, R.P., & Singal, S.K. (2007). A review on roughness geometry used in solar air heaters. *Solar Energy*, 81, 1340–1350. doi: 10.1016/j.solener.2007.01.017
- Chamoli, S., Thakur, N.S., & Saini, J.S. (2012). A review of turbulence promoters used in solar thermal systems. *Renewable and Sustainable Energy Reviews*, 16, 3154–3175. doi: 10.1016/j.rser.2012.01.021
- Radomski, P., Ziółkowski, P., de Sio, L., & Mikieliewicz, D. (2021). Computational fluid dynamics simulation of heat transfer from densely packed gold nanoparticles to isotropic media. *Archives of Thermodynamics*, 42(3), 87–113. doi: 10.24425/ather.2021.138111
- Ziółkowski, P., Radomski, P., Koulali, A., Kreft, D., Barański, J., & Mikieliewicz, D. (2025). Numerical study of heat transfer intensification in a chamber with heat generating by irradiated gold nanorods: One-way multiphysics and multiscale approach. *Energies*, 19(1), 181. doi: 10.3390/en19010181
- Bisht, V.S., Singh, D.B., Kumar, P., Jain, S., Bhandari, P., Mumtaz, M.A., & Rahman, M.H. (2026). Innovative spherical turbulator configurations for enhanced solar air heater performance: A numerical and analytical study. *Energy*, 342, 139601. doi: 10.1016/j.energy.2025.139601
- Lau, S., Kukreja, R., & McMillin, R. (1991). Effects of V-shaped rib arrays on turbulent heat transfer and friction of fully developed flow in a square channel. *International Journal of Heat and Mass Transfer*, 34(7), 1605–1616. doi: 10.1016/0017-9310(91)90140-A
- Singh, V., Jain, S., & Gupta, J. (2023). Analysis of the effect of perforation in multi-V rib artificial roughened single-pass solar air heater: Part A. *Experimental Heat Transfer*, 36(2), 163–182. doi: 10.1080/08916152.2021.1988761
- Singh, V., Jain, S., & Kumar, A. (2023). Establishment of correlations for thermo-hydraulic parameters due to perforation in a multi-V rib roughened single-pass solar air heater. *Experimental Heat Transfer*, 36(5), 597–616. doi: 10.1080/08916152.2022.2064940
- Lanjewar, A., Bhagoria, J.L., & Sarviya, R.M. (2011). Heat transfer and friction in solar air heater duct with W-shaped rib roughness on absorber plate. *Energy*, 36, 4531–4541. doi: 10.1016/j.energy.2011.03.054
- Ghildyal, A., Bisht, V.S., Rawat, K.S., & Bhandari, P. (2023). Effect of D-shaped, reverse D-shaped and U-shaped turbulators in solar air heater on thermo-hydraulic performance. *Archives of Thermodynamics*, 44(2), 3–20. doi: 10.24425/ather.2023.146556
- Ghildyal, A., Bisht, V.S., Bhandari, P., Thapliyal, S., Kaushik, S., Ranakoti, L., Bangari, R.S., Srivastav, A., Kanojia, N., & Paul, A.R. (2024). A comparative numerical evaluation of solar air heater performance having W-contoured, taper-contoured and reverse taper-contoured turbulators. *Archives of Thermodynamics*, 45(4), 189–196. doi: 10.24425/ather.2024.152008
- Chaudhari, M., Sharma, S.L., & Debbarma, A. (2023). Exergetic performance analysis of solar air heater with inverted L-shape ribs as roughness element. *Archives of Thermodynamics*, 44(3), 241–267. doi: 10.24425/ather.2023.147546
- Kumar, S., Bisht, V.S., Bhandari, P., Ranakoti, L., Negi, A., Bist, A.S., & Padalia, D. (2025). Computational analysis of modified solar air heater having combination of ribs and protrusion in S-shaped configuration. *International Journal on Interactive Design and Manufacturing*, 19(5), 3467–3478. doi: 10.1007/s12008-024-01972-2
- Alhilali, H.A.A., Al-Chlahawi, K., Zaroor, A.K., & Flayyih, M.A. (2025). Optimization of thermohydraulic parameters in solar air heater ducts featuring I-shaped ribs. *Heat Transfer*, 54, 3790–3814. doi: 10.1002/htj.23384
- Sriromreun, P., Thianpong, C., & Promvong, P. (2012). Experimental and numerical study on heat transfer enhancement in a channel with Z-shaped baffles. *International Communications in Heat and Mass Transfer*, 39(7), 945–952. doi: 10.1016/j.icheatmasstransfer.2012.05.016
- Bohra, J., Bisht, V.S., Bhandari, P., Rawat, K.S., Singh, J., Kumar, K., & Rawat, B. (2022). Effect of variable blockage height ratio on performance for solar air heater roughened with 45°-shaped baffles. *Materials Today: Proceedings*, 69(2), 153–157. doi: 10.1016/j.matpr.2022.08.279
- Singh, J., Bisht, V.S., Bhandari, P., Kumar, K., Singh, J., Alam, T., Dixit, S., Singh, S., & Khusnutdinov, R. (2023). Computational parametric investigation of solar air heater with dimple roughness in S-shaped pattern. *International Journal on Interactive Design and Manufacturing*, 18(5), 2969–2979. doi: 10.1007/s12008-023-01392-8



Co-published by
Institute of Fluid-Flow Machinery
Polish Academy of Sciences
Committee on Thermodynamics and Combustion
Polish Academy of Sciences

Copyright©2026 by the Authors under licence CC BY-NC-ND 4.0

<http://www.imp.gda.pl/archives-of-thermodynamics/>



Experimental Analysis of Energy Output in Evacuated Tube Heat Pipe Collector with Nanofluids

Shamkant Desale^{a*}, Premendra Bansod^a, Arakerimath Rachyya^b, Srinidhi Campli^c

^aMechanical Engineering Department, G H Rasoni College of Engineering & Management, Savitribai Phule University, Pune, Maharashtra 412207, India

^bMechanical Engineering Department, JSPM Rajshi Shahu College of Engineering, Pune, Maharashtra 411033, India

^cFaculty of Mechanical and Electrical Engineering, College of Military Engineering, Pune, Maharashtra 411031, India

*Corresponding author email: shamdesale27@gmail.com

Received: 10.06.2025; revised: 05.03.2026; accepted: 09.04.2026

Abstract

Solar heating devices have been extensively investigated to maximise energy utilisation and improve overall thermal efficiency. The present study aims to evaluate the performance of an evacuated tube heat pipe collector with nanofluid and compare it with a conventional heat pipe collector operating without nanofluid, with particular emphasis on the water outlet temperature. In the available literature, several studies have examined the effects of parameters such as the collector inclination angle, nanofluid concentration, and water inlet temperature on the thermal performance of evacuated tube solar collectors. However, a comprehensive comparative analysis incorporating all these parameters on a common experimental platform has not been adequately reported. In the present work, the variation in outlet temperature was systematically analysed by considering independent parameters such as the angle of inclination, copper oxide (CuO) nanofluid concentration, and water inlet temperature. Furthermore, the analysis of variance (ANOVA) was performed on the experimental dataset to evaluate the statistical significance and reliability of the obtained results.

Keywords: Evacuated pipe; ANOVA; Regression analysis; Heat pipe

Vol. 47(2026), No. 2, 159–165; doi: 10.24425/ather.2026.158682

Cite this manuscript as: Desale, S., Bansod, P., Rachyya, A., & Campli, S. (2026). Experimental Analysis of Energy Output in Evacuated Tube Heat Pipe Collector with Nanofluids. *Archives of Thermodynamics*, 47(2), 159–165.

1. Introduction

Solar energy is abundantly available in nature and serves as one of the most promising renewable energy sources. Efficient utilisation of solar energy at a low investment cost and with enhanced thermal efficiency has become increasingly important. Evacuated tube solar collectors are widely used to capture solar thermal energy for water heating applications. These systems can significantly improve thermal performance in industrial processes by conserving thermal energy, reducing operational time, and extending equipment lifespan. The thermal efficiency of solar energy systems has become a major focus in renewable energy research. The basic principle of solar heat collection in-

volves the absorption of solar radiation by a surface, resulting in an increase in temperature. Solar water heating systems are available in several configurations, among which heat pipe-based evacuated tube collectors have attracted considerable attention because of their superior heat transfer characteristics.

Heat pipes are extensively used in thermal management applications such as automobiles, electronic cooling systems, and laptop computers. With rapid advancements in technology, efficient thermal energy transfer has become essential in heat pipe technology. Nanomaterials are increasingly utilised as additives in working fluids such as water, engine oil and ethylene glycol to improve thermal performance. Nanoparticles such as silicon carbide and copper oxide possess superior thermal conductivity

Nomenclature

C_p – specific heat of water, J/(kg K)
 M_w – mass of water, kg
 Q – solar energy, J
 T – temperature of water, K

Subscripts and Superscripts

1, 2, 3, 4 – number of heat pipe

in – inlet, input
out – outlet
u – useful

Abbreviations and Acronyms

ANOVA – analysis of variance
SEM – scanning electron microscope
S/N – signal-to-noise
XRD – X-ray diffraction

compared to conventional base fluids, thereby enhancing heat transfer efficiency. A heat pipe operates on the principle of two-phase heat transfer, where the working fluid continuously undergoes evaporation and condensation within a closed system. When the evaporator section of the heat pipe is heated, the working fluid vaporises, increasing vapour pressure. The vapour then travels through the adiabatic section to the condenser region, where it releases latent heat and condenses back into liquid form. The condensed liquid subsequently returns to the evaporator section through gravity or capillary action, completing the cycle. The use of nanofluids as working fluids significantly improves the thermal performance of heat pipes because of their enhanced thermal conductivity and heat transfer properties.

Today, solar energy systems are gaining considerable importance because of their availability, economic viability, efficiency, and environmental sustainability in reducing carbon emissions and harmful gases. Elbrashy et al. [1] investigated the thermal performance of a solar air heater integrated with evacuated tubes and nano-enhanced copper oxide phase change material as a thermal storage enhancer. The nano-enhanced material was placed between the inlet and outlet flow passages to improve heat transfer rates, resulting in a thermal efficiency of 62.66%. Yurddas et al.[2] studied and optimised the thermal performance of evacuated tube solar collectors using various nanofluids. Their study concluded that nanofluids improve heat transfer performance, with copper oxide-water nanofluid exhibiting superior thermal characteristics. Gupta et al. [3] conducted research on the thermal performance optimisation of heat pipes using nanofluids and reported a collector efficiency of 66.40% at a nanofluid concentration of 1.1% and at an inclination angle of 58.5°C. Alqdah et al. [4] investigated the design, fabrication and performance evaluation of a shell-and-tube heat exchanger integrated with an evacuated tube solar collector. Their findings indicated that the system was reliable and effective for evaluating thermal performance under varying operating conditions such as solar radiation intensity, ambient temperature and inclination angle. Shafieian et al. [5] studied the thermal performance of evacuated tube heat pipe solar water heating systems during cold seasons. They concluded that additional thermal enhancement techniques are required during morning hours to improve solar energy absorption. Eidan et al. [6] experimentally investigated the performance of heat pipe evacuated tube collectors using Al_2O_3 and CuO nanofluids. Their results demonstrated that CuO nanofluid significantly improved thermal efficiency and accelerated temperature rise. Kim et al. [7] reviewed the thermal performance and efficiency of evacuated tube solar collectors using various nanofluids and conducted CFD analysis

of heat pipe solar water heaters. Their study confirmed that nanofluids considerably enhance thermal performance. Putra et al. [8] analysed the thermal performance of evacuated tube heat pipe solar collectors and concluded that both vacuum conditions and nanofluid properties are critical factors affecting collector efficiency. They also reported that thermosyphon evacuated tube collectors perform better than conventional heat pipe collectors. Joo et al. [9] experimentally analysed the effect of different working fluids, including water, ethanol, Flutec and methyl acetate, on evacuated tube solar collector performance. Their results indicated that methyl acetate provided the best performance at an inclination angle of 40°. Lopez et al. [10] investigated the thermo-hydraulic performance and entropy generation characteristics of evacuated tube solar collectors using pure water and nanofluids as working fluids. Mahendran et al. [11] studied the thermal efficiency enhancement of direct-flow evacuated tube solar collectors using titanium oxide nanofluids. Khalipe et al. [12] experimentally evaluated two-phase closed thermosyphon evacuated tube solar collectors with different nanofluids. Sakhrieh et al. [13] investigated the performance of five types of solar collectors and reported that evacuated tubes with black coatings exhibited the highest efficiency. Chaudhari et al. [14] experimentally analysed a heat pipe heat exchanger using zinc oxide nanofluid dispersed in water and observed that the heat transfer coefficient increased with temperature. Gupta et al. [15] studied evacuated tube collectors at different mass flow rates and concluded that the thermal system size could be reduced by nearly 30%. Sarr et al. [16] determined the optimum tilt angle for photovoltaic modules in Senegal and recommended monthly adjustment of the tilt angle for improved solar energy conversion. Mahendran et al. [17] investigated the variation of global solar radiation in East Coast Malaysia. Chakraborty et al. [18] reviewed the stability characteristics of nanofluids under varying temperature, pressure, magnetic field, and heat transfer conditions. Alabayechi et al. [19] performed numerical and thermal simulations using STAR-CCM+ software and reported that aluminium oxide nanofluids increased water temperature. Aggarwal et al. [20] reviewed energy storage media and hybrid enhancement techniques for improving evacuated tube collector performance. Yazdi et al. [21] studied nanofluid flow inside evacuated tube solar collectors and concluded that the collector efficiency increases with increasing nanofluid volume fraction.

Tang et al. [22] investigated the effect of collector tilt angle and observed similar efficiencies at 22° and 46° inclination angles. Sundar et al. [23] experimentally studied Al_2O_3 nanofluids in ethylene glycol-water mixtures and reported that the thermal

conductivity increased with nano-particle concentration but decreased with increasing temperature. Morrison et al. [24] developed a numerical model for heat transfer and fluid flow inside single-ended evacuated tubes and concluded that separate correlations are necessary for accurate flow rate prediction. Sukarno et al. [25] discussed challenges associated with nanofluid applications, including thermo-physical property determination, heat transfer behaviour and stability issues. Wole-Osho et al. [26] reviewed the limitations of nanofluids in solar thermal collectors and highlighted the need for further research on stability, hysteresis and long-term predictability. Ersöz et al. [27] investigated the influence of different working fluids on the energy and exergy performance of evacuated tube solar collectors with thermo-syphon heat pipes. Among the tested fluids – hexane, petroleum ether, chloroform, acetone, methanol and ethanol – acetone and chloroform demonstrated the highest exergy efficiency.

Although significant progress has been made in heat pipe solar collector technology, research on evacuated tube collectors using nanofluids as working fluids remains limited. Experimental and numerical investigations involving different working fluids are still required to determine optimal operating conditions. Most previous studies focused primarily on solar radiation effects; however, the integration of heat pipes and nanofluids can substantially enhance thermal energy absorption.

The vapour generated inside the heat pipe moves toward the condenser section, where heat is transferred to the manifold before the condensed liquid returns to the evaporator section, completing the continuous cycle. The performance of heat pipes is strongly influenced by parameters such as the inclination angle, solar radiation intensity and nanofluid concentration. Existing results indicate that copper oxide nanofluid is an effective working fluid for heat pipe applications. The present study utilises four evacuated tubes integrated with heat pipes, and it is anticipated that increasing the number of tubes and heat pipes will further improve the outlet water temperature and overall system efficiency.

2. Materials and methods

Nanoparticles consisting of nanometre-sized additives exhibit thermal properties, including the thermal conductivity, thermal diffusivity, viscosity and convective heat transfer coefficient, which differ from those of base fluids like water. Nanofluids are two-phase systems with phase one (liquid phase) and phase two (solid phase). The step method is used to prepare nanofluids. In this method, the first step produces dry powder by a chemical or physical method. Due to the trouble in making stable nanofluids by the two-step method, many progressive techniques have been developed to produce nanofluids. Copper oxide nanoparticles are manufactured at Adnano Technologies Pvt. Ltd., Shimoga, India, by the chemical precipitation method with multiple quality checks to obtain pure copper oxide nano-powder dispersed in water, as shown in Table 1.

Copper oxide nano-powder has an increased toughness and ductility, and an increased hardness. SEM and XRD techniques

were used for analysis, as shown in Figs. 1 and 2. A graph of particle size distribution is presented in Fig. 2. A particle distribution indicates the percentage of particles of different size.

Table 1. Properties of nanofluid used in research work.

CuO	Description
Weight per cent	1%, 3%, 5% (wt.)
Particle size	30–70 nm
Physical form	liquid
Morphology of dispersed particle	Near spherical
Colour purity	Brownish black 99.5%

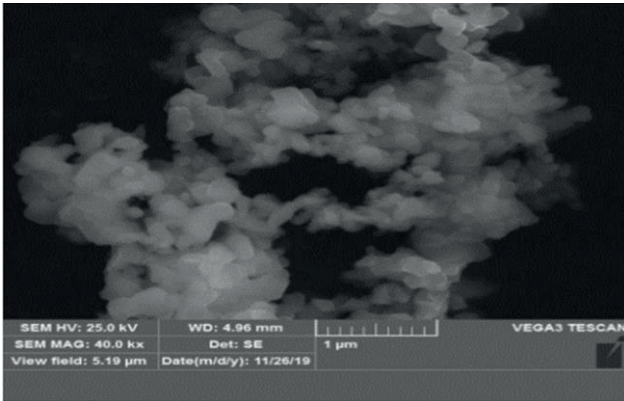


Fig. 1. SEM image of nanofluid.

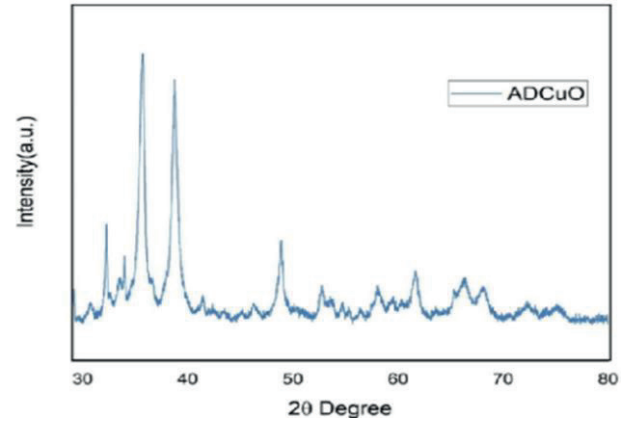


Fig. 2. XRD image of nanofluid.

The nanofluid was prepared by dispersing copper oxide with distilled water as a base fluid and filled in a heat pipe by weight per cent. The material used to prepare a heat pipe is copper. One end of the tube is well closed while the tube is vacuumed and charged with nanofluid from the other end, using a closing and opening mechanism, as shown in Fig. 3.

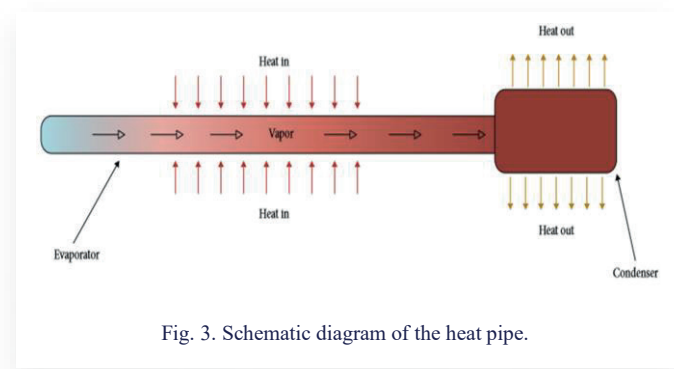


Fig. 3. Schematic diagram of the heat pipe.

During the experimentation, each of the copper tubes and evacuated tubes will be arranged in a tilted position and tested in outdoor conditions, but we are waiting for full sunshine hours for solar radiation. Monthly solar radiation at the location of Pune, India, for experimentation (longitude 74.15°, latitude 18.6°) is shown in Table 2. The water is taken as the working fluid and recirculated with the help of a pump. The heat pipe consists of three parts, which are the evaporator, the adiabatic part, and the condenser, as shown in Fig. 3. The working fluid inside the heat pipe evaporates as a result of transferring thermal energy to its surface, and the vapour moves upwards towards the condenser part, as shown in Fig. 4. The temperature was measured using K-type thermocouples positioned as shown in Fig. 5. Four temperature sensors were used to measure the temperature at the heat pipe condenser section, while one sensor was placed to measure the inlet temperature of water, and another was placed to measure the outlet temperature of water.

Table 2. Monthly solar radiation.

Month	Sunshine hours	Day length, hours
Jan	8.7	11.15
Feb	8.9	11.53
March	9.0	12.02
April	9.7	12.55
May	9.6	12.99
Oct	7.8	11.72
Nov	8.3	11.27
Dec	8.4	11.03

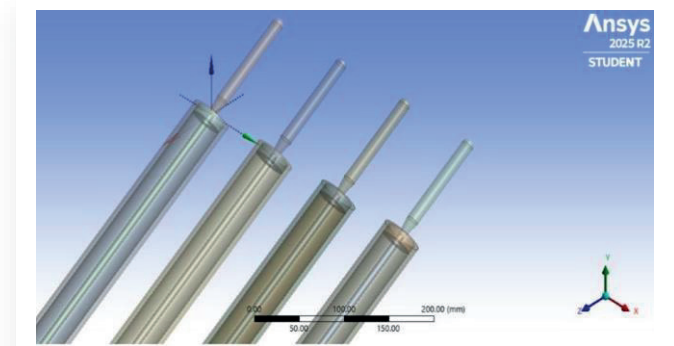


Fig. 4. Assembly of heat pipes with evacuated tubes.

Four evacuated tubes are installed as shown in Fig. 5. Heat pipes are filled with copper oxide nanofluid dispersed in deionised (DI) water to absorb solar energy. Solar energy is transferred to the recirculating water in the header of the condenser section of the heat pipe, and the recirculating water absorbs thermal energy from the condenser section of the heat pipe, completing the cycle.



Fig. 5. Experimental setup of evacuated tube solar collector.

Throughout the test, the ambient temperature, inlet water temperature, outlet water temperature and heat pipe condenser temperature were recorded. The mass flow rate at the inlet of the header is controlled by a rotameter. The temperature of heat pipes and recirculating water was measured by thermocouples placed as shown in the figure. The solar system consists of evacuated tubes, copper pipes, a tank with a circulation pipe, and a suitable frame for mounting. There are four copper tubes installed with some inclination, as shown in Fig. 5. These copper tubes are surrounded by borosilicate glass tubes. The inlet pipe is connected to the header, and tubes are press-fitted in the header. Another thermocouple was placed inside the storage tank. Water was circulated through the header. During the experimentation, each of the copper tubes and evacuated tubes will be arranged in a tilted position and tested in outdoor conditions.

The rate of useful solar energy (Q_u) is calculated by:

$$Q_u = M_w C_p (T_{out} - T_{in}), \quad (1)$$

where M_w is the mass of water and C_p is the specific heat of water, T_{out} is the outlet temperature of water, and T_{in} is the inlet temperature of water.

The incident solar radiation on the evacuated tube collector, which can be considered as input energy (Q_{in}), so the thermal efficiency of the system can be found as:

$$\eta = Q_u / Q_{in}. \quad (2)$$

3. Results and discussion

During the experimentation, each of the copper tubes and evacuated tubes was arranged in a tilted position and tested in outdoor conditions. The temperatures in the heat pipe condenser are gathered in Table 3, where T_1 is the temperature at heat pipe 1, T_2 is the temperature at heat pipe 2, T_3 is the temperature at heat pipe 3, and T_4 is the temperature at heat pipe 4.

Table 3. Temperatures at the heat pipe condenser.

T_1 (°C)	T_2 (°C)	T_3 (°C)	T_4 (°C)
105	104	102	97
105	105	96	97
106	104	102	97
107	106	105	98
104	110	90	92
103	97	87	90

The graphs of inlet temperature, inclination angle of 40° to the ground level and outlet temperature captured throughout the day at full sunshine hours for different nanofluid concentrations are plotted in Figs. 6–8.

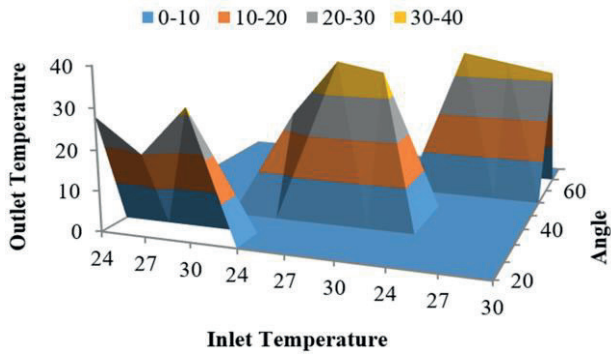


Fig. 6. Variation of temperature with tilt angle for 1% copper oxide nanofluid used in the heat pipe.

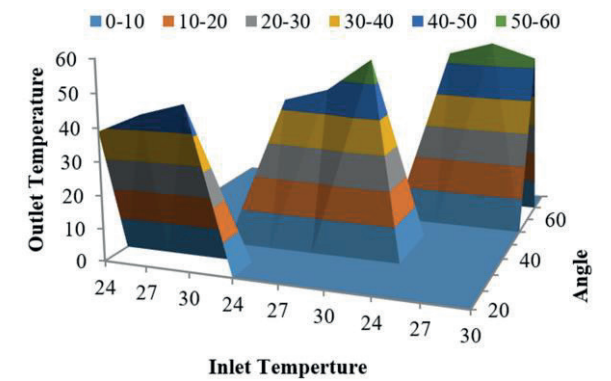


Fig. 7. Variation of temperature with tilt angle for 3% copper oxide nanofluid used in the heat pipe.

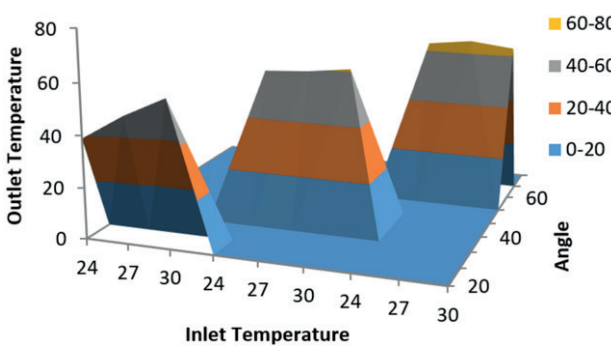


Fig. 8. Variation of temperature with tilt angle for 5% copper oxide nanofluid used in the heat pipe.

Based on the literature, the authors have decided to determine the variation of angle, copper oxide percentage, and inlet temperature in three levels taken for testing. Taguchi's robust design method was used with three parameters and three variable levels to cover a wide range. The control factors and levels (V1–V3 and L1–L3) are listed in Table 4.

A total of 27 experiments were conducted with three levels and three factors. For the experimental data set, ANOVA analysis was carried out for the prediction of the quality of the data.

Table 4. Control factors and levels.

Variable	Parameter	L1	L2	L3
V1	Angle (°)	20	40	60
V2	CuO (%)	1	3	5
V3	Inlet temperature (°C)	24	27	30

Table 5 shows the details for the analysis of variance (DF, SS, MS, F and P) for the outlet temperature. From the above table, it has been observed that p measure values for all the input parameters, such as the angle of tilt in degrees, copper oxide percentage, and input temperature, are less than 0.05. Hence, all these input parameters are identified as significant factors.

Table 5. Analysis of variance for outlet temperature.

Source	DF	SS	MS	F	P
Angle (°)	2	758.0	379.000	46.28	0.000
CuO (%)	2	816.7	408.333	49.86	0.000
Inlet temperature (°C)	2	356.2	178.111	21.75	0.000
Error	20	163.8	8.189		
Total	26	2094.7			

In the present experimentation, the optimum value of the angle in degrees is 40°, the optimum value of the copper percentage is 5 percent, and the input temperature is 30°C, as shown in Table 6.

Table 6. Optimum parameters.

Factors	Optimum values
Angle (°)	40
CuO (%)	5
Inlet temperature (°C)	30

Previous studies regarding heat pipe solar water heating systems have focused on solar radiation only, but by using a heat pipe and nanofluid in the solar collector, more energy is absorbed. The vapours inside the heat pipe rise towards the condenser section, where they transfer their heat through the manifold and return to the evaporator section, and the cycle continues. Heat pipe characteristics are dependent on the angle of inclination, solar radiation, and the percentage of nanofluid used. The above results show that copper oxide nanofluid is an effective working fluid for the heat pipe.

The present work is carried out with four evacuated tubes and heat pipes. If more tubes and a heat pipe are used, the outlet

temperature will further increase, thereby resulting in the economics of material/price to heat capture. From the above result, the maximum outlet temperature is found to be about 68°C as shown in Table 7.

Table 7. Comparison of S/N ratio.		
	Output temperature (°C)	S/N ratio
Predicted	66.6	36.70
Confirmed	68.0	36.65

The developed experimental setup in this work well predicts the increase in outlet temperature. As the performance of the heat pipe depends on the temperature of the evaporator and condenser area, more experimental and numerical work is required to explore the interaction effect among all the parameters of the evacuated tube heat pipe by using a nanofluid. Among the challenges for future research work are nanofluid preparation, in particular concerning the particle size, nanofluid stability, thermal conductivity and morphology.

4. Conclusions

The overall performance of the evacuated-tube solar collector was calculated. The outlet temperature was observed to vary with input parameters like the angle of inclination, CuO nanofluid concentration (% vol), and water inlet temperature. The outlet temperature was found to depend on the incoming solar radiation and the number of tubes, and the intensity of solar heat absorption was increased with the use of CuO nanofluid. The ANOVA analysis was carried out for the prediction of the quality of the data. The p-measure values for all input parameters, such as tilt angle, copper oxide percentage, and input temperature, were found to be less than 0.05, indicating a good level of significance. The best results for the water outlet temperature of 68°C were obtained for a 40° angle of inclination, 5% copper oxide nanofluid dispersed in distilled water, and 30°C of water inlet temperature. An increased concentration of CuO can be considered, however the possibility of contamination and fouling effects needs to be explored.

References

[1] Elbrashy, A., Aboutaleb, F., El-Fakharany, M., & Essa, F.A. (2023). Experimental study of solar air heater performance with evacuated tubes connected in series and involving nano-copper oxide/paraffin wax as thermal storage enhancer. *Environmental Science and Pollution Research*, 30(2), 4603–4616. doi: 10.1007/s11356-022-22462-6

[2] Yurddaş, A. (2020). Optimization and thermal performance of evacuated tube solar collector with various nanofluids. *International Journal of Heat and Mass Transfer*, 152, 119496. doi: 10.1016/j.ijheatmasstransfer.2020.119496

[3] Gupta, N.K., Sharma, A., Rathore, P.K.S., & Verma, S.K. (2020). Thermal performance optimization of heat pipe using nanofluid: response surface methodology. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*. 42(11), 590. doi: 10.1007/s40430-020-02668-w

[4] Alqdah, K.S., Alfredi, A., Alnuman, N., Aljohani, M., Almutairi, M., & Alhazmi, A. (2020). Design, Fabrication and Testing of Shell and Tube Heat Exchanger Integrated with Vacuum Tubes Solar Water Heater. *Journal of Energy Research and Reviews*, 6(3), 1–16. doi: 10.9734/jenrr/2020/v6i330167

[5] Shafieian, A., Khiadani, M., & Nosrati, A. (2019). Thermal performance of an evacuated tube heat pipe solar water heating system in cold season. *Applied Thermal Engineering*, 149, 644–657. doi: 10.1016/j.applthermaleng.2018.12.078

[6] Eidan, A.A., AlSahlani, A., Ahmed, A.Q., Al-fahham, M., & Jalil, J.M. (2018). Improving the performance of heat pipe-evacuated tube solar collector experimentally by using Al₂O₃ and CuO/acetone nanofluids. *Solar Energy*, 173, 780–788. doi: 10.1016/j.solener.2018.08.013

[7] Kim, H., Kim, J., & Cho, H. (2017). Review of thermal performance and efficiency in evacuated tube solar collector with various nanofluids. *International Journal of Air-Conditioning and Refrigeration*, 25(02), 1730001. doi: 10.1142/S2010132517300014

[8] Putra, N., Kristian, M.R., David, R., Haliansyah, K., & Ariantara, B. (2016). Thermal performance of evacuated tube heat pipe solar collector. In *AIP Conference Proceedings*, 1737(1), 050004. AIP Publishing LLC. doi: 10.1063/1.4949307

[9] Joo, H.J., & Kwak, H.Y. (2017). Experimental analysis of thermal performance according to heat pipe working fluids for evacuated tube solar collector. *Heat and Mass Transfer*, 53(11), 3267–3275. doi: 10.1007/s00231-017-2029-0

[10] López-Núñez, O.A., Lara, F., Gonzalez-Angeles, A., Cárdenas-Robles, A., Ramírez-Minguela, J.J., & Alfaro-Ayala, J.A. (2024). Assessment of thermohydraulic performance and entropy generation in an evacuated tube solar collector employing pure water and nanofluids as working fluids. *Heliyon*, 10(8), e29309. doi: 10.1016/j.heliyon.2024.e29309

[11] Mahendran, M., Ali, T.Z.S., Shahrani, A., & Bakar, R.A. (2014). The efficiency enhancement on the direct flow evacuated tube solar collector using water-based titanium oxide nanofluids. *Applied Mechanics and Materials*, 465, 308–315. doi: 10.4028/www.scientific.net/AMM.465-466.308

[12] Khalipe, V., & Deshmukh, P. (2015). Experimental study of evacuated tube two phase closed thermosyphon (TPCT) solar collector with nanofluid. *Journal of Mechanical and Civil Engineering (IOSRJMCE)*, 3(21), 156–161.

[13] Sakhrich, A., & Al-Ghandoor, A. (2013). Experimental investigation of the performance of five types of solar collectors. *Energy Conversion and Management*, 65, 715–720. doi: 10.1016/j.enconman.2011.12.038

[14] Chaudhari, V., Dharmadhikari, M., & Kolhe, V. (2023). Performance Evaluation of Heat Pipe Heat Exchanger with Nanofluid: An Experimental Study. *Letters in Applied NanoBioScience*, 12(4), 121. doi: 10.33263/LIANBS124.121

[15] Gupta, M., Prasad, R.B., & Singh, A. (2023). Thermal Performance of Evacuated Tube Solar Collector using Water and CuO/Water Nanofluid. *AIJR Proceedings*, pp. 120–127. doi: 10.21467/proceedings.161.13

[16] Sarr, A., Kebe, C.M., & Ndiaye, A. (2021). Determination of the optimum tilt angle for photovoltaic modules in Senegal. *African Journal of Environmental Science and Technology*, 15(6), 214–222. doi: 10.5897/AJEST2021.2988

[17] Mahendran, M., Lee, G.C, Shahrani, A., Bakar, R.A., Kadirgama, K., Amir, A.R., & Sharma, K.V. (2013). Diurnal pattern and estimation of global solar radiation in East Coast Malaysia. *International Journal of Automotive and Mechanical Engineering*, 8, 1162–1175. doi: 10.15282/ijame.8.2013.7.0095

[18] Chakraborty, S., & Panigrahi, P.K. (2020). Stability of nanofluid: A review. *Applied Thermal Engineering*, 174, 115259. doi: 10.1016/j.applthermaleng.2020.115259

[19] Al-Abayechi, Y., Alaiwi, Y., & Al-Khafaji, Z. (2024). Exploration of Key Approaches to Enhance Evacuated Tube Solar Collector Efficiency. *Journal of Advanced Research in Numerical Heat Transfer*, 19(1), 1–14. doi: 10.37934/amht.19.1.114

[20] Aggarwal, S., Kumar, R., Lee, D., Kumar, S., & Singh, T. (2023). A comprehensive review of techniques for increasing the efficiency of evacuated tube solar collectors. *Heliyon*, 9(4), e15185. doi: 10.1016/j.heliyon.2023.e15185

[21] Yazdi, M.H., Solomin, E., Fudholi, A., Divandari, G., & Sopian, K. (2021). Thermal Performance of Nanofluid Flow Inside Evacuated Tube Solar Collector. *International Journal of Heat & Technology*, 39(4), 1262–1270. doi: 10.18280/ijht.390424

[22] Tang, R., Yang, Y., & Gao, W. (2011). Comparative studies on thermal performance of water-in-glass evacuated tube solar water heaters with different collector tilt-angles. *Solar Energy*, 85(7), 1381–1389. doi: 10.1016/j.solener.2011.03.019

[23] Sundar, L.S., Ramana, E.V., Singh, M.K., & Sousa, A.C.M. (2014). Thermal conductivity and viscosity of stabilized ethylene

glycol and water mixture Al₂O₃ nanofluids for heat transfer applications: An experimental study. *International Communications in Heat and Mass Transfer*, 56, 86–95. doi: 10.1016/j.icheatmasstransfer.2014.06.009

[24] Morrison, G.L., Budihardjo, I., & Behnia, M. (2005). Measurement and simulation of flow rate in a water-in-glass evacuated tube solar water heater. *Solar Energy*, 78(2), 257–267. doi: 10.1016/j.solener.2004.09.005

[25] Sukarno, D.H. (2017). Challenges for nanofluid applications in heat transfer technology. In *Journal of Physics: Conference Series*, 795(1), 012020. IOP Publishing. doi: 10.1088/1742-6596/795/1/012020

[26] Wole-Osho, I., Okonkwo, E.C., Abbasoglu, S., & Kavaz, D. (2020). Nanofluids in solar thermal collectors: review and limitations. *International Journal of Thermophysics*, 41, 157. doi: 10.1007/s10765-020-02737-1

[27] Ersöz, M.A. (2016). Effects of different working fluid use on the energy and exergy performance for evacuated tube solar collector with thermosyphon heat pipe. *Renewable Energy*, 96, 244–256. doi: 10.1016/j.renene.2016.04.058



Co-published by
Institute of Fluid-Flow Machinery
Polish Academy of Sciences
Committee on Thermodynamics and Combustion
Polish Academy of Sciences

Copyright©2026 by the Authors under licence CC BY-NC-ND 4.0

<http://www.imp.gda.pl/archives-of-thermodynamics/>



Efficiency of Energy Conversion Cycles in Finite Time Thermodynamics: The Case of Nuclear-Driven Carnot Cycle

Patrick Dumaz

CEA/IRESNE, Nuclear Energy Division, CEA Cadarache, 13108 Saint-Paul-lez-Durance, France
Author email: patrick.dumaz@cea.fr

Received: 30.10.2025; revised: 01.04.2026; accepted: 12.05.2026

Abstract

This paper aims to clarify the conditions required to derive, within the framework of Finite Time Thermodynamics, the efficiency of “endoreversible cycles” (Eq. (1) of this paper). It is shown that the steady-state enthalpy and entropy balances of the working fluid at maximum power, the finite time thermodynamics optimum, lead to this efficiency. This result is obtained without introducing the “finite time” of the cycle, the most usual approach. To compute this efficiency for real systems, the heat source and sink temperatures must be clearly defined. In complex systems such as nuclear reactors, several choices of the upper temperature are possible. For a rational choice – using the steam generator as the heat source – these temperatures are relatively low, and the finite time thermodynamics optimum corresponds to unrealistic operating conditions, slightly underestimating actual efficiencies. Conversely, higher upper temperature values yield more realistic conditions but overestimate efficiency. The finite time thermodynamics limits are revealed by computing all thermodynamic cycle parameters and introducing irreversibilities in the working fluid. The cycle specific net work (MJ/kg) appears to be a more relevant optimisation criterion for real systems. This supports the idea that acceptable predictions from finite time thermodynamics efficiency result from compensating errors: low maximum temperatures balanced by the ideal cycle assumption. A general efficiency equation better illustrates why actual efficiency remains below Carnot’s, clarifying optimisation issues in energy conversion systems and the trade-off between reducing irreversibilities and maximising average heat source temperature.

Keywords: Carnot cycle; Finite time thermodynamics; Power output optimum; Efficiency optimum

Vol. 47(2026), No. 2, 167–178; doi: 10.24425/ather.2026.158683

Cite this manuscript as: Dumaz, P. (2026). Efficiency of Energy Conversion Cycles in Finite Time Thermodynamics: The Case of Nuclear-Driven Carnot Cycle. *Archives of Thermodynamics*, 47(2), 167–178.

1. Introduction

The origin of this work lies in recognition of extensive literature on "Finite Time Thermodynamics" (FTT) and "Endoreversibility", with the proposal of a straightforward equation, Eq. (1) hereafter, aimed to provide a more realistic estimation of the efficiency of energy conversion systems, particularly in comparison with the Carnot efficiency. This equation is:

$$\eta_t = 1 - \sqrt{\frac{T_d}{T_u}}, \quad (1)$$

with T_u being said to be the heat source temperature and T_d the heat sink temperature (both in Kelvin).

The main objective of this paper is to better understand the conditions and assumptions necessary to obtain this elegant equation.

Nomenclature

h	– specific enthalpy, J/kg
$\dot{h}_d$	– heat sink heat transfer coefficient, W/K
$\dot{h}_u$	– heat source heat transfer coefficient, W/K
$\dot{m}$	– cycle mass flowrate, kg/s
$\dot{P}$	– cycle power output, W
P_i	– pressures of the Carnot cycle, $i = 1$ to 4, Pa
$\dot{Q}_d$	– heat transfer with the surroundings for the heat sink, W
$\dot{Q}_u$	– heat transfer with the surroundings for the heat source, W
r	– specific ideal gas constant, J/(kg·K)
s	– specific entropy, J/(kg·K)
T_d	– heat sink temperature, K
T_i	– temperatures of the Carnot cycle, $i = 1$ to 4, K
T_u	– heat source temperature, K

$\dot{W}_{ij}$ – work exchanged with the surroundings in the (i, j) process, W

Greek symbols

γ	– ratio of specific heats at constant pressure and at constant volume
η	– thermodynamic cycle efficiency
η_t	– finite time thermodynamic optimum efficiency

Subscripts and Superscripts

d	– heat sink parameters
u	– heat source parameters

Abbreviations and Acronyms

FTT	– finite time thermodynamics
PWR	– pressurised water-cooled reactor

like thermoelectricity, Ouerdane et al. [6], or with the introduction of irreversibilities, Wu and Kiang [7] or Feidt and Costea [8]. Le et al. [9] or Kaushik et al. [12] have shown that Eq. (1) could be useful for other cycles than the Carnot cycle originally considered. In this paper, only Carnot-like cycles will be examined.

It is worth noting from this abundant scientific literature that many of these studies share a common limitation: the absence of a clear definition of the thermodynamic system under consideration, which could explain difficulties encountered in properly defining T_u .

Our purpose is to revisit these analyses, to better understand the real interest of Eq. (1) and not at all to propose new optimisations of real system energy conversion cycles. Thus, in Section 2.1, we present a derivation of this equation without introducing a cycle time. In the following Section 2.2, some Eq. (1) numerical results will be compared to real cycle efficiencies.

In Section 2.3, we will examine further all the implications of the maximum power output by determining all the cycle variables in very simple cases (use of an ideal gas as the cycle working fluid).

In section 3, we will see that for a “Carnot cycle”, with internal irreversibilities, here non-isentropic compression and expansion (of course with irreversibilities, strictly speaking, it is no longer a Carnot cycle), we obtain a different result in terms of compromise between the maximum efficiency and maximum power output of the cycle.

Finally in Section 4, we will propose an “academic extension” of the Carnot efficiency to real cycles. The equation obtained is not of interest to calculate the efficiency of real systems, but it reveals the underlying causes, in fact well known, to be well below the Carnot efficiency.

It is important to note that this work is essentially academic in nature as it involves discussing a theory, the FTT, that is itself also essentially academic. The ambition of this paper is also pedagogical: to better understand the FTT theoretical background.

2. “Endoreversible” Carnot cycle calculation

Our objective is to revisit the derivation of Eq. (1) by a thermodynamic approach based on well-defined thermodynamic systems and without the introduction of a cycle “finite time” (for

example, cycle time introduced in [4]). Being in steady-state, we thought there is no real need to introduce such a time.

The cycle calculated is a Carnot cycle (Fig. 1) with two heat exchangers for the heat source at T_u and the heat sink at T_d .

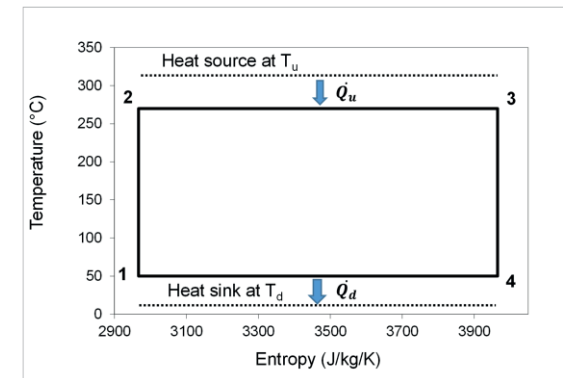


Fig. 1. The FTT Carnot cycle with its two heat exchangers.

The thermodynamic system considered is the working fluid having a mass flow rate $\dot{m}$ and describing the cycle formed by the four successive states 1 to 4 (Fig. 1), so it does not include the “materials” constituting the two heat exchangers. The cycle is characterised by four enthalpies (h_1 to h_4) and four entropies (s_1 to s_4).

As it is a Carnot cycle, we have:

- $s_2 = s_1$ and $s_4 = s_3$ (isentropic process), and
- $T_3 = T_2$ and $T_1 = T_4$ (isothermal heat addition and isothermal heat rejection).

To introduce the two heat exchangers, two heat transfer coefficients, $\dot{h}_u$ and $\dot{h}_d$, must be introduced to calculate the heat transfers, so the following simple equations are used:

– for the heat source

$$\dot{Q}_u = \dot{h}_u(T_u - T_2),$$

– for the heat sink

$$\dot{Q}_d = \dot{h}_d(T_d - T_1).$$

These heat transfer coefficients are assumed constant in the FTT approach, and in fact, they are the product of the actual heat transfer coefficient (in W/(m² K)) by the heat exchanger surface.

As a consequence, T_1 is the real low temperature seen by the working fluid of the cycle, and T_2 is the real high temperature of the cycle. To achieve such constant temperatures, with heat addition (or heat rejection), is only possible during a phase transition of the working fluid or by combining heat addition (or rejection) with the expansion (or compression) of a gas as the working fluid. For this simple thermodynamic cycle, the efficiency is, of course, given by the Carnot equation:

$$\eta_c = 1 - \frac{T_1}{T_2}.$$

The FTT purpose is to go further by expressing the efficiency with T_u and T_d , thus two temperatures not directly experienced by the thermodynamic system, but which may be of interest for

the process evaluation. Usually, a thermodynamic analysis must use system temperatures, for example, the “exchanged entropy” $\delta Q/T$ must be calculated with “the temperature of the part of the system receiving it” (the Gibbs formulation). Therefore, in principle, using T_u and T_d would imply that the two heat exchangers (i.e., the materials responsible for heat conduction) are included in the thermodynamic system. Such a system will be considered in Section 4.

T_u and T_d are also assumed constant; it is often said that the two heat reservoirs are of infinite thermal capacity.

For a nuclear reactor like a PWR (pressurized water-cooled reactor), the very simple equation giving $\dot{Q}_u$ raises some questions, see Section 2.2. A rational assumption seems to take T_u as the primary coolant temperature, with several dozens of tons per second of water flow in the primary circuit, almost constant water temperatures in this circuit are obtained. In other real systems, such as coal power plants, much larger variations in the heat source primary side temperature are observed, several hundred degrees of Celsius, making the use of this hypothesis even more questionable.

2.1. Efficiency at the maximum power output

With the mass flow rate $\dot{m}$ of the working fluid used in this cycle, the system of ten equations to solve is made of the four energy balances:

$$\dot{m}(h_2 - h_1) = \dot{W}_{12},$$

$$\dot{m}(h_3 - h_2) = \dot{W}_{23} + \dot{h}_u(T_u - T_2),$$

$$\dot{m}(h_4 - h_3) = \dot{W}_{34},$$

$$\dot{m}(h_1 - h_4) = \dot{W}_{41} + \dot{h}_d(T_d - T_1),$$

$\dot{W}_{ij}$ being the work exchanged with the surroundings by the $(i-j)$ process of the cycle.

In addition, we have the four entropy balances:

$$s_2 = s_1,$$

$$\dot{m}(s_3 - s_2) = \frac{\dot{h}_u(T_u - T_2)}{T_2},$$

$$s_4 = s_3,$$

$$\dot{m}(s_1 - s_4) = \frac{\dot{h}_d(T_d - T_1)}{T_1},$$

and finally:

$$T_3 = T_2,$$

$$T_4 = T_1.$$

For a working fluid with $h = h(P, T)$ and $s = s(P, T)$, and having chosen T_u , T_d , $\dot{h}_u$ and $\dot{h}_d$, we have these 10 equations to determine 13 unknowns:

- the four pressures and the four temperatures,
- the four works exchanged with the surroundings, $\dot{W}_{ij}$,
- and the cycle mass flowrate.

It is worth recalling that we are in steady-state, so all these unknowns are constant.

In the FTT approach, $T_u - T_2$ will be varied in order to find the maximum cycle power output. We will see just after that it is enough to fix $T_u - T_2$ to calculate the cycle efficiency and the cycle power output (specificity of the Carnot cycle). But to determine the 13 unknowns, two additional cycle parameters have to be fixed; we will see that there is an interest to determine all the cycle parameters, see Section 2.3.

The cycle power output $\dot{P}$ (a positive value is considered here) is given by the cycle energy balance (by summing up the 4 enthalpy equations):

$$\dot{P} = |\dot{W}_{12} + \dot{W}_{23} + \dot{W}_{34} + \dot{W}_{41}| = \dot{Q}_u + \dot{Q}_d,$$

with $x = T_u - T_2$ and $y = T_d - T_1$.

We have:

$$\dot{P} = \dot{h}_u x + \dot{h}_d y.$$

And the cycle entropy balance (by summing up the 4 entropy equations) gives:

$$\frac{\dot{h}_u(T_u - T_2)}{T_2} + \frac{\dot{h}_d(T_d - T_1)}{T_1} = \frac{\dot{h}_u x}{T_u - x} + \frac{\dot{h}_d y}{T_d - y} = 0.$$

It could be noted that these last two equations represent the global cycle energy and entropy balance. Therefore, it could be argued that the initial system of 13 equations was indeed not useful. In fact, this system is necessary for the complete determination of the cycle variables, see Section 2.3.

One can calculate:

$$y = \frac{T_d \dot{h}_u x}{x(\dot{h}_u + \dot{h}_d) - T_u \dot{h}_d}.$$

It is now possible to calculate the power output and the cycle efficiency as a function of x only:

$$\dot{P} = \dot{h}_u x \left(1 - \frac{T_d}{T_u - x(\dot{h}_u + \dot{h}_d)/\dot{h}_d} \right), \quad (2)$$

$$\eta = \frac{\dot{P}}{\dot{h}_u x} = 1 - \frac{T_d}{T_u - x(\dot{h}_u + \dot{h}_d)/\dot{h}_d}. \quad (3)$$

This Eq. (3) is a continuously decreasing hyperbola as a function of x . For non-negative values of x , the maximum efficiency is obtained for $x = 0$, this is the Carnot efficiency of a cycle producing no work, no heat being transferred to the cycle (because $x = 0$). We determine x_{max} , giving the maximum power, so the FTT optimum is found by solving $\frac{d\dot{P}}{dx} = 0$:

$$x_{max} = \frac{T_u \dot{h}_d}{\dot{h}_u + \dot{h}_d} \left(1 - \sqrt{\frac{T_d}{T_u}} \right).$$

And finally,

$$\eta(x_{max}) = 1 - \sqrt{\frac{T_d}{T_u}} = \eta_t.$$

The attractiveness of this equation lies in a very simple expression depending only on T_u and T_d and not on $\dot{h}_u$ and $\dot{h}_d$, making it independent of the two heat exchanger characteristics. It is worth noting that in the previous calculations, no assumptions have been made regarding the Carnot cycle's working fluid.

At this stage of our demonstration, one can already observe that if $\dot{h}_u$ and $\dot{h}_d$ are of the same order of magnitude, an apparently reasonable assumption for nuclear PWR but which could be discussed (see Section 2.2), and for typical T_u and T_d PWR temperatures, roughly 600 K and 300 K, one obtains x_{max} of about 90 K and y_{max} of about 60 K. These values are quite high compared to those of PWR systems, which usually operate with lower temperature differences, about 30 K for x . It is worth recalling that our choice of T_u means that we have considered the PWR steam generator as the cycle heat source heat exchanger, thus T_u is the primary coolant mean temperature. As already mentioned, a well identified heat exchanger seems the most suitable choice in the frame of FTT.

Returning to the works of the FTT pioneering authors, one of the main difference of some of these works, compared to the previous derivation, is the assumption of a low temperature of the Carnot cycle (T_1) equal to the heat sink temperature (T_d). This implies very large $\dot{h}_d$ values, so a value much larger than the $\dot{h}_u$ value. This could be named the "Chambadal-Novikov hypothesis" [2,3]. This assumption is almost opposite to our reference PWR case, where $\dot{h}_u$ and $\dot{h}_d$ are similar, however it is a justifiable assumption (see Section 2.2 below), which gives:

$$x_{max} = T_u - T_2 = T_u \left(1 - \sqrt{\frac{T_d}{T_u}} \right).$$

So, T_2 at maximum power output is:

$$T_2 = \sqrt{T_d T_u}.$$

And finally,

$$\eta = 1 - \frac{T_d}{\sqrt{T_d T_u}} = 1 - \sqrt{\frac{T_d}{T_u}} = \eta_t.$$

Another way to determine the maximum power output is to calculate $\dot{P}$ as a function of η instead of x . From Eq. (3),

$$x = \frac{\dot{h}_d}{\dot{h}_u + \dot{h}_d} \left(T_u - \frac{T_d}{1 - \eta} \right).$$

Introducing this value into Eq. (2) gives:

$$\dot{P} = \frac{\dot{h}_u \dot{h}_d}{\dot{h}_u + \dot{h}_d} \left(T_u - \frac{T_d}{1 - \eta} \right) \eta. \quad (4)$$

We see that $\dot{P}$ is proportional to $\dot{h}_u \dot{h}_d / (\dot{h}_u + \dot{h}_d)$, which is the overall heat transfer coefficient of the heat source and heat sink "heat exchangers" in series. This overall heat transfer coefficient has a clear meaning for efficiencies close to zero, where we have:

$$\dot{P} = \frac{\dot{h}_u \dot{h}_d}{\dot{h}_u + \dot{h}_d} (T_u - T_d) \eta.$$

We confirm that $\dot{P}$ (Eq. (4)) is equal to zero for the maximum efficiency (which is a "Carnot efficiency"):

$$\eta_c = 1 - \frac{T_d}{T_u}.$$

The determination of the maximum power output by $\frac{d\dot{P}}{d\eta} = 0$ gives:

$$(1 - \eta)^2 = \frac{T_d}{T_u},$$

leading to η_t of Eq. (1).

We can easily calculate the maximum power output:

$$\dot{P}_{max} = \frac{\dot{h}_u \dot{h}_d}{\dot{h}_u + \dot{h}_d} (\sqrt{T_u} - \sqrt{T_d})^2.$$

The typical variation of $\dot{P}$ (and $\dot{Q}_u$) as a function of η (Eq. (4)) is shown in Fig. 2.

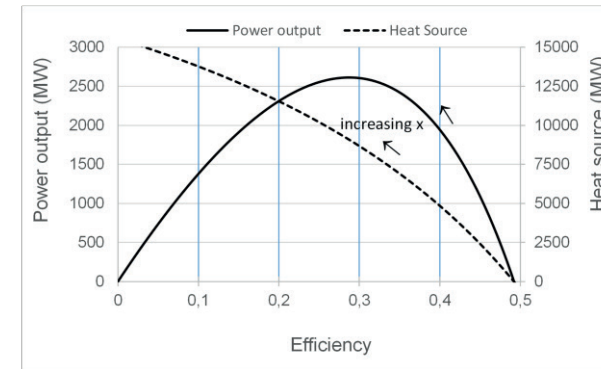


Fig. 2. Power output and cycle heat source as a function of efficiency for $x = T_u - T_2$ variation.

We will see (in Section 3) that such a variation is not observed in a real system. In fact, the previous equations cannot well explain the "trade-off" to be made between efficiency and power output.

2.2. Discussion of the applicability of Eq. (1)

Like Curzon and Ahlborn in [4], we try to see whether Eq. (1) offers good approximations of real system efficiency. For nuclear power plants, assuming $\dot{Q}_u = \dot{h}_u(T_u - T_2)$, the issue is not only whether T_u remains constant, but also the system's ability to handle the large variation of $\dot{Q}_u$. This large variation of $\dot{Q}_u$ behind the FTT approach (see Fig. 2) seems difficult to achieve by the same system. In a PWR reactor, the core's thermal design imposes specific core material temperatures, and therefore, the core power is fixed (by heat conduction within the core materials). The following simple equations can roughly describe the series of heat transfers between the core materials and the thermodynamic cycle:

$$\dot{Q}_u = \dot{h}'(T_{fuel} - T') = \dot{h}''(T' - T_2).$$

So,

$$\dot{Q}_u = \frac{\dot{h}'\dot{h}''}{\dot{h}' + \dot{h}''} (T_{fuel} - T_2),$$

T_{fuel} being representative of the fuel temperature (it could be the mean temperature of the UO_2 pellets), T' is the mean primary circuit temperature, and $\dot{h}'$, $\dot{h}''$ are the equivalent heat transfer coefficients of the core and steam generator, respectively. They are roughly constant for a given core and steam generator size. Both are a combination of conduction and convection heat transfers, but conduction is the dominant phenomenon.

When we assume $T_u = T'$ (said in the introduction to be a rational assumption), it can be seen that the variation in $\dot{Q}_u$ cannot be obtained by varying T_{fuel} ; this variation requires changing the number of nuclear fuel elements (each fuel element producing a constant power). In this case, the FTT approach is justified by the constant value of $\dot{h}'$ (which corresponds to $\dot{h}_u$ here).

On the other hand, if we choose the fuel temperature for T_u , we have:

$$\dot{h}_u = \frac{\dot{h}'\dot{h}''}{\dot{h}' + \dot{h}''}.$$

And to be consistent with the FTT approach (constant $\dot{h}_u$ value), we cannot vary the heat source power by changing the number of nuclear fuel elements. Thus, it is the T' variation that will cause the variation in $\dot{Q}_u$. The large variation of the latter will lead to unusual values of the primary coolant temperature, with:

$$T' = \frac{\dot{h}'T_{fuel} + \dot{h}''T_2}{\dot{h}' + \dot{h}''}.$$

In this case, we can consider that the variations of $\dot{Q}_u$ occur without change in the "heat source system".

Our reference utilisation of Eq. (1) will consider the reactor primary coolant temperatures as the heat source temperature T_u . We have already noted that this temperature has small variations (lower than 40°C for a PWR). For the condenser secondary side (to determine T_d), temperature variations are even smaller (about 10°C). In both cases, the arithmetic mean of these temperatures will be used in Eq. (1).

2.2.1. Case of a 900 MWe PWR

The primary coolant mean temperature is about 304°C [15]. For the condenser secondary side (considering a sea site with direct cooling), the mean temperature is about 20°C. The "gross efficiency", so the ratio of the gross electrical generation over the heat transferred by the steam generators, is about 34% (see [15] or the IAEA Database on Nuclear Power Reactors). Taking into account the generator efficiency, the real thermodynamic cycle efficiency is about 35%.

By using Eq. (1), we obtain approximately 29%, which slightly underestimates the actual value. Of course, with these temperatures (T_u and T_d), the Carnot efficiency is equal to 49%, so a far too optimistic value. But the correct use of the Carnot efficiency requires the use of working fluid temperatures (so T_1 and T_2), thus the steam generator secondary side temperature (typically about 273°C) and the condensing temperature on the condenser's primary side (about 33°C) result in a Carnot efficiency of 44 %. With this latest result, it could be argued that the FTT efficiency provides a better approximation of the real efficiency.

Using these PWR data, we have also estimated the two FTT "heat transfer coefficients". We have: $\dot{h}_u = 90 \text{ MW/K}$ (with a heat source of 2800 MW for a temperature difference of 304°C–273°C), and $\dot{h}_d = 140 \text{ MW/K}$ (with a heat sink of 1820 MW for a temperature difference of 33°C–20°C). We recall that these latter values are calculated for a primary coolant heat source temperature, this heat source being about 2800 MW.

These data are used in the following numerical applications, presented in Section 2.2.3.

2.2.2. Case of a French sodium cooled reactor

For this type of reactor, the mean sodium temperature of the steam generator primary side is about 420°C. For a heat sink with direct condenser cooling, the temperature to be considered is again 20°C. The thermodynamic efficiency of this cycle is about 42%, while 35% is obtained by Eq. (1).

As in the previous PWR case, the FTT efficiency tends to give a lower value (around 20%) than the actual one. However, in both cases, we can agree that it provides a conservative estimate of the plant's efficiency.

2.2.3. Numerical applications for various heat source choices

We propose numerical applications of Eq. (2) and Eq. (3) with $\dot{h}_u$ and $\dot{h}_d$ values estimated in Section 2.2.1. As expected with this heat transfer coefficients choice, for typical reactor power outputs (about 1000 MW) the $T_u - T_2$ value is about 30 K (black dot in Fig. 3) with a much higher efficiency and a much lower power output compared to the FTT optimum, which is obtained for a $T_u - T_2$ value of about 100 K (Fig. 3). For this low T_2 temperature (about 200°C) an acceptable efficiency (0.29) is obtained because this low temperature is used in an ideal thermodynamic cycle: the Carnot cycle.

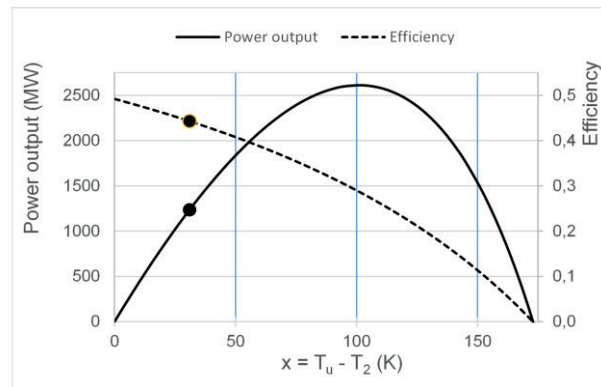


Fig. 3. Cycle power output and efficiency as a function of $x = T_u - T_2$ (here: $T_u = 577$ K, $\dot{h}_u = 90$ MW/K, $T_d = 293$ K and $\dot{h}_d = 140$ MW/K).

The FTT optimum power is about 2600 MW. For realistic use of this optimum, it would be necessary to reduce $\dot{h}_u$ and $\dot{h}_d$ by a factor of three to align with the typical core power. This would mean reducing the heat exchange surfaces of the steam generator and condenser. We are not aware of any commercial nuclear reactor operating under these conditions (i.e. smaller heat exchangers with larger temperature differences).

To be truly comprehensive on the FTT approach, we examine other justifiable choices of the nuclear reactor heat source temperature T_u , in fact to be closer to the nuclear fuel temperatures, see the beginning of Section 2.2. Two T_u choices are proposed: the fuel cladding temperature (in contact with the primary coolant with a temperature close to the saturation temperature of the primary circuit) and the mean fuel pellet temperature (temperature values from reference [15]). The new $\dot{h}_u$ values are

calculated like in Section 2.2.1, of course, the T_u increase leads to a $\dot{h}_u$ decrease.

In both cases, the variation of T_2 (or x), thus of $\dot{Q}_u$, results in a change in the primary coolant temperature (see comments at the beginning of Section 2.2). The FTT optimum of these different assumptions are presented in Table 1.

Table 1. FTT optimal values under different heat source hypotheses, defined by $(T_u, \dot{h}_u)$.

Heat source	PWR reference	Fuel cladding	Mean fuel pellet	Coal plant
T_u (°C)	304	345	715	1000
$\dot{h}_u$ (MW/K)	90	39	6.3	4
x (°C)	101	151	431	644
$\dot{P}$ (MW)	2611	1828	1235	1340
Efficiency	0.29	0.31	0.455	0.52
T_2 (°C)	203	194	284	356
$\sqrt{T_d T_u}$ (°C)	138	153	265	338

It can be observed that, as T_u increases, x and the power outputs move towards more realistic values. On the contrary, as expected with this increase, Eq. (1) now overestimates the actual efficiency. With the $\dot{h}_u$ decrease, the use of the "Chambadal-Novikov hypothesis" is more and more justified, thus the "Chambadal-Novikov temperature" ($\sqrt{T_d T_u}$) becomes an interesting approximation.

To discuss further this issue, a coal-fired power plant is examined. Here, we have a "simpler" heat source at much higher temperatures. We observe a temperature decrease from 1700°C to 300°C for the coal combustion products to produce subcritical steam at approximately 500°C and 100 to 150 bar [16]. Due to the poor heat transfer with the combustion products, the $\dot{h}_u$ value is low. To be consistent with previous calculations, we will continue to determine T_u with a mean temperature and $\dot{h}_u$ like in Section 2.2.1 (for a boiler secondary side temperature of 310°C). The FTT optimum values (Table 1) are perfectly consistent with the previous PWR "mean fuel pellet" case. We have a reasonable FTT value for T_2 (steam boiler being at saturation temperatures between 310°C and 340°C), a good "Chambadal-Novikov temperature" but an efficiency that is much too high (the actual cycle efficiency being about 35%). This latter result is in disagreement with [4]. We believe that Curzon and Ahlborn did not use the correct heat source temperature T_u for their coal power plant case, as it appears that Eq. (1) was applied using the steam temperature at the boiler outlet.

To conclude this section, it is worth noting again the relevance of the temperatures obtained when the "Chambadal-Novikov" hypothesis can be made (thus when $\dot{h}_d \gg \dot{h}_u$). This also concerns the maximum power output equation:

$$\dot{P}_{max} = \dot{h}_u (\sqrt{T_u} - \sqrt{T_d})^2.$$

2.3. Complete determination of the cycle variables

To go further and gain a better understanding of the practical implications of the previous calculations, we propose to calcu-

late all the cycle variables, in particular the cycle mass flowrate. To achieve this, it is necessary to choose two additional cycle parameters (see the description of the system of 10 equations and 13 unknowns in Section 2.1).

To simplify significantly the computations, the working fluid is assumed now to be an ideal gas with an entropy variation for an $(a-b)$ process given by:

$$s_b - s_a = r \left(\ln \left(\frac{T_b}{T_a} \right)^{\frac{\gamma}{\gamma-1}} - \ln \left(\frac{P_b}{P_a} \right) \right),$$

where r being the specific ideal gas constant (in J/kg/K), and γ is the ratio of specific heats at constant pressure and at constant volume.

Of course, such a working fluid is far from the steam/water used in PWR or conventional power plants cycles, but our objective here is only to explore all the possible implications of the FTT analysis, not to get new insights in the optimisation of these energy conversion systems.

For the two additional cycle parameters required, we propose to set the maximum and minimum pressures of the cycle, so P_2 and P_4 , the two remaining cycle pressures are then:

$$P_1 = P_2 \left(\frac{T_1}{T_2} \right)^{\frac{\gamma}{\gamma-1}},$$

$$P_3 = P_4 \left(\frac{T_2}{T_1} \right)^{\frac{\gamma}{\gamma-1}}.$$

From Section 2.1 computations, we know that the cycle entropy balance can be used to determine T_1 as a function of T_2 . With:

$$\dot{m}(s_3 - s_2) = \frac{\dot{h}_u(T_u - T_2)}{T_2},$$

we have:

$$\dot{m} = \frac{\dot{h}_u x}{r T_2 \ln \left(\frac{P_2}{P_3} \right)}.$$

So,

$$\dot{m} = \frac{\dot{h}_u x}{r(T_u - x) \left(\ln \left(\frac{P_2}{P_4} \right) + \frac{\gamma}{\gamma-1} \ln \left(\frac{\dot{h}_d T_d}{\dot{h}_u T_u - x(\dot{h}_u + \dot{h}_d)} \right) \right)}.$$

As seen in Fig. 4, for our "reference PWR case" (with the parameters from Fig. 3), the cycle mass flowrate is almost proportional to x . Therefore, a significantly higher mass flow rate would be required at the point of maximum power output (16 200 kg/s in this case) compared to the high-efficiency range.

Even with a reduction by a factor of three in the heat transfer coefficients (see Section 2.2.3), a very high flow rate is still obtained. These high flowrates are, of course, also a consequence of the choice of a simple air Carnot cycle. Additionally, such variations in flow rate seem difficult to achieve without changes in the heat transfer coefficients (assumed constant in the FTT approach) and in the pressure losses within the heat exchangers, which are also neglected in the FTT approach. For higher T_u assumptions, lower mass flow rates are observed at the FTT optimum, decreasing by a factor of three for the 'mean fuel pellet' case.

Now a very important cycle parameter can be computed, the cycle specific net work $\dot{P} / \dot{m}$ (in J/kg), which is equal to:

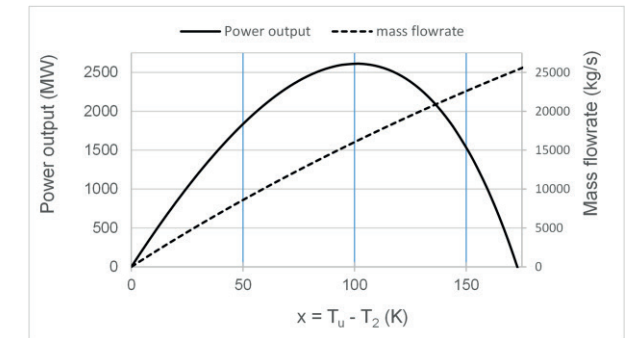


Fig. 4. Variations of the power output and of the mass flowrate (reference PWR case, air working fluid, $P_2 = 200$ bar and $P_4 = 1$ bar).

$$\frac{\dot{P}}{\dot{m}} = r(T_u - x) \left(1 - \frac{T_d \dot{h}_d}{T_u \dot{h}_d - x(\dot{h}_u + \dot{h}_d)} \right) \times \left(\ln \left(\frac{P_2}{P_4} \right) + \frac{\gamma}{\gamma-1} \ln \left(\frac{T_d \dot{h}_d}{T_u \dot{h}_d - x(\dot{h}_u + \dot{h}_d)} \right) \right).$$

In fact, high specific net works mean lower cycle mass flow rates for the same power output. For our "reference PWR case", the specific net work is maximum when x is close to zero, thus when the efficiency is maximum, see Fig. 5. This could justify that PWR systems operate well below the FTT optimum of this reference case.

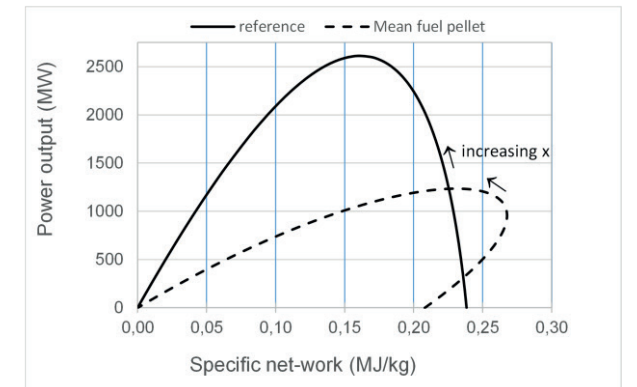


Fig. 5. Power output as a function of the specific net work for $x = T_u - T_2$ variation.

For our "reference PWR case", the specific net work is maximised when x is close to zero (both the power output and the cycle mass flowrate tend to zero), which corresponds to the maximum efficiency, see Fig. 5. For higher T_u assumptions, we have a different behaviour with a well identified maximum value of the specific net work, this value appearing much closer to the FTT optimum power.

To better understand these differences, we attempt to compute the maximum value of the specific net work. This can be achieved by making use of "the Chambadal-Novikov hypothesis" ($\dot{h}_d \gg \dot{h}_u$), which gives:

$$\frac{\dot{P}}{\dot{m}} = r(T_u - x) \left(1 - \frac{T_d}{T_u - x} \right) \left(\ln \left(\frac{P_2}{P_4} \right) + \frac{\gamma}{\gamma-1} \ln \left(\frac{T_d}{T_u - x} \right) \right).$$

The zero of the derivative of this function leads to:

$$\frac{T_d}{T_u - x} + \ln\left(\frac{T_d}{T_u - x}\right) = 1 - \frac{\gamma-1}{\gamma} \ln\left(\frac{P_2}{P_4}\right).$$

Introducing the Lambert W function gives:

$$\frac{T_d}{T_u - x} = W\left(e^{1 - \frac{\gamma-1}{\gamma} \ln\left(\frac{P_2}{P_4}\right)}\right).$$

So,

$$x = \frac{T_u W - T_d}{W}.$$

Using the previously mentioned air characteristics, we have:

$$x = T_u - \frac{T_d}{0.4}.$$

We can see that for high T_u cases, positive x values are obtained at the specific net work maximum, which explains the dashed curve in Fig. 5.

Finally, regardless of the assumption made for T_u , these results confirm that real systems operate close to the maximum of the specific net work, which is an effective way to limit the size of components for a given power output.

To conclude this section, in which we have seen that different values of T_u can be assumed for a complex system like a PWR, we can state the following:

- for “low T_u and high $\dot{h}_u$ ” values, our reference case (the most rational choice for a PWR with a well-identified heat source exchanger), the operating conditions at the FTT optimum are not realistic, while the FTT efficiency provides a conservative estimation of the real efficiency,
- for “high T_u and low $\dot{h}_u$ ” values, the operating conditions at the FTT optimum are more realistic, but the FTT efficiency overestimates the real efficiency like the efficiency given by the Carnot equation.

It is worth noting that this conservative FTT efficiency for the “low T_u ” case is obtained with low maximum cycle temperatures (much lower than the actual ones) balanced by an ideal thermodynamic cycle, the Carnot cycle, without any losses. All these results clearly question and challenge the universal validity of Eq. (1).

3. More realistic relationship between power output and efficiency

We believe that one of the drawbacks of the previous approach is that it does not adequately illustrate the trade-off between maximum power output and maximum efficiency. To address this better, cycle irreversibilities must be introduced. Therefore, we now assume that the compression process (1–2) and the expansion process (3–4) are not isentropic. The cycle is no more an endoreversible cycle.

This was the objective of reference [7] or [8] with the introduction of a constant ratio:

$$R_s = \frac{s_3 - s_2}{s_4 - s_1} < 1.$$

Here $(s_4 - s_1) - (s_3 - s_2)$ is the increase of entropy (strictly positive) resulting from the irreversible compression (1–2) and from the irreversible expansion (3–4).

The update of the calculations from Section 2.1 is straightforward, involving an update of the global entropy balance:

$$\frac{\dot{h}_u(T_u - T_2)}{T_2} = R_s \frac{\dot{h}_d(T_d - T_1)}{T_1}.$$

So:

$$y = \frac{T_d \dot{h}_u x}{x(\dot{h}_u + R_s \dot{h}_d) - R_s \dot{h}_d T_u}.$$

Thus:

$$\dot{P} = \dot{h}_u x \left(1 - \frac{T_d \dot{h}_d}{T_u R_s \dot{h}_d - x(\dot{h}_u + R_s \dot{h}_d)}\right),$$

and

$$\eta = 1 - \frac{T_d \dot{h}_d}{T_u R_s \dot{h}_d - x(\dot{h}_u + R_s \dot{h}_d)}.$$

This latter equation gives:

$$x = \frac{\dot{h}_d}{\dot{h}_u + R_s \dot{h}_d} \left(R_s T_u - \frac{T_d}{1 - \eta}\right),$$

and

$$\dot{P} = \frac{\dot{h}_u \dot{h}_d}{\dot{h}_u + R_s \dot{h}_d} \eta \left(R_s T_u - \frac{T_d}{1 - \eta}\right).$$

The derivation of the previous equation gives the efficiency at the maximum power output:

$$\eta = 1 - \sqrt{\frac{T_d}{R_s T_u}}. \quad (5)$$

This efficiency is lower than the efficiency given by Eq. (1) and remains independent of $\dot{h}_u$ and $\dot{h}_d$. Increasing irreversibilities, thus decreasing R_s , reduces efficiency, which becomes zero for:

$$R_s = \frac{T_d}{T_u}.$$

The relationship between power output and efficiency obtained is shown in Fig. 6.

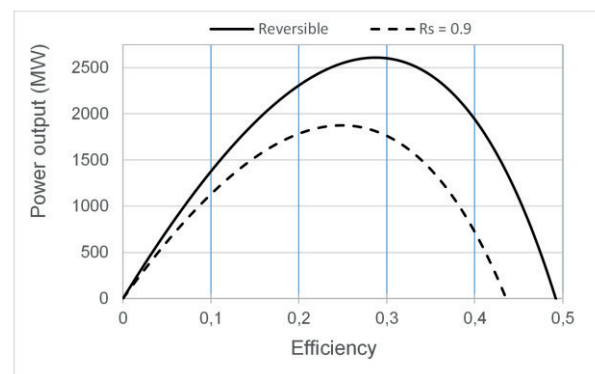


Fig. 6. Power output as a function of cycle efficiency, with and without irreversibilities (reference PWR case).

The curve is very similar to the reversible case, with maximum efficiency occurring at zero power output. We believe it still does not properly represent the power – efficiency relationship.

Our viewpoint is that R_s being constant, the entropy production by irreversibilities does not depend of the pressure ratio of the (1–2) compression and the (3–4) expansion, making this analysis of the “irreversible Carnot cycle” irrelevant. So, in accordance with Feidt and Costea [8], we propose assuming that these irreversibilities will generate an entropy increase equal to:

$$\dot{m}(s_2 - s_1) = \alpha (T_2 - T_1)$$

and

$$\dot{m}(s_4 - s_3) = \alpha (T_3 - T_4).$$

To simplify the calculations, α is assumed to be a constant positive value, identical for the compression and expansion. The cycle entropy balance is now:

$$\frac{\dot{h}_u x}{T_u - x} + \frac{\dot{h}_d y}{T_d - y} + 2\alpha(T_u - x - T_d + y) = 0.$$

This is a second-order equation with respect to y , so a more complicated equation than the cycle entropy equation of Section 2.1. Solving it gives:

$$y = T_d + \frac{1}{2} \left(B - \sqrt{B^2 + \frac{2T_d \dot{h}_d}{\alpha}} \right),$$

with,

$$B = \frac{\dot{h}_u x + (T_u - x)[2\alpha(T_u - x) - \dot{h}_d]}{2\alpha(T_u - x)}.$$

Finally, we have:

$$\dot{P} = \dot{h}_u x + \dot{h}_d \left(T_d + \frac{1}{2} \left(B - \sqrt{B^2 + \frac{2T_d \dot{h}_d}{\alpha}} \right) \right), \quad (6a)$$

and

$$\eta = 1 + \frac{\dot{h}_d}{\dot{h}_u x} \left(T_d + \frac{1}{2} \left(B - \sqrt{B^2 + \frac{2T_d \dot{h}_d}{\alpha}} \right) \right). \quad (6b)$$

The variations of power output and efficiency as a function of x (Fig. 7) show that this time, we have two distinct maximum values of interest: one for efficiency and one for power output.

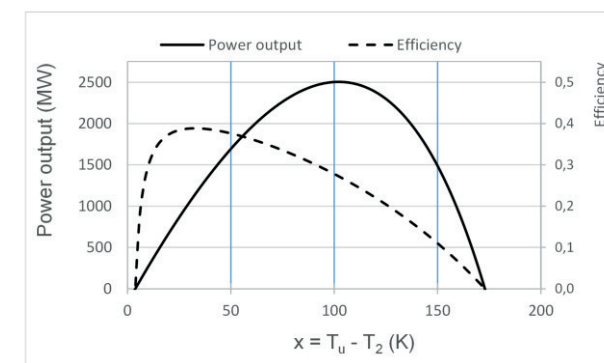


Fig. 7. Power output and efficiency as a function of $x = T_u - T_2$ (reference PWR case and $\alpha = 1000 \text{ W/(K m}^2\text{)}$).

The value of α used ($1000 \text{ W/(K m}^2\text{)}$) is somewhat arbitrary, although it could give entropic losses of the correct order of

magnitude (about 0.5 MW/K for PWR conditions).

For very low x , the thermodynamic cycle is not feasible, at higher x values, very low power outputs are achieved, and the heat generated by irreversible compression and expansion is removed by the heat sink. The maximum efficiency is now obtained for non-zero values of x . For our reference case, these x values (about 30 K , Fig. 7) are close to those of real systems (thanks to the choice of α). The x value at the FTT optimum is still very high ($\sim 100 \text{ K}$) and is accompanied by a significant reduction in efficiency, compared to the maximum value.

For higher T_u cases, the optimum efficiency and power output are obtained at large x values (310 K and 450 K).

The power output as a function of η (Fig. 8) now differs a lot from the variations obtained without irreversibilities (Fig. 2) or with a questionable introduction of irreversibilities (Fig. 6).

The marker (black dot) in Fig. 8 indicates the point corresponding to the actual x values of a PWR reactor, 31 K for our reference case, 442 K for the “mean fuel pellet” case.

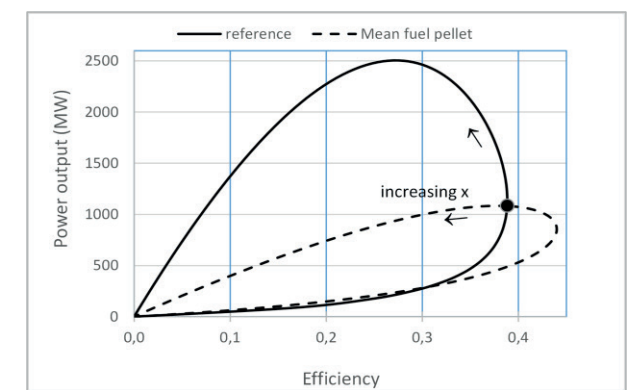


Fig. 8. Power output as a function of efficiency ($\alpha = 1000 \text{ W/(K m}^2\text{)}$).

For the former, this point corresponds to the maximum efficiency, for the latter to the maximum power output. Thus, as without irreversibilities, the “high T_u ” case leads to a more realistic FTT optimum. From the variation of the cycle specific net work with respect to efficiency (Fig. 9), it is observed that, in both cases, the actual PWR temperatures correspond to a point (black dot) very close to the maximum net work.

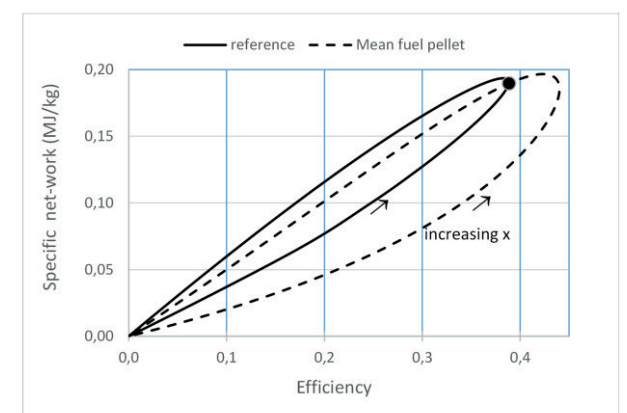


Fig. 9. Cycle net work as a function of efficiency ($\alpha = 1000 \text{ W/(K m}^2\text{)}$).

For the “high T_u ” case, higher efficiency can be achieved at smaller x values, but this would result in unreasonable primary circuit temperatures at the optimum efficiency (the PWR primary pressure being 155 bar). It would be interesting to explore this temperature range with a more realistic thermodynamic cycle.

Finally, as expected, we can observe that with this more realistic approach to accounting for compression and expansion irreversibilities, the efficiency at the maximum power output is no longer independent of $\dot{h}_u$ and $\dot{h}_d$, which is a key characteristic of the FTT optimum, still obtained with Eq. (5).

4. An equation for the efficiency of energy conversion cycles

To further support our view on the too restrictive vision of cycle optimisation issues using the FTT approach, we propose in this section a more general efficiency equation. This equation will better illustrate why the actual efficiency is much lower than the Carnot efficiency and provide another perspective on the system optimisation issue.

A real cycle in steady state is a thermodynamic system, made of working fluids (and eventually of solid media in particular if heat exchangers are part of the system), exchanging heat and work with the surroundings, and having a set of internal irreversible processes (friction, diffusion, conduction, mixing ...), producing entropy.

We assume that the exchange of heat with the surroundings can be described by a series of elementary terms:

- ❖ $\delta\dot{Q}_{u,i}$ with the heat sources, $T_{u,i}$ being here the temperature of the thermodynamic system receiving $\delta\dot{Q}_{u,i}$, and
- ❖ $\delta\dot{Q}_{d,j}$ with the heat sinks, $T_{d,j}$ being the temperature of the thermodynamic system losing $\delta\dot{Q}_{d,j}$ (cycle heat losses to the environment can be accounted for in this term).

These lead to the following variation of the system entropy by heat exchanges:

$$\sum_i \frac{\delta\dot{Q}_{u,i}}{T_{u,i}} + \sum_j \frac{\delta\dot{Q}_{d,j}}{T_{d,j}}.$$

The entropy production by irreversibilities inside the cycle components (turbines, pumps, heat exchangers, ...) will be described by k elementary terms:

$$\sum_k \dot{\sigma}_k.$$

With $\dot{W}_l$ denoting the set of exchange of work with the surroundings, the global energy and entropy balances of the cycle are:

$$\dot{P} = |\sum_l \dot{W}_l| = \sum_i \delta\dot{Q}_{u,i} + \sum_j \delta\dot{Q}_{d,j},$$

$$\sum_i \frac{\delta\dot{Q}_{u,i}}{T_{u,i}} + \sum_j \frac{\delta\dot{Q}_{d,j}}{T_{d,j}} + \sum_k \dot{\sigma}_k = 0.$$

We introduce the mean entropic temperature $\bar{T}_u$ for the heat sources and the mean entropic temperature $\bar{T}_d$ for the heat sinks by:

$$\frac{\sum_i \delta\dot{Q}_{u,i}}{\bar{T}_u} = \sum_i \frac{\delta\dot{Q}_{u,i}}{T_{u,i}},$$

$$\frac{\sum_j \delta\dot{Q}_{d,j}}{\bar{T}_d} = \sum_j \frac{\delta\dot{Q}_{d,j}}{T_{d,j}}.$$

It is worth noting that cycle heat losses to the environment will induce an increase of $\bar{T}_d$ signifying a decrease in efficiency, see Eq. (7) just hereafter.

Thus, the cycle efficiency will be given by:

$$\eta = \frac{\dot{P}}{\sum_i \delta\dot{Q}_{u,i}} = 1 + \frac{\sum_j \delta\dot{Q}_{d,j}}{\sum_i \delta\dot{Q}_{u,i}} = 1 - \frac{\bar{T}_d}{\bar{T}_u} - \frac{\bar{T}_d}{\sum_i \delta\dot{Q}_{u,i}} \sum_k \dot{\sigma}_k. \quad (7)$$

In this equation, we have the two well-known reasons which explain that a real system efficiency is always lower than the Carnot efficiency:

- The difficulty to maintain constant temperatures for both the heat source side and the heat sink side, and as close as possible to the available heat source temperatures and heat sink temperatures;
- The use of real machines and components with internal irreversibilities.

We must acknowledge that a similar equation has already been proposed in [7], but for a Carnot cycle only, with irreversibilities in both the compression and expansion processes, so with constant heat source and heat sink temperatures.

We propose to apply Eq. (7) to the "FTT Carnot cycle" calculated in section 2.1. To do that, the thermodynamic system considered now includes the two theoretical heat exchangers (in addition of the cycle working fluid). With this new definition of the thermodynamic system, T_u and T_d are now temperatures seen by the thermodynamic system. With the variables of Section 2.1, Eq. (7) gives:

$$\eta = 1 - \frac{T_d}{T_u} - \frac{T_d}{\dot{Q}_u} (s_{i,u} + s_{i,d}).$$

$s_{i,u}$ and $s_{i,d}$ being the entropy production by the heat transfer irreversible processes within the two theoretical heat exchangers. We have:

$$s_{i,u} = \dot{Q}_u \left(\frac{1}{T_2} - \frac{1}{T_u} \right) = \frac{\dot{h}_u x^2}{T_u(T_u - x)},$$

$$s_{i,d} = -\dot{Q}_d \left(\frac{1}{T_d} - \frac{1}{T_1} \right) = \frac{\dot{h}_d y^2}{T_d(T_d - y)}.$$

Thus, we have an efficiency depending on x and y (T_u , T_d , $\dot{h}_u$ and $\dot{h}_d$ being fixed variables):

$$\eta = 1 - \frac{T_d}{T_u} - \frac{T_d}{\dot{h}_u x} \left(\frac{\dot{h}_u x^2}{T_u(T_u - x)} + \frac{\dot{h}_d y^2}{T_d(T_d - y)} \right).$$

To eliminate y , we can use again the entropy balance:

$$\frac{\dot{h}_u x}{T_u} + \frac{\dot{h}_d y}{T_d} + \frac{\dot{h}_u x^2}{T_u(T_u - x)} + \frac{\dot{h}_d y^2}{T_d(T_d - y)} = 0.$$

which can be rewritten to obtain the entropy balance from Section 2.1:

$$\frac{\dot{h}_u x}{T_u - x} + \frac{\dot{h}_d y}{T_d - y} = 0,$$

thus:

$$y = \frac{T_d \dot{h}_u x}{x(\dot{h}_u + \dot{h}_d) - \dot{h}_d T_u}.$$

So:

$$s_{i,d} = \frac{\dot{h}_d y^2}{T_d(T_d - y)} = -\frac{\dot{h}_u x}{(T_u - x)} \frac{\dot{h}_u x}{x(\dot{h}_u + \dot{h}_d) - \dot{h}_d T_u}.$$

The efficiency is now:

$$\eta = 1 - \frac{T_d}{T_u} - \frac{T_d}{\dot{h}_u x} \left(\frac{\dot{h}_u x^2}{T_u(T_u - x)} - \frac{\dot{h}_u x}{(T_u - x)} \frac{\dot{h}_u x}{x(\dot{h}_u + \dot{h}_d) - \dot{h}_d T_u} \right).$$

This equation can easily be rewritten as:

$$\eta = 1 - \frac{T_d \dot{h}_d}{T_u \dot{h}_d - x(\dot{h}_u + \dot{h}_d)}.$$

Thus, Eq. (3) from Section 2.1 is obtained.

Equation (7) can also be used to introduce the irreversibilities in the compression process (1–2) and the expansion process (3–4), so:

$$\eta = 1 - \frac{T_d}{T_u} - \frac{T_d(s_{i,u} + s_{i,d})}{\dot{Q}_u} - \frac{2\alpha(T_2 - T_1)}{\dot{Q}_u}.$$

Using previous calculations, one has:

$$\eta = 1 - \frac{T_d}{T_u} - \frac{T_d}{\dot{h}_u x} \left(\frac{\dot{h}_u x^2}{T_u(T_u - x)} + \frac{\dot{h}_d y^2}{T_d(T_d - y)} \right) - \frac{2\alpha(T_u - x - T_d + y)}{\dot{h}_u x}.$$

With the entropy balance to compute y :

$$\frac{\dot{h}_u x}{T_u - x} + \frac{\dot{h}_d y}{T_d - y} + 2\alpha(T_u - x - T_d + y) = 0,$$

we will obtain Eq. (6).

It must be acknowledged that the usefulness of Eq. (7) for computing the system efficiency (as in the previous examples) is very limited. In fact, its interest is essentially academic, serving to clarify energy conversion optimisation issues through:

- temperatures as close as possible to the maximum and minimum available values,
- limiting the irreversibilities of the system’s components and machines.

Of course, optimisation means result in an increase of the system cost. For certain cycles, a compromise has to be made between these two optimisation paths. For example, in a steam Rankine cycle, water reheating, which involves mixing of cold water and steam, allows an increase of the temperature $\bar{T}_u$, but mixing generates irreversibilities. These irreversibilities reduce the cycle efficiency, but the increase in $\bar{T}_u$ has finally a more significant contribution.

5. Final comments and conclusions

The finite-time thermodynamics (FTT) approach to endoreversible cycles enables the formulation of an elegant relationship known as the FTT efficiency. Derivation of this expression requires consideration of a Carnot cycle. However, given the well-known limitations of this idealised cycle, the Carnot efficiency provides only limited insight into the performance of real systems. Consequently, the same limitation may apply to the FTT

efficiency, which is, in essence, a reformulation of the Carnot efficiency expressed in different variables.

It is worth recalling that the derivation of this equation presented in this paper is a quite classical and simple approach in thermal sciences (enthalpy and entropy balances in steady-state conditions), in particular without introducing a “finite time” for the cycle. Indeed, being in steady state, the use of a “finite time” may appear as a paradox.

The use of the FTT efficiency to estimate the efficiency of real systems leads to the problem of an unambiguous choice of the heat source temperature T_u . For a nuclear reactor like a PWR, it seems logical to choose the primary coolant temperature, thus the temperature of the steam generator primary side in order to have a well identified heat exchanger. In this case, we have relatively moderate T_u values, and it is shown that the FTT optimum (the maximum power output which gives the FTT efficiency) gives an acceptable estimation of efficiency (with a slight underestimation, thus a conservative value). But this optimum corresponds to unrealistic operating conditions (much too large differences of temperature within the two FTT heat exchangers). It is these unrealistic conditions which in fact allow the acceptable estimation of efficiency by FTT.

On the contrary, when much larger values of T_u can be assumed, for example by choosing a heat source extending to the nuclear fuel, the FTT optimum was achieved under more realistic operating conditions but with an overestimated efficiency. The determination of all the cycle parameters, obtained with an ideal gas, confirms these conclusions. These results clearly challenge the universality of the FTT efficiency equation.

This discussion about the heat source temperature has highlighted the relevance of the results obtained (cycle temperatures and power output) when the “Chambadal-Novikov” hypothesis can be made, thus when the heat transfer coefficient at the heat sink side is much larger than the heat transfer coefficient at the heat source side. Finally, we recommend to have a well identified thermodynamic system to minimise ambiguity in the selection of temperatures.

Since the FTT approach is driven mainly by maximising cycle work output, we have discussed the “trade-off” between the maximum of efficiency, the maximum of power output and the maximum of specific net work. We have a better illustration of this issue with the introduction of irreversibilities within the cycle and not only on the heat source and heat sink sides. Regardless of the assumption made for T_u , our calculations tend to show that PWR reactors are designed in the high specific net work range.

To further illustrate these cycle optimisation issues, an academic “generalisation” of the Carnot efficiency is presented. This generalisation highlights another important compromise: the trade-off between maximising the heat source temperature (and minimising the heat sink temperature) and limiting the irreversibilities of the system’s components and machines.

Having considered in this paper only Carnot cycles with constant heat source temperatures, complementary analyses and computations on the Rankine cycles and on the Brayton cycles will be proposed in a further paper.

References

[1] Yvon, J. (1955). The Saclay Reactor: Two years experience on heat transfer by means of compressed gas. In *Proceedings of the International Conference on the Peaceful Uses of Atomic Energy*, 8–20 August, Geneva, Switzerland.

[2] Chambadal, P. (1958). Le choix du cycle thermique dans une usine génératrice nucléaire. *Revue Générale de l'Electricité*, 67, 332–345.

[3] Novikov, I.I. (1958). The efficiency of atomic power stations, a review. *Journal of Nuclear Energy II*, 7, 125–128. doi: 10.1016/0891-3919(58)90244-4

[4] Curzon, F.L., & Ahlborn, B. (1975). Efficiency of a Carnot engine at maximum power output, *American Journal of Physics*, 43(1), 22–24. doi: 10.1119/1.10023

[5] Feidt, M. (2017). The History and Perspectives of Efficiency at Maximum Power of the Carnot Engine. *Entropy*, 19(7), 369. doi: 10.3390/e19070369

[6] Ouerdane, H., Apertet, Y., Goupil, C., & Lecoeur, P., (2015). Continuity and boundary conditions in thermodynamics: From Carnot's efficiency to efficiencies at maximum power. *European Physical Journal, Special Topics*, 224, 839–864. doi: 10.1140/epjst/e2015-02431-x

[7] Wu, C., & Kiang, R.L. (1992). Finite-time thermodynamic analysis of a Carnot engine with internal irreversibility, *Energy*, 17(12), 1173–1178. doi: 10.1016/0360-5442(92)90006-L

[8] Feidt, M., & Costea, M. (2019). Improvement of Carnot and Chambadal model of thermomechanical engine by taking account of transfer and production of entropy, *32nd International Conference on Efficiency, Cost, Optimization, Simulation and Environmental Impact of Energy Systems*, 23–28 June, Wrocław, Poland.

[9] Le, V.L., Feidt, M., Kheiri, A., & Jumel, S. (2013). Optimisation de la puissance du cycle organique de Rankine pour le cas supercritique en considérant le cycle endoréversible, *Congrès Français de Thermique*, 28–31 May, Gérardmer, France. doi: 10.13140/2.1.1391.8401

[10] Haseli, Y. (2016). The equivalence of minimum entropy production and maximum thermal efficiency in endoreversible heat engine. *Heliyon*, 2(5), e00113. doi: 10.1016/j.heliyon.2016.e00113

[11] Feidt, M. (2017). Thermodynamique optimale en dimensions physiques finies. *Techniques de l'ingénieur*. doi: 10.51257/a-v1-be8010

[12] Kaushik, S.C., Tyagi, S.K., & Kumar, P. (2017). *Finite Time Thermodynamics of Power and Refrigeration Cycles*, Springer. doi: 10.1007/978-3-319-62812-7

[13] Feidt, M. (2016). *Thermodynamique et optimisation énergétique des systèmes et proceeds*. Lavoisier, Paris.

[14] Engineering school entrance exam topics (2023). *Ecole polytechnique, Ecoles normales supérieures, Filière MP, Epreuve n° 5*. https://www.ens.psl.eu/sites/default/files/23_mp_sujet_phy.pdf [accessed 22 Nov. 2025].

[15] Drevon, G. (1983). *Les Réacteurs nucléaires à eau ordinaire (Collection du CEA) Série synthèse*. Eyrolles.

[16] Suresh, M.V.J.J., Reddy, K.S., & Kolar, A.K. (2012). Thermodynamic analysis of a coal-fired power plant repowered with pressurized pulverized coal combustion. *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, 226(1), 5–16. doi: 10.1177/0957650911418421



Co-published by
Institute of Fluid-Flow Machinery
Polish Academy of Sciences
Committee on Thermodynamics and Combustion
Polish Academy of Sciences

Copyright©2026 by the Authors under licence CC BY-NC-ND 4.0

<http://www.imp.gda.pl/archives-of-thermodynamics/>



Performance evaluation of cogeneration plant at various turbine inlet temperatures

Sumanraju Veerabathini^{a*}, Narsimhulu Sanke^b, Ramamohan Rao Tinnavelli^c

^aMaturi Venkata Subba Rao Engineering College, Nadergul, Hyderabad 501510, Telangana, India
^bOsmania University, University College of Engineering, OU, Hyderabad 500007, Telangana, India
^cSecond Vasavi College of Engineering, Ibrahimbag, Hyderabad 500031, Telangana, India
^{*}Corresponding author email: sumanrajuv@gmail.com

Received: 15.05.2025; revised: 04.03.2026; accepted: 05.03.2026

Abstract

This experimental study examines the efficiencies of a cogeneration plant under various constant turbine inlet temperatures. The study considers the effects of changes in mass flow rate, inlet pressure and inlet temperature on the electrical and overall efficiencies of the system. The findings indicate that the electrical and overall efficiencies are relatively consistent across different mass flow rates but vary slightly across different inlet pressures. The highest overall efficiency and electrical efficiency are obtained at a mass flow rate of 21.56 kg/s, an inlet pressure of 83 kG/cm² and an inlet temperature of 507°C. Under these operating conditions, the overall efficiency reached 58.06% and the electrical efficiency was 26.26%. These findings suggest that the optimum conditions for this system depend on the specific requirements and priorities.

Keywords: Mass flow rate of steam; Electrical efficiency; Overall efficiency; Heat-to-power ratio

Vol. 47(2026), No. 2, 179–186; doi: 10.24425/ather.2026.158684

Cite this manuscript as: Veerabathini, S., Sanke, N., & Tinnavelli, R.R. (2026). Performance evaluation of cogeneration plant at various turbine inlet temperatures. *Archives of Thermodynamics*, 47(2), 179–186.

1. Introduction

Thermal power plants are significant sources of electricity generation around the world and their efficient operation is crucial for meeting the growing energy demand. Cogeneration, also known as combined heat and power (CHP), is a technology that allows for the simultaneous generation of electricity and heat from the same energy source. By utilising the waste heat generated during electricity generation, cogeneration systems can significantly improve the energy efficiency of thermal power plants.

One of the key factors that affects the performance of cogeneration systems is the steam flow parameters, such as pressure, temperature and mass flow rate. These parameters have been shown to have a significant impact on the thermal efficiency, electricity generation, and overall energy utilisation of cogeneration systems.

The purpose of this research is to look into the performance of a thermal power plant cogeneration system by varying steam flow parameters through hourly-based experimentation. This study will provide valuable insights into optimising cogeneration systems and contribute to developing more energy-efficient thermal power plants. Ozdil et al. [1] conducted an in-depth analysis of a 14.25 MW cogeneration plant in Adana, Turkey, utilising the first and second laws of thermodynamics. Their study examined the system's key components and assessed its thermodynamic performance. Findings revealed that exergy destruction in the boiler accounted for a significant 42% of the system's total irreversibility. Gambini and Vellini [2] focused on the development of highly efficient heat and power cogeneration systems and explored strategies for enhancing overall energy efficiency. High-efficiency cogeneration, as defined by compliance with specific criteria for accessing support systems, is rewarded with energy

Nomenclature

CV – calorific value, kJ/kg
 F_i – fuel energy input, kW
 h – specific enthalpy, kJ/kg
 m_s – mass flow rate of steam, kg/s
 P – pressure, kG/cm²
 Q_s – steam input energy, kW
 Q_u – process heat, kW
 T – temperature, °C, K
 W_t – power generated, kW
 $I, \dots, I_0$ – point states in thermodynamic cycle

Greek symbols

δ – heat-to-power ratio

η_e – electrical efficiency, %
 η_o – overall efficiency, %

Subscripts and Superscripts

i – inlet
 o – output

Abbreviations and Acronyms

CHP – combined heat and power
 FW – feed water
 HU – heat utilisation
 4E – energy-exergy-environment-economics

efficiency certificates (white certificates) based on the energy savings achieved annually.

Chicco et al. [3] explored the potential of distributed cogeneration technologies to improve efficiency and cater to niche markets. They evaluated cogeneration plants using three primary metrics: energy savings, environmental impact and financial viability. Their study introduced a method for analysing electrical and thermal production using incremental indicators, with a focus on microturbines and internal combustion engines — the two most common small-scale cogenerative prime movers. Dinakaran et al. [4] advanced the concept of bagasse cogeneration, emphasising the efficient generation of heat and electricity from sugarcane waste (bagasse) in sugar mills. Their research addressed the economic aspects of modern cogeneration systems within the sugarcane industry. By replacing inefficient mill turbines with hydraulic drives and DC (direct current) motors, sugar mills can achieve high operational efficiency (65–75%) and supply surplus power to the grid, thereby boosting revenue streams for sugar plants. Gulhane et al. [5] published a study focused on optimising energy use while minimising its adverse impacts. The paper highlights several key findings based on extensive research and first-hand observations of local power plants. The primary objective was to determine the magnitude and sources of irreversibilities in a 35 TPH (tons per hour) boiler within a 6 MW captive power plant. These insights aim to help system designers identify the processes contributing most to energy losses. Gambini and Vellini [2] worked on advancing high-efficiency cogeneration systems for heat and power production while striving to enhance overall energy efficiency. ‘High-efficiency cogeneration’ refers to systems that meet the necessary conditions and protocols for integration into the cogeneration support framework. Such systems are annually awarded energy efficiency certificates (white certificates) proportional to the energy savings achieved. Gladysz et al. [6] developed a sophisticated algorithm to evaluate the share of cogeneration in district heating systems. The study concluded with a set of empirical equations that demonstrated how various factors influence the optimal cogeneration ratio, both for systems with and without thermal storage. Vazini Modabber et al. [7] conducted a ‘4E (energy-exergy-environment-economics) analysis’ of a power and water cogeneration plant using MATLAB

to evaluate its energy efficiency, total costs and environmental impact. Bhalodia et al. [8] developed a thermal analysis to enhance the efficiency of a 25 MW plant by identifying and quantifying thermodynamic losses within the cogeneration facility. Silveira et al. [9] introduced a thermo-economic functional analysis based on the Second Law of thermodynamics, which was applied to four cogeneration systems under investigation. Assuming constant electricity generation and steam processing rates in the exergy framework, the proposed method aimed to minimise the energy production cost (EPC) of the cogeneration plant. Among the configurations analysed, the most economical system, requiring no additional firing, comprised a gas turbine coupled with a heat recovery steam generator. Kamate et al. [10] analysed a bagasse-based cogeneration system with an extraction condensing steam turbine and backpressure for a typical 2500 TCD (tonnes crushed per day) sugar refinery, focusing on energy efficiency. The study combined traditional energy analysis with advanced energy methods to evaluate overall and component efficiencies and quantify thermodynamic losses. Korobitsyn et al. [11] investigated cogeneration and found that efficiency could be enhanced by modifying the basic cycle to recover heat from turbine exhaust for steam production in a steam turbine bottoming cycle. Barigozzi et al. [12] demonstrated how the performance of a waste-to-energy cogeneration plant could be improved by optimising the condensation system, highlighting the benefits of combining wet and dry cooling systems. Kanoglu et al. [13] evaluated the efficiency of various building cogeneration systems, measuring and analysing energy and exergy efficiencies. The study considered systems utilising steam turbines, gas turbines, diesel engines and geothermal energy for simultaneous production of electricity and heat (e.g. for space heating and hot water). Actual operating data were used for analysis and performance assessment.

Manesh et al. [14] advanced the development of the R-curve concept through sophisticated exergetic, exergoeconomic, and exergoenvironmental analyses. They proposed a systematic approach for optimally designing and assessing cogeneration systems using a detailed cogeneration targeting model. Bisulandu et al. [15] reported that turbine inlet temperature (TIT) is one of the strongest direct drivers of gas turbine performance: increasing TIT typically raises engine specific work and cycle thermal

efficiency, and in combined/CHP plants increases the available heat for the bottoming cycle or process steam, improving overall energy utilisation. Liu et al. [16] examined the effect of ambient temperature and humidity on TIT and overall plant output: higher inlet air temperatures reduce mass flow and compressor performance, lowering net output and often lowering overall CHP electrical efficiency unless inlet-air cooling or other strategies are used. Kazemiani-Najafabadi and Rad [17] emphasised that the several recent 4E studies have quantified that an ‘optimum’ TIT exists when environmental penalties and capital/operating costs (e.g. advanced cooling or blade materials) are accounted for. For CHP, optimal TIT must be balanced against the temperature/quality required by the process steam or district heating load. Mehregan et al. [18] in their study proposed a new combined heat and power (CHP) system configuration featuring a dual prime mover: a proton exchange membrane (PEM) fuel cell and a Stirling engine. The hybrid integration of the fuel cell and Stirling engine enables assessment of potential performance improvements and identification of optimal operating conditions compared to conventional single-prime mover CHP systems. Sanjay et al. [19] evaluated the thermodynamic performance of an MS9001 gas turbine-based cogeneration cycle equipped with a two-pressure heat recovery steam generator (HRSG), considering various blade-cooling methods. In this configuration, one steam drum in the HRSG produces steam to satisfy the turbine blade cooling demand, while a second steam drum supplies steam for process heating. Feidt et al. [20] examined the optimisation of these systems, focusing on maximising exergy while considering practical constraints, such as mechanical energy limitations, heat demand requirements, and physical constraints like heat source availability and temperature limitations of the system.

The purpose of this research is to determine efficiency of a cogeneration plant by varying the steam flow parameters at different constant temperatures at the inlet of the steam turbine.

2. Description of the extraction condensing steam turbine cogeneration plant

The system under study is the most common layout of a modern cogeneration plant in the sugar industry. The plant in this design burns all of the bagasse it produces to generate steam. After meeting the plant’s minimum requirement for process steam, any excess steam is routed to the condenser, where it can potentially generate more electricity.

To estimate the performance characteristics of the steam turbine, it is required to measure the steam parameters at the inlet and stage levels. Steam temperature, pressure and mass flow rate at the inlet of the turbine and at the stages of the turbine are considered as the influencing parameters. To measure these parameters, pressure gauges, thermocouples and Venturi meters are located at the inlet and stage levels.

3. Experimental analysis

3.1. Methodology

The experimental parameters for the cogeneration plant included pressure, steam temperature and mass flow rate, which were

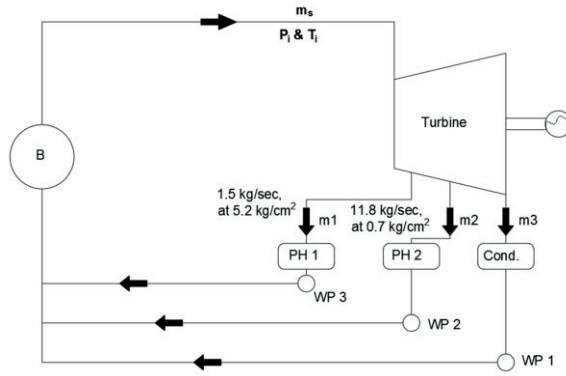


Fig. 1. Extraction condensing steam turbine cogeneration plant: B – boiler; Cond. – condenser; PH1, PH2 – process heat at extraction stages 1 and 2; WP1–WP3 – water pumps, m_1 – m_3 – steam mass flow rates at extraction stages.

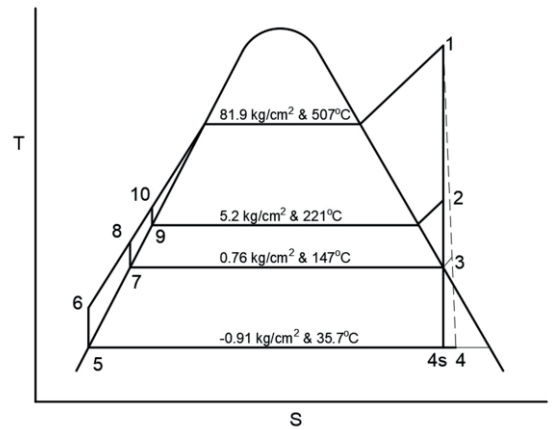


Fig. 2. Temperature-entropy (T - S) diagram.

tested in various combinations. The tests were performed at several constant inlet temperatures. System performance was evaluated in terms of power output and overall efficiency. Figure 1 presents the sequence of steps used to estimate the performance of a steam turbine-based cogeneration system, which is the most widely used configuration in industry (Fig. 2). While the general methodology is applicable to all cogeneration systems, the subsequent formulations specifically focus on steam turbine cogeneration. The key output parameters analysed include electrical efficiency, heat-to-power ratio and overall plant efficiency. Cogeneration involves the simultaneous generation of electricity and useful heat, meaning the system produces two forms of energy. Therefore, a common benchmark is required to assess its efficiency. According to the first law of thermodynamics, the primary performance indicators are electrical efficiency for conventional power plants and overall energy efficiency for cogeneration systems. These metrics assess only the quantitative characteristics of energy conversion.

The electrical efficiency is given by the equation:

$$\eta_e = \frac{W_t}{Q_s} \times 100, \quad (1)$$

where W_t is the electrical power received directly at the generator output, and Q_s is the steam energy input to the turbine:

$$Q_s = m_s(h_1 - h_{10}), \quad (2)$$

where m_s is the steam mass flow rate, h_1 is the specific enthalpy of the steam entering the turbine, and h_{10} is the specific enthalpy of the feed water to the boiler.

The electrical efficiency is calculated in relation to the steam energy input to the turbine at various constant temperatures: 507°C, 500°C and 495°C.

The overall efficiency of the plant is given by the following equation:

$$\eta_o = \frac{W_t + Q_u}{F_i} \times 100, \quad (3)$$

where W_t is the power or electrical energy generated as determined by a digital indicator, and Q_u is the process heat extracted from the turbine at usable temperatures during the in-stage extraction process.

The energy input from the fuel source is given as:

$$F_i = m_f CV, \quad (4)$$

where m_f is the mass flow rate of bagasse and CV is its calorific value of 9500 kJ/kg.

The heat-to-power ratio is expressed by the formula:

$$\delta = \frac{Q_u}{W_t}. \quad (5)$$

The heat energy usage in stages 1 and 2 of the turbine is as follows:

$$\text{heat utilisation at stage 1} = m_1 h_1, \quad (6)$$

$$\text{heat utilisation at stage 2} = m_2 h_2, \quad (7)$$

where m_1 and m_2 represent the steam mass flow rates at the first and second extraction stages, respectively, and h_1 and h_2 denote the specific enthalpy at the respective pressure and temperature conditions.

Finally, the heat output (Q_u) and the total energy output (Q_o) are represented by the following relations, respectively:

$$Q_u = m_1 h_1 + m_2 h_2, \quad (8)$$

$$Q_o = W_t + Q_u. \quad (9)$$

3.2. Results and discussion

The inlet pressure fluctuates between 81.3 and 84.9 kG/cm², as seen in Fig. 3 and Table 2. While the overall efficiency falls between 56.04% and 58.06%, the electrical efficiency falls between 23.84% and 26.26%. The findings show that across the investigated input pressures, both electrical and total efficiency remain largely constant. Nevertheless, the maximum electrical efficiency of 26.26% and the highest overall efficiency of 57.05% are produced by the highest inlet pressure of 84.1 kG/cm². These findings indicate that 84.1 kG/cm² is the ideal input pressure for this system's maximum efficiency (Table 2).

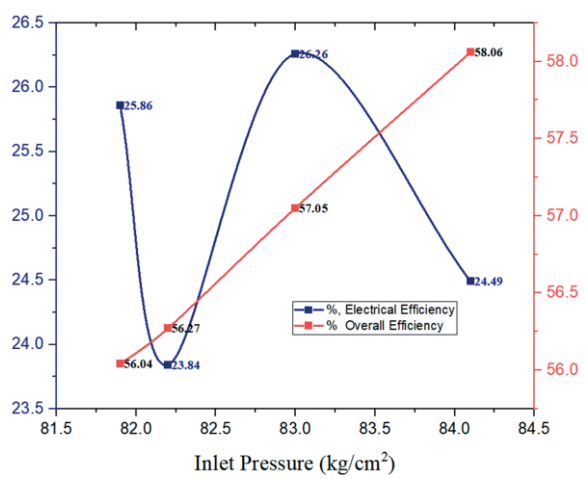


Fig. 3. Electrical efficiency and overall efficiency vs. inlet pressure at 507°C.

Figure 4 and Table 1 and 2 illustrate the electrical efficiency and total efficiency, as a function of the inlet mass flow rate at 507°C. The range of the mass flow rate is 21.56–22.58 kg/s. The overall efficiency ranges from 56.04% to 58.06%, while the electrical efficiency ranges from 23.84% to 26.26%. According to the data, the electrical and total efficiencies remain comparatively constant throughout the various mass flow rates. The best overall efficiency of 58.06% is achieved at the highest mass flow rate of 22.58 kg/s, while the highest electrical efficiency of 26.26% is achieved at a mass flow rate of 21.56 kg/s (Table 2). As a result, the system's ideal settings would be based on the specific requirements and objectives.

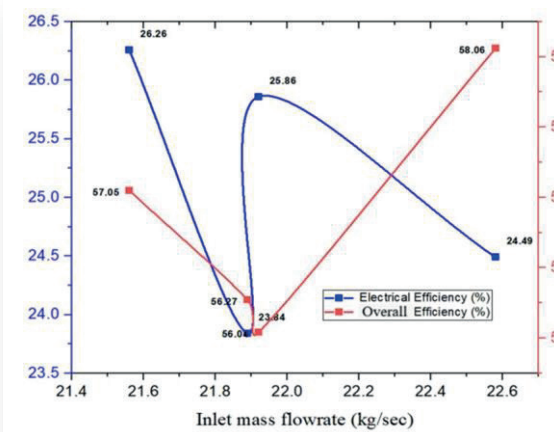


Fig. 4. Electrical efficiency and overall efficiency vs. inlet mass flow rate at 507°C.

The intake pressure varies from 81.7 kG/cm² to 85.3 kG/cm², as shown in Fig. 5 and Table 4. The overall efficiency is between 56.23% and 57.27%, while the electrical efficiency is between 24.31% and 25.53%. The data shows that there are minor differences in electrical and total efficiency between various inlet pressures. According to the findings, an input pressure of 82.8 kG/cm² produced the maximum total efficiency of 57.27%, with an electrical efficiency of 25.2% (Table 4). Accordingly, an input pressure of 82.8 kG/cm² would be the ideal condition for this system, while a mass flow rate of 21.56 kg/s would yield the best electrical

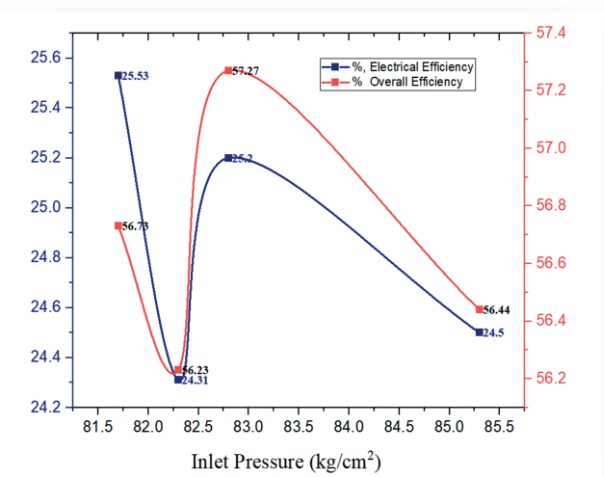


Fig. 5. Electrical efficiency and overall efficiency vs. inlet pressure at 500°C.

efficiency of 26.26%. As a result, the particular needs and priorities would determine the best conditions for this system.

The mass flow rate ranges from 21.67 kg/s to 22.67 kg/s, as shown in Fig. 6 and Tables 3 and 4. The overall efficiency is

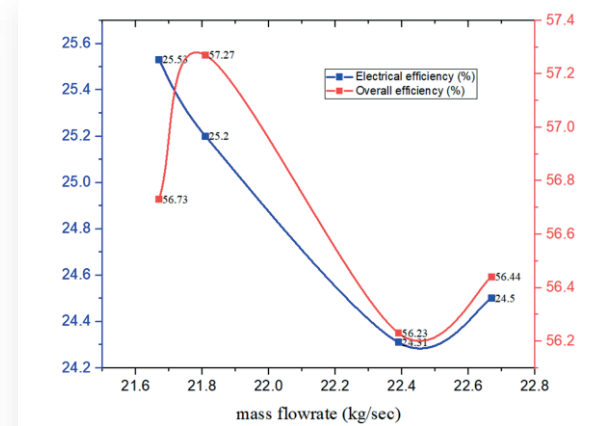


Fig. 6. Electrical efficiency and overall efficiency vs. inlet mass flow rate at 500°C.

Table 1. Conditions for steam flow input parameters and net power output at 507°C.

Inlet				Ext. 1			Ext. 2			FW		W _t , kW
m _f , kg/s	m _s , kg/s	T _i , °C	P _i , kG/cm ²	m ₁ , kg/s	T ₁ , °C	P ₁ , kG/cm ²	m ₂ , kg/s	T ₂ , °C	P ₂ , kG/cm ²	T ₁₀ , °C	P ₁₀ , kG/cm ²	
9.86	21.92	507	81.9	1.47	240	5.11	14.27	156	0.88	174	120	15227
10.14	22.58	507	84.1	1.50	238	5.10	13.05	154	0.87	174	120	14892
9.83	21.89	507	82.2	1.44	241	5.00	13.52	156	0.89	176	120	14061
9.69	21.56	507	83.0	1.61	228	5.20	13.75	146	0.87	170	120	15374

Table 2. Electrical efficiency and overall efficiency at various inlet mass flow rates and pressures at 507°C.

m _f , kg/s	m _s , kg/s	T _i , °C	P _i , kG/cm ²	Q _s , kW	F _i , kW	Q _o , kW	W _t , kW	δ	η _e , %	η _o , %
9.86	21.92	507	81.9	58883.95	94323.37	52860.78	15227	2.47	25.86	56.04
10.14	22.58	507	84.1	60814.43	97192.52	59354.76	14892	2.98	24.49	58.06
9.83	21.89	507	82.2	58991.87	94203.82	53010.65	14061	2.77	23.84	56.27
9.69	21.56	507	83.0	58538.42	92769.25	53859.97	15374	2.50	26.26	57.05

Table 3. Conditions for steam flow input parameters and net power output at 500°C.

Inlet				Ext. 1			Ext. 2			FW		W _t , kW
m _f , kg/s	m _s , kg/s	T _i , °C	P _i , kG/cm ²	m ₁ , kg/s	T ₁ , °C	P ₁ , kG/cm ²	m ₂ , kg/s	T ₂ , °C	P ₂ , kG/cm ²	T ₁₀ , °C	P ₁₀ , kG/cm ²	
9.83	21.81	500	82.8	1.50	221	5.2	11.86	147	0.77	171	120	15480
10.00	22.67	500	85.3	1.56	217	5.2	14.25	149	0.85	169	119	15436
9.75	22.39	500	82.3	1.53	213	4.9	12.31	148	0.70	168	119	14801
9.89	21.67	500	81.7	1.50	190	5.3	12.22	145	0.73	164	120	15483

Table 4. Electrical efficiency and overall efficiency at various inlet mass flow rates and pressures at 500°C.

m _f , kg/s	m _s , kg/s	T _i , °C	P _i , kG/cm ²	Q _s , kW	F _i , kW	Q _o , kW	W _t , kW	δ	η _e , %	η _o , %
9.83	21.81	500	82.8	61432.31	94084.2	52860.78	15480	2.48	25.20	57.27
10.00	22.67	500	85.3	63015.85	95638.4	59354.76	15436	2.49	24.50	56.44
9.75	22.39	500	82.3	60880.51	93486.5	53010.65	14801	2.55	24.31	56.23
9.89	21.67	500	81.7	60645.68	94801.5	53859.97	15483	2.47	25.53	56.73

Table 5. Conditions for steam flow input parameters and net power output at 495°C.

Inlet				Ext. 1			Ext. 2			FW		W _t , kW
m _f , kg/s	m _s , kg/s	T _i , °C	P _i , kG/cm ²	m ₁ , kg/s	T ₁ , °C	P ₁ , kG/cm ²	m ₂ , kg/s	T ₂ , °C	P ₂ , kG/cm ²	T ₁₀ , °C	P ₁₀ , kG/cm ²	
9.61	28.4	495	82.1	1.50	221	5.2	11.86	147	0.77	171	120	15359
8.97	28.6	495	84.3	1.56	217	5.2	14.25	149	0.85	169	119	15341
9.92	30.2	495	83.4	1.53	213	4.9	12.31	148	0.70	168	119	15462
9.19	29.3	495	85.3	1.50	190	5.3	12.22	145	0.73	164	120	15316

Table 6. Electrical efficiency and overall efficiency at various inlet mass flow rates and pressures at 495°C.

m _f , kg/s	m _s , kg/s	T _i , °C	P _i , kG/cm ²	Q _s , kW	F _i , kW	Q _o , kW	W _t , kW	δ	η _e , %	η _o , %
9.61	28.4	495	82.1	60952.64	92171.5	15359	48440.1	2.47	25.20	52.55
8.97	28.6	495	84.3	62619.68	85835.4	15341	48412.6	2.98	23.71	56.40
9.92	30.2	495	83.4	62087.00	94921.1	15462	49370.5	2.77	24.90	52.01
9.19	29.3	495	85.3	62920.36	87987.3	15316	49863.4	2.50	24.34	56.60

between 56.23% and 57.27%, while the electrical efficiency is between 24.31% and 25.53%. At a mass flow rate of 21.81 kg/s, the highest overall efficiency of 57.27% was achieved. At a mass flow rate of 21.67 kg/s, the greatest electrical efficiency of 25.53% was achieved (Table 4).

Figure 7 and Table 6 show the change in the efficiency for inlet pressure varied between 82.1 kG/cm² and 85.3 kG/cm². The total and electrical efficiencies range from 23.71% to 25.20% and from 23.71% to 25.2%, respectively. It should be noted that an increased inlet pressure typically results in an increased overall efficiency. However, this relationship is inconsistent, as it can be observed that at the greatest inlet pressure of 85.3 kG/cm², overall efficiency actually slightly declines. It is evident that an input pressure of 84.3 kG/cm² gave the greatest overall efficiency of 56.60% (Table 6).

Figure 8 and Tables 5 and 6 depict the change in the efficiency for a mass flow rate range of 28.4 kg/s to 30.2 kg/s. The electrical efficiency ranges from 23.71% to 25.2%, and the overall efficiency ranges from 52.01% to 56.6%. The highest overall efficiency of 56.6% was obtained at a mass flow rate of

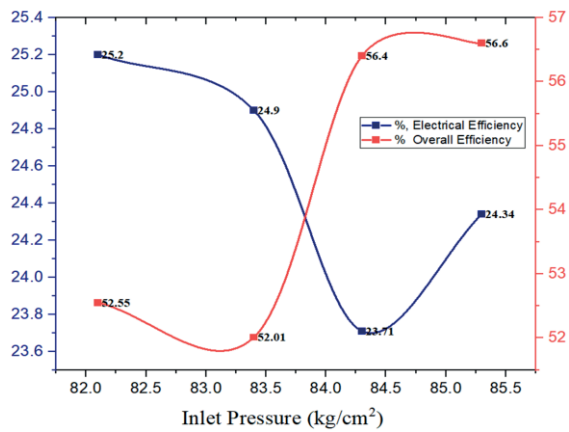


Fig. 7. Electrical efficiency and overall efficiency vs. inlet pressure at 500°C.

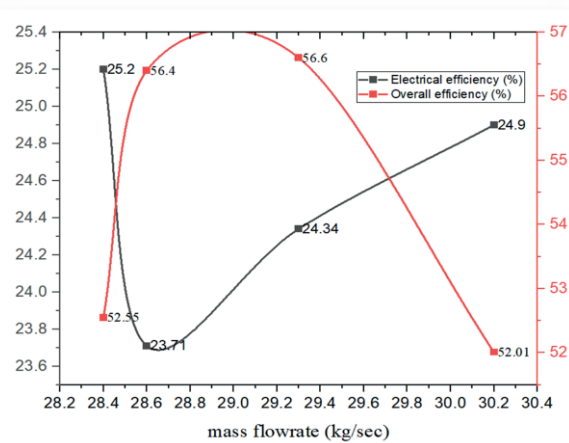


Fig. 8. Electrical efficiency and overall efficiency vs. inlet mass flow rate at 495°C.

22.6 kg/s. The highest electrical efficiency of 25.20% was obtained at a mass flow rate of 28.4 kg/s (Table 6). The maximum electrical and overall efficiencies obtained at constant temperatures of 495°C, 500°C and 507°C are tabulated in Tables 7 and 8, respectively, and depicted in Fig. 9.

The data in Fig. 9 indicate that as the inlet temperature increases, both electrical and overall efficiency increase. The increase in electrical efficiency is 0.63% with an increase in inlet temperature from 495°C to 500°C and 0.79% between 500°C and 507°C. The increase in overall efficiency is 0.67% between increased inlet temperatures of 495°C and 500°C and 0.79% between 500°C and 507°C. Inlet temperature has a significant impact on the efficiency of the system being measured. The higher the temperature of the steam entering the system, the

Table 7. Maximum electrical efficiency at various inlet mass flow rates and pressures, at different inlet temperatures.

T _i , °C	m _s , kg/s	P _i , kG/cm ²	η _e , %
507	21.56	83.0	26.26
500	21.67	81.7	25.53
495	28.40	82.1	25.20

Table 8. Maximum overall efficiency at various inlet mass flow rates and pressures, at different inlet temperatures.

T _i , °C	m _s , kg/s	P _i , kG/cm ²	η _o , %
507	22.58	84.1	58.06
500	21.81	82.8	57.27
495	29.30	85.3	56.60

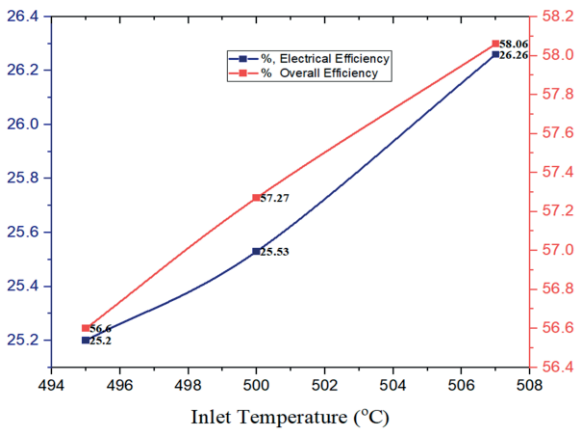


Fig. 9. Effect of inlet temperature on the electrical efficiency and overall efficiency.

higher the efficiency of the system. This could be due to factors such as reduced losses, improved thermal efficiency or increased power output.

4. Conclusions

Based on the given results and discussion, it can be concluded that the steam turbine's electrical efficiency is not only affected by the pressure and mass flow rate of the steam. According to the findings, the greatest efficiency value of 26.26% was achieved at a steam pressure of 83 kG/cm² with a mass flow rate of 21.3 kg/s and a corresponding temperature of 507°C. Conversely, the lowest efficiency value of 23.71% was observed at a pressure of 84.3 kG/cm², a mass flow rate of 22.1 kg/s and a corresponding temperature of 495°C. The maximum overall efficiency of 58.06% was recorded with an inlet steam pressure of 83.1 kG/cm² at a constant temperature of 507°C, while the minimum of 52.01% was observed at an inlet steam pressure of 84.3 kG/cm² at a constant temperature of 495°C. As a result, the efficiency of the plant is influenced by every fractional value, based on practical values, and their continuous changes by altering the parameters of the extraction stage affect the plant's efficiency at different constant temperatures of 507°C, 500°C and 495°C, causing variations in electrical and overall efficiency irrespective of the inlet pressure and steam mass flow rate. Consequently, the parameters of the steam extraction stages play a crucial role in determining the electrical and overall efficiency of the cogeneration power plant.

The highest electrical efficiencies were achieved at constant input temperatures of 507°C, 500°C, and 495°C, which are 26.26% at 83 kG/cm², 25.53% at 81.7 kG/cm², and 25.2 % at 83.4 kG/cm², respectively.

For constant input temperatures of 507°C, 500°C and 495°C, the maximum overall efficiency values reached 58.06% at the pressure of 84.1 kG/cm², 57.27% at the pressure of 82.8 kG/cm², and 55.72% at the pressure of 83 kG/cm², respectively.

Controlling the inlet temperature is one way to improve the efficiency of the system, but it is also important to consider other factors that may be contributing to losses or inefficiencies in the system. It is evident that as working inlet temperatures rose from 495°C to 500°C, the electrical efficiency and total efficiency increased by 0.33% and 0.73% percent, respectively. The values of electrical efficiency and overall efficiency increased by 0.67% and 0.79%, respectively, as operational intake temperatures rose from 500°C to 507°C, indicating that the efficiencies were boosted by raising temperatures and the amount of fuel consumption.

References

- [1] Ozdil, N.F., Tantekin, A., & Pekdur, A. (2018). Performance assessment of a cogeneration system in food industry. *Journal of Thermal Engineering*, 4(2), 1847–1854. doi: 10.18186/journal-of-thermal-engineering.382412
- [2] Gambini, M., & Vellini, M. (2014). High efficiency cogeneration: Performance assessment of industrial cogeneration power plants. *Energy Procedia*, 45, 1255–1264. doi: 10.1016/j.egypro.2014.01.131
- [3] Chicco, G., & Mancarella, P. (2006). Performance evaluation of cogeneration systems: An approach based on incremental indicators. *Proceedings of the 6th WSEAS International Conference on Power Systems* (pp. 34–39), 22–24 Sept., Lisbon, Portugal.
- [4] Dinakaran, C., Purushotham, S., & Harikrishna, S.M. (2016). Study of a cogeneration plant in a sugar mill using bagasse as fuel. *International Electrical Engineering Journal*, 7(6), 2226–2240.
- [5] Gulhane, S.J., & Thakur, A K. (2013). Exergy analysis of boiler in cogeneration thermal power plant. *American Journal of Engineering Research*, 2(10), 385–392.
- [6] Gładysz, P., & Ziębik, A. (2013). Complex analysis of the optimal coefficient of the share of cogeneration in district heating systems. *Energy*, 62, 12–22. doi: 10.1016/j.energy.2013.04.032
- [7] Vazini Modabber, H., & Khoshgoftar Manesh, M.H. (2020). 4E analysis of power and water cogeneration plant based on integrated MED-TVC and RO desalination units. *International Journal of Thermodynamics*, 23(2), 107–126. doi: 10.5541/ijot.616899
- [8] Bhalodia, D.H., & Yadav, S. (2014). Thermodynamic analysis of cogeneration power plant. *International Journal of Engineering Science and Research Technology*, 3(4), 5012–5016.
- [9] Silveira, J.L., & Tuna, C.E. (2004). Thermoeconomic analysis method for optimization of combined heat and power systems – part II. *Progress in Energy and Combustion Science*, 30(6), 673–678. doi: 10.1016/j.pecs.2003.06.001
- [10] Kamate, S.C., & Gangavati, P.B. (2009). Exergy analysis of cogeneration power plants in sugar industries. *Applied Thermal Engineering*, 29(5–6), 1187–1194. doi: 10.1016/j.applthermaleng.2008.06.016
- [11] Korobitsyn, M.A. (1998). *New and advanced energy conversion technologies. Analysis of cogeneration, combined and integrated cycles*. Febodruk BV.
- [12] Barigozzi, G., Perdichizzi, A., & Ravelli, S. (2011). Wet and dry cooling systems optimization applied to a modern waste-to-energy cogeneration heat and power plant. *Applied Energy*, 88(4), 1366–1376. doi: 10.1016/j.apenergy.2010.09.023

- [13] Kanoglu, M., & Dincer, I. (2009). Performance assessment of cogeneration plant. *Energy Conversion and Management*, 50(1), 76–81. doi: 10.1016/j.enconman.2008.08.029
- [14] Khoshgoftar Manesh, M.H., Navid, P., Blanco Marigorta, A.M., Amidpour, M., & Hamed, M.H. (2013). New procedure for optimal design and evaluation of cogeneration system based on advanced exergoeconomic and exergoenvironmental analyses. *Energy*, 59, 314–333. doi: 10.1016/j.energy.2013.06.017
- [15] Bisulandu, M.B.-J.R., Ilinca, A., Tsimba Mboko, M., & Mbozi Mbozi, L. (2023). Thermodynamic performance of a cogeneration plant driven by waste heat from cement kilns exhaust gases. *Energies*, 16(5), 2460. doi: 10.3390/en16052460
- [16] Liu, Z., Ren, X., Yan, Z., Zhu, H., Zhang, T., Zhu, W., & Li, X. (2019). Effect of inlet air heating on gas turbine efficiency under partial load. *Energies*, 12(17), 3327. doi: 10.3390/en12173327
- [17] Kazemiani-Najafabadi, P., & Rad, E.A. (2021). Introducing an optimum gas turbine inlet temperature (TIT) based on a 4E analysis: a case study of northeastern Iran. *Journal of Thermal Analysis and Calorimetry*, 146, 403–413. doi: 10.1007/s10973-020-09955-7
- [18] Mehregan, M., Sheykhi, M., Alizadeh Kharkeshi, B.A., Emamian, A., Aliakbari, K., & Rafiee, N. (2023). Performance analysis and optimization of combined heat and power system based on PEM fuel cell and β type Stirling engine. *Energy Conversion and Management*, 283, 116874. doi: 10.1016/j.enconman.2023.116874
- [19] Sanjay, R., Singh, O.P., & Prasad, B.N. (2009). Comparative performance analysis of cogeneration gas turbine cycle for different blade cooling means. *International Journal of Thermal Sciences*, 48(7), 1432–1440. doi: 10.1016/j.ijthermalsci.2008.11.016
- [20] Feidt, M., & Costea, M. (2012). Energy and exergy analysis and optimization of combined heat and power systems. Comparison of various systems. *Energies*, 5(9), 3701–3722. doi: 10.3390/en5093701



Co-published by
Institute of Fluid-Flow Machinery
Polish Academy of Sciences
Committee on Thermodynamics and Combustion
Polish Academy of Sciences

Copyright©2026 by the Authors under licence CC BY-NC-ND 4.0

<http://www.imp.gda.pl/archives-of-thermodynamics/>



Theoretical analysis on bio-magnetic Casson hybrid nanofluid blood flow through a stenotic artery with a heat source/sink

Suneetha Sangapatnam^{a*}, Gunisetty Ramasekhar^b,
Anusuiya Chakravarty^a, Venugopal Pallapu^a

^aYogi Vemana University, Kadapa 516005, Andhra Pradesh, India

^bRajeev Gandhi Memorial College of Engineering and Technology, Nerawada 'X' Roads, Nandyal 518501, Andhra Pradesh, India

*Corresponding author email: suneethayvu@gmail.com

Received: 10.07.2025; revised: 05.03.2026; accepted: 05.03.2026

Abstract

The current study focuses on numerical analysis of magnetohydrodynamic Casson hybrid nanofluid blood flow carrying hybrid nanoparticles through a porous stenotic artery. In order to improve thermal conductivity and flow management in the presence of heat source/sink, gold-silver nanoparticles are suspended in a base fluid blood. The governing highly non-linear differential equations for momentum and energy were solved employing the MATLAB's bvp4c solver, ensuring high accuracy and stability. The effects of various physical parameters including magnetic parameter, permeability, curvature, Casson parameter, Eckert number and heat source/sink parameter are examined on velocity, temperature, skin friction, and Nusselt number profiles. Results indicate that higher magnetic field strength and permeability declined velocity due to Lorentz and frictional resistance, while increases in curvature and Casson parameter improve flow rates. Temperature distribution is notably elevated by magnetic forces, viscous dissipation and internal heating, with hybrid nanoparticles contributing to enhanced thermal energy. The correctness of the model is validated by its strong agreement with previously reported results. These results offer important new information for biological uses such as tailored medication delivery, heat-assisted therapies and blood flow regulation in diseased arteries, offering a reliable computational framework for future research involving nanoparticle-assisted magnetohydrodynamic flow through porous biological channels.

Keywords: Blood/gold-silver nanoparticles; Bvp4c method; Stenotic artery; Heat source/sink; Casson fluid

Vol. 47(2026), No. 2, 187–199; doi: 10.24425/ather.2026.158686

Cite this manuscript as: Sangapatnam S., Ramasekhar, G., Chakravarty, A., & Pallapu, V. (2026). Theoretical analysis on bio-magnetic Casson hybrid nanofluid blood flow through a stenotic artery with a heat source/sink. *Archives of Thermodynamics*, 47(2), 187–199.

1. Introduction

Human physiological fluids serve essential purposes in lubrication, protection and transportation. Blood is the most intricate physiological ionic fluid that conveys essential components, including organic compounds, nutrients and oxygen to cells, while simultaneously removing waste products from metabolism and carbon dioxide from those cells, constituting 7% of the human

body weight [1]. Among many components that make up blood are enzymes, hormones, sugar glucose, ionised mineral substances, carbon dioxide, cellular membranes, haemoglobin and proteins (albumin, globulin and fibrinogen). It shields the body from mechanical and microbiological damage and controls the body's overall temperature, osmotic pressure and pH [2]. Blood is a non-Newtonian fluid. Examining non-Newtonian fluids in connection to extending surfaces has received attention in rec-

Nomenclature

C_f – skin friction
 C_p – specific heat capacity, J/(kg K)
 (ρC_p) – effective heat capacity, J/(m³ K)
 Ec – Eckert number
 $f'(\eta)$ – dimensionless velocity
 k – thermal conductivity, W/(m K)
 K – porosity parameter
 K^* – permeability of porous medium, m²
 L_0 – length, m
 M – magnetic parameter
 Nu – Nusselt number
 Pr – Prandtl number
 Q – heat source parameter
 r – radial coordinate, m
 R – width of unblocked region, m
 R_0 – artery radius, m
 $Re_r^{1/2}$ – local Reynolds number
 t – time, s
 T – temperature, K
 u, v – velocity component, m/s
 x – axial coordinate, m

Greek symbols

β – Casson parameter
 γ – curvature parameter
 η – similarity index
 θ – dimensionless temperature
 μ – viscosity, kg/(m s)
 ν – kinematic viscosity, m²/s
 ρ – density, kg/m³
 σ – electrical conductivity, S/m
 ϕ_1 – nanoparticle volume fraction
 χ – dimensionless maximum height of the stenosis
 ψ – stream function, m²/s

Subscripts and Superscripts

f – fluid
 hnf – hybrid nanofluid
 w – wall
 $(\cdot)', \dots, (\cdot)'''$ – first- and higher-order differentiation with respect to η

Abbreviations and Acronyms

HNF – hybrid nanofluid
MHD– magnetohydrodynamics

ent decades. The unique and diverse applications of these fluids have garnered significant interest, including their roles in ink-jet printing, polymer manipulation and geological flow systems, among others.

The Casson fluid model is a well-known non-Newtonian rheological model that was created to explain the flow behaviour of viscoplastic fluids with shear-thinning and yield stress properties. It was originally introduced by Casson to characterise the flow of printing inks and has since been extended to model a variety of complex fluids. Human blood, honey, tomato sauce and jelly exemplify the Casson fluid. Numerous academics are concentrating on the examination of Casson fluid behaviour due to its substantial applications. Casson fluid flow behaviour in an unstable boundary layer across an extending surface was studied in [3]. A quantitative study of Casson fluid dynamics on flat and cylindrical surfaces, taking into account elements like chemical reactions and Joule heating, was carried out in [4] using computational techniques to assess the impact of several parameters on heat and mass transfer rates. In [5], the peristaltic flow of a Casson hybrid nanofluid was examined in a wavy-walled channel, deliberating the influence of induced magnetic fields, heat generation and entropy production. The analysis highlights the impact of the Casson fluid parameter on flow impedance and thermal transport, as well as the use of hybrid nanoparticles in enhancing thermal conductivity and heat transfer. Mehdi et al. [6] focused on entropy generation in magnetohydrodynamic Casson fluid flow with the effects of thermal radiation in a porous micro-annulus, and the study provides valuable insights into energy efficiency and thermal management in engineering systems. A numerical simulation of mass and heat movement in hydromagnetic Casson fluid flow under sinusoidal boundary conditions was conducted in [7]. The study offers a thorough examination of how different physical characteristics affect the flow behaviour.

Unusual local intravascular endothelial development in many blood vessel locations, such as the abdominal, cerebral, carotid, thoracic and femoral arteries, can exacerbate arterial stenosis [8]. One of the most harmful outcomes of arterial stenosis is increased resistance and reduced blood flow to the particular vascular bed that the artery supplies. The study of stenotic hemodynamics in narrowed arteries is crucial because it has many applications in cardiovascular diseases such as atherosclerosis, aneurysms, angina or heart attacks, which are among the leading causes of death worldwide. Cardiovascular diseases, which commonly arise in the coronary arteries, are associated with fatty material accumulation inside the arterial wall lumen. Numerous theoretical and experimental studies have examined the specific causes of stenosis formation and how it affects blood flow properties [9].

Coughing, chest pain, fatigue, palpitations, fainting, breathing difficulties, weakness and dizziness are some of the symptoms of artery stenosis. Arteriosclerosis is the name given to the conditions that arise from a partial blockage that impairs blood circulation and poses serious health risks, such as strokes, cerebral ischemia and heart ischemia [10,11]. The blood vessel’s permeability is a crucial characteristic of actual arteries. This characteristic enables oxygen and other essential nutrients to diffuse into the blood flow as it flows through the channel. Hemodynamic transport with artery wall permeability has been the subject of numerous computer studies in recent years. Adopting a cylindrical artery model balances computational simplicity with physiological relevance. It captures key hemodynamic features, that is, velocity profiles – WSS (wall shear stress) – pressure gradients and facilitates efficient simulation of complex flow with nanoparticles and magnetic fields. Pincombe and Mazumdar [12] investigated the impact of post-stenotic dilatation on the blood flow pattern; the non-Newtonian character of the blood was covered in their study. Additionally, Pincombe

et al. [13] discussed the same result in relation to coronary arteries that have post-stenotic dilatation. Wong et al. [14] later provided a theoretical study, in which they computed the blood flow resistance in a damaged artery. From the analysis of Akbar et al. [15], exact solutions for the unsteady pressure-driven peristaltic transport in regular and irregular channels with temperature-dependent nanofluid viscosity were obtained. A comprehensive investigation on hemodynamic flow in a composite artery with a porous wall was carried out in [16]. In the investigation, blood was considered a Newtonian fluid. Moreover, Singh et al. [17] conducted a theoretical investigation of magneto-hydrodynamic blood flow through a porous artery with stenosis, Tripathi [18] performed a theoretical study of blood flow in tilted arteries affected by an obliquely applied magnetic field, and Eldesoky [19] investigated unsteady pulsatile magneto-hydrodynamic blood flow through a porous medium with slip boundary conditions and body acceleration effects. Furthermore, several studies have reported the effects of permeability and numerous stenoses.

The magnetic field is important because it helps to regulate blood flow during surgery or thermal therapy. Therefore, the importance of heat transfer through a cylindrical artery using the Casson liquid and gold nanoparticles (GNPs) was investigated [20]. Using blood Au nanoparticles and micropolar nanoliquid from a curved surface that shrinks and stretches, a new study on radiative flow was carried out by Nisar et al. [21]. Akbar et al. [22] investigated the magnetic force effects and differently shaped nanoparticles in diverging tapering arteries with stenoses via a blood flow model. There has not been any research done on using metallic nanoparticles of different shapes with water as the base fluid. A radially symmetric but axially non-symmetric stenosis is used to depict the blood flow. A further significant contribution of the present work was the examination of the symmetric distribution of wall shear stress associated with resistive impedance, as well as the enhancement of these quantities with increasing stenosis severity. Surgical site infections are reduced when silver nanoparticles are applied to surgically implanted catheters. In the emerging discipline of nanomedicine, it may also serve as a thrombolytic and anticoagulant [23,24]. It also possesses anti-inflammatory, anti-fungal, anti-permeability, anti-angiogenic and drug-carrying qualities. Hence, hybrid nanofluid (HNF) produced from blood with suspended Au and Ag nanoparticles is considered a highly potent and sophisticated chemical instrument for the diagnosis and management of illnesses like HIV (human immunodeficiency virus), cancer, heart issues and other infections. Heat transmission using Ag-Au/blood hybrid nanofluid flow via an artery stenosis was investigated in [25]. A 3D numerical model making use of blood as the base fluid was investigated in [26] for tracking velocity, temperature and pressure in a pathologically dilated artery with Ag-Au. Zaman et al. [27] studied the effect of MHD (magnetohydrodynamics) on power law flow through an angled artery. In a channel subjected to pressure fluctuations and an external magnetic field, Eldesoky et al. [28] examined the MHD flow and peristaltic heat transfer of a suspended viscous fluid.

The influence of heat transfer on MHD flow of micropolar fluids across porous media was shown by [29]. Rehman

et al. [30] studied magnetised and non-magnetised Casson fluid flow around a cylinder, finding reduced flow momentum under a magnetic field. This offers insights into the behaviour of non-Newtonian fluids when magnetic fields are present. Jena et al. [31] looked at how an inclined magnetic field could interact with nanofluid to improve its heat transfer properties. Pattnaik et al. [32] looked at how velocity slip affected the MHD flow of a micropolar fluid across a stretching surface, presenting alterations in velocity due to suction and enhanced rate coefficients. This type of flow can occur in arteries that have been stenosed, or narrowed, due to plaque buildup. A higher value of the Brownian motion parameter enhances the fluid temperature, according to a quantitative analysis of the magnetohydrodynamic blood flow via a permeable artery with radiative influence by [33]. Some noteworthy results in nanofluid flow issues with MHD impact are presented in the following papers: Ali et al. [34] presented magnetohydrodynamic flow of a nanofluid past a moving slender needle, considering nanoparticle aggregation and viscous dissipation effects. In another work, Ali et al. [35] investigated magnetohydrodynamic flow of a tangent hyperbolic nanofluid over a vertically stretched surface using Levenberg-Marquardt backpropagation-based artificial neural networks. Further, Ali et al. [36] examined an unsteady magnetohydrodynamic fluid flow between two parallel plates experiencing uniform oscillations, considering modified Hall current and activation energy effects, and, recently, utilised artificial neural networks to model MHD flow of a tangent hyperbolic nanofluid containing motile microorganisms over a vertically stretched slender surface [37]. Jalali et al. [38] investigated the simulation of pulsatile blood flow for magnetic drug targeting (MDT), focusing on drug particle dispersion through symmetric and asymmetric stenosed vessels. The research of Akbar et al. [39] uses the finite element method (FEM) to produce numerical solutions and artificial neural networks (ANN) to enhance visualisation in a computational study of Prandtl-nanofluid flow inside tapered arteries. The nanofluid exhibits enhanced flow and heat transfer characteristics relative to pure blood, suggesting its effectiveness in biomedical therapies like precise drug delivery and blood flow simulation.

Recent advances in nanomedicine have introduced the use of hybrid nanofluids-mixtures of two or more different nanoparticles dispersed in a base fluid for enhanced drug delivery, diagnostic imaging and thermal therapy. Among various nanomaterials, silver and gold nanoparticles have emerged as particularly attractive due to their high thermal conductivity, biocompatibility, and antibacterial and plasmonic properties. Gold nanoparticles are extensively used in photothermal cancer therapy and biosensors, while silver nanoparticles are prominent in vascular stents, antibacterial coatings and controlled drug release systems [40,41]. The glaring absence of research on the behaviour of non-Newtonian Casson fluid inside a cylindrical stenosed artery in the previously stated studies reveals a substantial research gap. Additionally, more needs to be done to understand the intricate relationships that exist between blood flow in arteries that are cylindrically stenosed and physical processes like the MHD effect. In the research of blood flow in this complex vascular structure, there is still much to learn about the study of na-

nanoparticles, particularly four distinct types: copper (Cu), aluminium oxide, silver (Ag), and gold (Au) of cylindrical shape. The characteristics and applications of various nanoparticle kinds in medicine and drug delivery vary. By closely examining their blood flow dynamics, particularly in complex geometries like cylindrically stenosed arteries, there may be new developments in targeted medication delivery and medical treatments.

To address the aforementioned shortcomings, the use of cylindrical-shaped silver and gold nanoparticles is considered in our work. These particles have great heat conductivity and a high aspect ratio, among other special thermal characteristics. When cylindrical-shaped silver and gold nanoparticles are added to a base fluid, the base fluid's viscosity, flow characteristics and thermal conductivity are all improved. The study by Tanveer et al. [42] investigated magnetohydrodynamic flow and heat transfer of a silver–aluminium oxide (Ag–Al₂O₃) hybrid nanofluid within a trapezoidal cavity. A Cu–Al₂O₃/water hybrid nanofluid flowing via a wavy enclosure exposed to inclined magnetic fields was investigated by [43].

Research by Zaman et al. [44] on the effect of magnetic force on hybrid nanofluid embedded with a heat source/sink on blood flowing beyond stenosed arteries shows that a multiplication of hybrid nanoparticles leads to an amplification of the heat transfer rate in biological systems. Modelling the tissue or vascular environment as a porous medium provides an effective framework for capturing the complex interactions between fluid and structural components. Tissues such as the extracellular matrix (ECM), capillary walls and cellular matrices present significant hydraulic resistance to fluid flow. This porous medium approach becomes particularly critical in pathological conditions like atherosclerosis, tumour-induced angiogenesis, or fibrotic tissues, where abnormal cellular growth or plaque deposition severely alters the porosity and permeability of the medium. In atherosclerotic arteries, for instance, the accumulation of lipids and immune cells narrows the vessel lumen and changes the local porosity, leading to elevated shear stress and reduced perfusion. Similarly, in the tumour microenvironment, irregular and leaky vasculature surrounded by dense stromal tissue behaves like a low-permeability porous medium. Modelling such zones helps predict nanoparticle-based drug delivery, solute transport and pressure gradients more accurately [45,46]. Furthermore, the interaction between blood plasma, macromolecules and therapeutic nanoparticles within these porous media dictates the diffusive and convective mass transfer processes. Therefore, porous media models are extensively used to simulate filtration across capillary membranes, interstitial fluid movement and lymphatic clearance, offering a more physiologically realistic perspective [47,48].

Incorporating non-uniform heat sources and sinks in thermal transport models is essential for representing spatially heterogeneous thermal phenomena in biological tissues [49]. Unlike conventional uniform source models, non-uniform terms account for localised metabolic activities, inflammation-induced heating, external therapies like laser ablation, hyperthermia or cryotherapy, and blood perfusion-dependent heat redistribution [50]. For instance, during laser-based photothermal therapy, the absorption of light by target chromophores generates localised heat

that decays radially. A spatially varying source term can accurately capture this radial thermal profile [51]. Likewise, in feverish tissues, elevated metabolic heat production is not uniformly distributed but varies with inflammatory cytokine gradients. From a mathematical modelling perspective, these heat source/sink terms are commonly introduced as space-dependent functions (e.g., Gaussian or exponential profiles) in the bioheat equation (Pennes or dual-phase-lag models). Their presence significantly impacts temperature fields, tissue damage thresholds and thermal conductivity profiles, especially under transient or nonlinear physiological conditions. Moreover, entropy generation due to thermal gradients and viscous dissipation becomes pronounced in such scenarios, making it a valuable metric in the thermodynamic optimisation of hyperthermia protocols and heat-sensitive nanoparticle delivery systems [52].

In this work, the movement of blood containing Au–Ag nanoparticles in a Casson hybrid nanofluid through a stenosed artery embedded in a porous medium is examined in the presence of thermal radiation and a heat source/sink. The model incorporates magnetohydrodynamics and a porous medium, which are mainly used in physiologically and biomedically relevant contexts. The study's main focus is on analysing the impacts of several parameters, including the magnetic parameter, radiation parameter and porosity parameter. Numerical solutions are found for the governing nonlinear partial differential equations using the bvp4c method in the MATLAB solver. The present study has applications in various sectors in microcirculatory transport, hyperthermic tumour ablation and nanoparticle-assisted drug targeting, offering potential improvements in clinical diagnostics and personalised therapy planning.

2. Motivation of the proposed model

Cardiovascular diseases involving arterial stenosis require precise modelling of blood flow to improve diagnostic and therapeutic techniques. The effect of magnetohydrodynamics on blood moving through arteries with cosine-shaped stenosis was examined. This study is motivated by the need to bridge that gap by exploring hybrid nanoparticle-enhanced Casson fluid flow through a porous stenotic artery under magnetic and thermal influences. Casson fluid models better represent blood rheology under low shear conditions, accounting for the yield stress exhibited by blood in microcirculatory systems. Gold–silver nanoparticles offer superior heat conduction, while magnetic fields provide control over targeted blood flow modulation. By numerically analysing the impact of key parameters, this work aims to offer insights relevant to biomedical uses such as heat treatments and magnetically guided medication distribution, contributing to more effective and minimally invasive cardiovascular treatment strategies and advancing biomedical technologies reliant on magneto-thermal control of blood flow.

Research questions

- 1) How bio-magnetic forces (Lorentz force) influence hybrid nanofluid blood flow in a stenotic artery?
- 2) How Casson rheology modifies velocity, shear stress and temperature distribution?

- 3) How a heat source/sink affects thermal regulation within a partially occluded artery?
- 4) How hybrid nanoparticles enhance heat transfer and modify hemodynamic resistance?

3. Mathematical formulation of the problem

- 1) In the mathematical analysis, the incompressible 2D Casson hybrid nanofluid flow through a stenosed artery is assumed. Moreover, the effects of the magnetic field, porous medium and heat source/sink on the Casson hybrid nanofluid flow through a stenosed artery are examined.
- 2) Arterial blood is assumed to flow through an incompressible stenosed artery, and the characteristic half-length of the stenosis is taken as $L_0/2$. The cylindrical coordinates are considered (r, θ, x) , where the x -axis is taken along the direction of the horizontal artery, while θ and r represent the radial and circumferential directions, respectively.
- 3) In the current model, the flow is considered along the axial direction x and r is perpendicular to the flow. The temperature T is set to the arterial wall temperature T_w , signifying that the wall is preserved at a constant temperature.
- 4) In the physical model, blood flow is along the x -axis and the r -axis is taken perpendicular to the blood flow, as presented in Fig. 1; thermophysical properties of hybrid nanofluid are shown in Table 1.
- 5) As Fig. 1 portrays, blood is moving over a stenosed blood artery with an unobstructed area of $2R_0$; $R(x)$ is the radius of the artery and χ is the maximum height of stenosis.

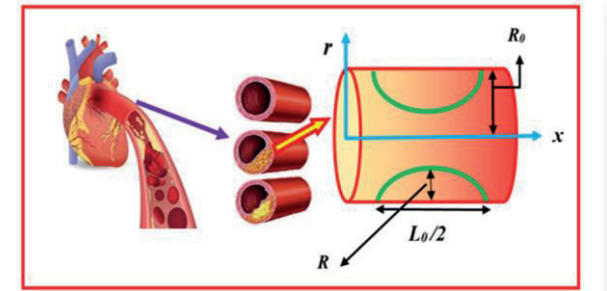


Fig. 1. The artery's physical structure.

Table 1. Thermophysical characteristics of base fluid and nanoparticles (Shahzad et al. [57]).

Property	Unit	Blood	Silver (Ag)	Gold (Au)
ρ	kg/m ³	1063	10 500	19 282
C_p	J/(kg K)	3594	235	129
k	W/(m K)	0.492	429	310
σ	1/(Ω m)	0.667	6.3×10 ⁷	4.1×10 ⁶
Pr	–	21		

A stenosis profile is constructed to represent the stenosed region as follows:

$$\frac{R(x)}{R_0} = \begin{cases} 1 - \frac{\chi}{2} \left[1 + \cos\left(\frac{4\pi x}{L_0}\right) \right], & -\frac{L_0}{4} < x < \frac{L_0}{4} = R_0, \\ 1, & \text{otherwise.} \end{cases} \quad (1)$$

The mathematical flow equations are as follows [53–56]:

$$\frac{\partial(ru)}{\partial x} + \frac{\partial(rv)}{\partial r} = 0, \quad (2)$$

$$u \left(\frac{\partial}{\partial x} + v \frac{\partial}{\partial r} \right) u = \frac{\mu_{hnf}}{\rho_{hnf}} \left(1 + \frac{1}{\beta} \right) \frac{\partial}{r \partial r} \left(r \frac{\partial u}{\partial r} \right) - \frac{\mu_{hnf}}{\rho_{hnf} K^*} u - \frac{\sigma_{hnf} B^2}{\rho_{hnf}} u, \quad (3)$$

$$\left(u \frac{\partial}{\partial x} + v \frac{\partial}{\partial r} \right) T = \frac{k_{hnf}}{(\rho C_p)_{hnf}} \frac{\partial}{r \partial r} \left(r \frac{\partial T}{\partial r} \right) + \frac{\sigma_{hnf} B^2 u^2}{(\rho C_p)_{hnf}} + \frac{Q_0}{(\rho C_p)_{hnf}} (T - T_\infty), \quad (4)$$

where u and v are the velocity components, T is the temperature, σ_{hnf} , μ_{hnf} , ρ_{hnf} and k_{hnf} are the electrical conductivity, viscosity, density and thermal conductivity of hybrid nanofluids, respectively, β is the Casson parameter, $(\rho C_p)_{hnf}$ is the effective heat capacity, K^* represents the permeability of the porous medium, B is the applied magnetic field strength, and Q_0 is the volumetric flow rate.

The boundary conditions are:

$$\left. \begin{aligned} u &= u_0, \quad v = 0 \text{ and } T = T_w & \text{at } r = R(x), \\ u &\rightarrow 0, \text{ and } T \rightarrow T_\infty & \text{as } r \rightarrow \infty \end{aligned} \right\}, \quad (5)$$

where u_0 is the characteristic velocity.

The thermo-physical nature is determined by:

$$\left. \begin{aligned} A_1 &= \frac{\mu_{hnf}}{\mu_f}, & A_2 &= \frac{\rho_{hnf}}{\rho_f}, & A_3 &= \frac{\sigma_{hnf}}{\sigma_f}, \\ A_4 &= \frac{(\rho C_p)_{hnf}}{(\rho C_p)_f}, & A_5 &= \frac{k_{hnf}}{k_f}, \end{aligned} \right\}. \quad (6)$$

where A_1, A_2, A_3, A_4, A_5 denote the viscosity ratio parameter, density ratio parameter, electrical conductivity ratio parameter, effective heat capacity ratio and thermal conductivity ratio parameters, respectively.

The appropriate self-similarity modulations are:

$$u = \frac{u_0 x}{L_0} f'(\eta), \quad v = -\frac{R}{r} \sqrt{\frac{u_0 v_f}{L_0}} f(\eta), \quad \theta(\eta) = \frac{T - T_\infty}{T_w - T_\infty}, \quad \eta = \frac{r^2 - R^2}{2R}, \quad (7)$$

where v_f is the kinematic viscosity of the fluid and η is the similarity index.

The modified Eq. (1) can be written as:

$$f = \begin{cases} 1 - \frac{\chi}{2} [1 + \cos(4\pi x)], & -\frac{1}{4} < x < \frac{1}{4}, \\ 1, & \text{otherwise.} \end{cases} \quad (8)$$

where $f = \frac{R(x)}{R_0}$, and $\chi = \frac{\lambda}{R_0}$ is the non-dimensional measure of stenosis in the referring artery.

Substituting Eq. (7) into Eqs. (2)–(4), modified dimensionless equations are obtained:

$$\frac{\mu_{hnf}/\mu_f}{\rho_{hnf}/\rho_f} \left[(1 + 2\gamma\eta) \left(1 + \frac{1}{\beta} \right) f''' + 2\gamma f'' \right] + f f'' - f'^2 - \frac{\mu_{hnf}/\mu_f}{\rho_{hnf}/\rho_f} K f' - \frac{\sigma_{hnf}/\sigma_f}{\rho_{hnf}/\rho_f} M f' = 0, \quad (9)$$

$$\frac{k_{hnf}/k_f}{(\rho c_p)_{hnf}/(\rho c_p)_f} \frac{1}{Pr} [(1 + 2\gamma\eta)\theta'' + 2\gamma\theta'] + f\theta' - f'\theta + \frac{\sigma_{hnf}/\sigma_f}{(\rho c_p)_{hnf}/(\rho c_p)_f} MEc(f')^2 + \frac{PrQ}{(\rho c_p)_{hnf}/(\rho c_p)_f} \theta = 0, \quad (10)$$

where:

$$\gamma = \frac{v_f L_0}{u_0 R^2}, \quad K = \frac{\mu_f L_0}{\rho_f K^* u_0}, \quad M = \frac{\sigma_f B^2 L_0}{u_0 \rho_f},$$

$$Pr = \frac{k_f}{(\mu c_p)_f}, \quad Ec = \frac{U_w^2}{c_p(T_w - T_\infty)},$$

are the curvature parameter, porosity parameter, magnetic parameter, Prandtl number, Eckert number, and U_w is the wall velocity.

The dimensionless boundary conditions are:

$$\left. \begin{aligned} f(\eta) = 0, \quad f'(\eta) = 1, \quad \theta(\eta) = 1, \quad \eta = 0, \\ f'(\eta) \rightarrow 0, \quad \theta(\eta) \rightarrow 0, \quad \eta \rightarrow \infty \end{aligned} \right\}. \quad (11)$$

The physical quantities of C_f and Nu are:

$$C_f = \frac{2\tau_w}{\rho_f U_w^2}, \quad \tau_w = \mu_{hnf} \left. \frac{\partial u}{\partial r} \right|_{r=R}, \quad (12)$$

$$Nu = \frac{x q_w}{k_f (T_w - T_\infty)}, \quad q_w = -k_{hnf} \left. \frac{\partial T}{\partial r} \right|_{r=R}, \quad (13)$$

where τ_w is the wall shear stress and q_w represents the heat flux.

Equations (12)–(13) take the following dimensionless form:

$$Re_r^{1/2} C_f = A_1 f''(0), \quad (14)$$

$$\xi_1' = \xi_2, \quad \xi_2' = \xi_3,$$

$$\xi_3' = - \left[\frac{1}{(1+2\gamma\eta)(1+\frac{1}{\beta})} \left[\frac{A_1}{A_2} 2\gamma\xi_3 + \xi_1\xi_3 - \xi_2^2 - \frac{A_1}{A_2} K\xi_2 - \frac{A_3}{A_2} M\xi_2 \right] \right],$$

$$\xi_4' = \xi_5$$

$$\xi_5' = - \left[\frac{1}{(1+2\gamma\eta)\frac{A_5}{A_4} Pr} \left[\frac{A_5}{A_4} \frac{1}{Pr} 2\gamma\xi_5 + \xi_1\xi_5 - \xi_2\xi_4 + \frac{A_3}{A_4} MEc\xi_2^2 + \frac{1}{A_4} PrQ\xi_4 \right] \right],$$

and the boundary conditions are:

$$\left. \begin{aligned} \xi_1(\eta) = 0, \quad \xi_2(\eta) = 1, \quad \xi_4(\eta) = 1 \quad \text{at } \eta = 0, \\ \xi_2(\eta) \rightarrow 0, \quad \xi_4(\eta) \rightarrow 0 \quad \text{as } \eta \rightarrow \infty \end{aligned} \right\}.$$

For the numerical computations, a convergence tolerance of 10^{-6} and a step size of 0.01 were employed throughout the analysis. When compared with the shooting approach and the bvp4c MATLAB software, this level of accuracy is much greater.

$$Re_r^{1/2} Nu_r = -A_5 \theta'(0), \quad (15)$$

where $Re_r^{1/2}$ represents the local Reynolds number, and Nu_r denotes the local Nusselt number.

4. Numerical methodology

The highly nonlinear coupled ordinary differential equations (ODEs) (9)–(11) are challenging to resolve numerically. A number of methodologies may be employed to solve nonlinear differential equations, including boundary layer movement. Several methods may be used, which involve semi-analytical, analytical, and computational approaches. In order to solve this numerically, the present study used the bvp4c MATLAB package.

Step 1: New unknown variables are introduced into the system of higher-order non-linear ordinary differential equations, following the procedure:

$$\begin{aligned} f &= \xi_1, \quad f' = \xi_2, \quad f'' = \xi_3, \quad f''' = \xi_3', \\ \theta &= \xi_4, \quad \theta' = \xi_5, \quad \theta'' = \xi_5'. \end{aligned}$$

Step 2: To solve the higher-order ODEs effectively, this method reduces them to a system of first-order ODEs, to simplify the computation in MATLAB solver by utilising syntax of the bvp4c solver like `sol = bvp4c(codesys, bcfun, solinit)`:

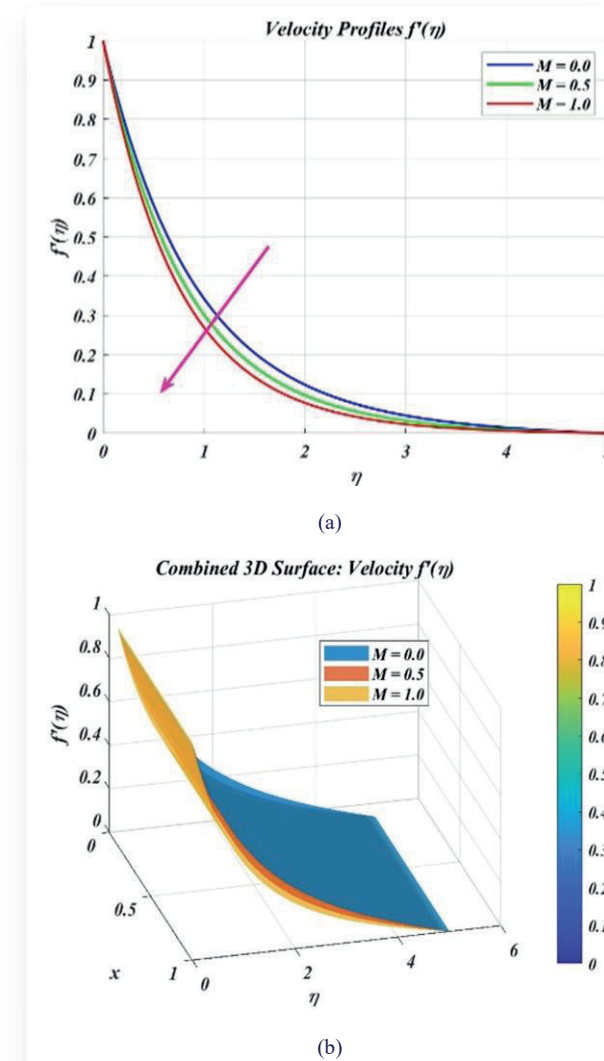


Fig. 2. Impact of M on the velocity profile.

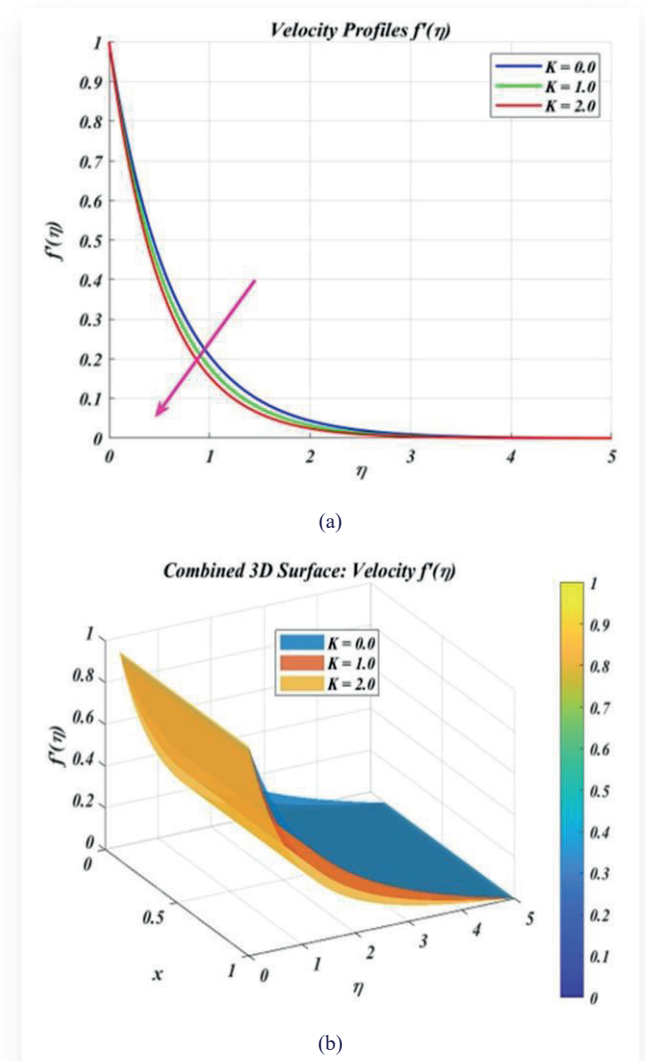


Fig. 3. Impact of K on the velocity profile.

graphs reveal various key characteristics. The study used computational methods to find the right answer to the altered highly chaotic coupled equations that described how the Casson nanofluid flowed along a stenotic artery. This research phase made use of the bvp4c procedure in the MATLAB solver, as well as the current examination and a number of significant active parameters. Within these ranges, the figures show the impact of several significant factors: $0.0 < M < 0.5$, $0.0 < K < 0.2$, $0.0 < \gamma < 0.4$, $0.0 < \beta < 1.0$, $0.0 < Q < 0.6$, $0.0 < Ec < 0.2$.

Figure 2 shows a declining trend in the velocity profile as M (magnetic field strength) increases. The presence of an electrically charged fluid has been shown to generate Lorentz power, which works against the natural inclination of blood to flow through arteries in the same direction as a magnetic field that is being applied.

The charged and magnetic particles in blood will probably spin when a magnetic field is applied to it because of the magneto force. With increasing resistance, more flow energy is converted into frictional losses, thereby diminishing the fluid's capacity to maintain its velocity. Figure 3 shows a decline in the velocity profile with a higher value of K (permeability parameter), as the resistance dissipates more of the flow energy into frictional

losses, reducing the fluid's capacity to sustain velocity. Figure 4 illustrates how the curvature parameter (γ) affects the velocity profile; that is, the velocity profile increases as γ values rise. Physically, the blood vessel's curvature changes when the stenosis causes the arterial walls to shrink. Changes in blood circulation and associated patterns may result from this occurrence, which could have significant physiological implications.

As observed from Fig. 5, the velocity profile increases as the Casson parameter (β) becomes larger; this trend occurs because the fluid's resistance to deformation decreases when its yield stress decreases, which is correlated with a larger β value. Consequently, the fluid exhibits a more Newtonian-like behaviour, allowing it to flow more freely. This effect is particularly noticeable in complex geometries such as stenosed arteries, where reduced internal resistance significantly enhances the fluid motion. It can be seen from Fig. 6 that the magnetic field creates resistance to the fluid's mobility, slowing down the flow. The opposing force from the magnetic field results in internal energy dissipation in the form of heat, a phenomenon known as Joule heating. Moreover, the reduced flow velocity weakens convective heat transfer, allowing thermal energy to accumulate in the

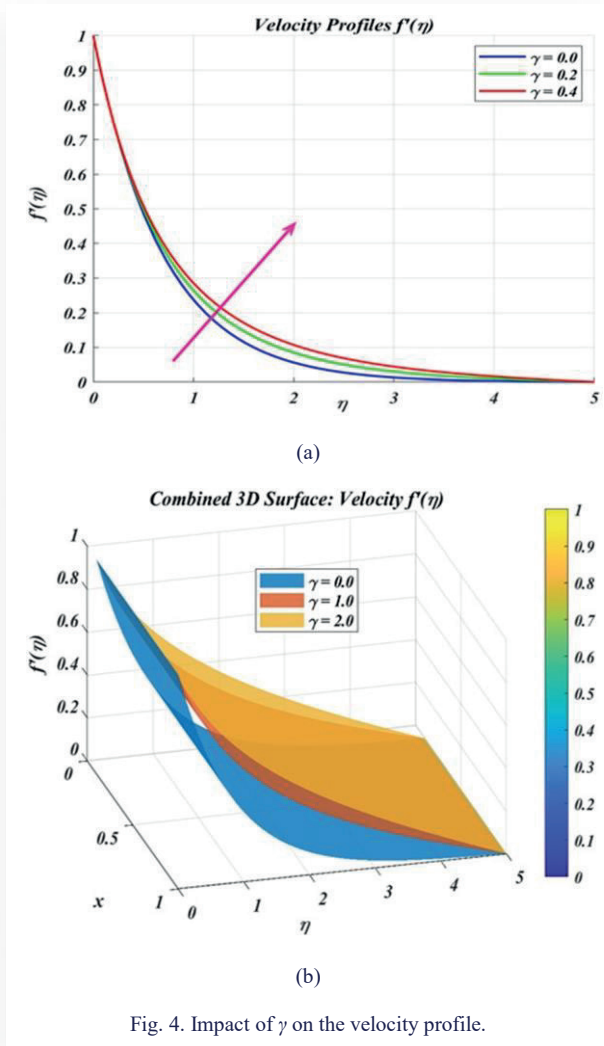


Fig. 4. Impact of γ on the velocity profile.

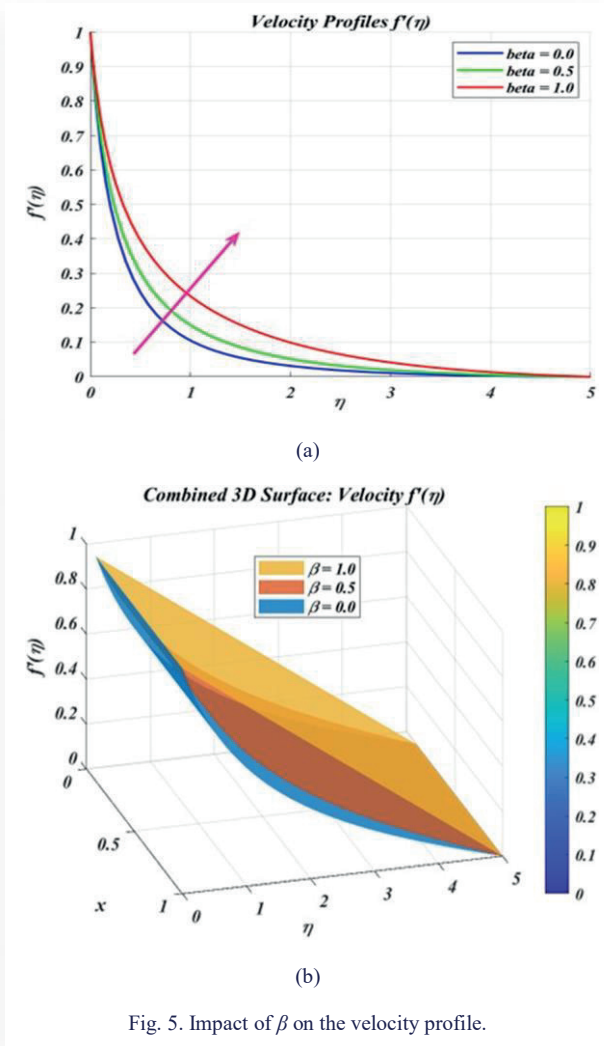


Fig. 5. Impact of β on the velocity profile.

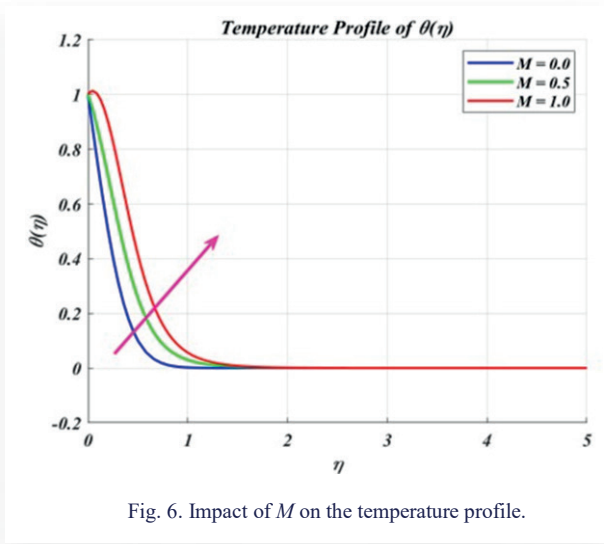


Fig. 6. Impact of M on the temperature profile.

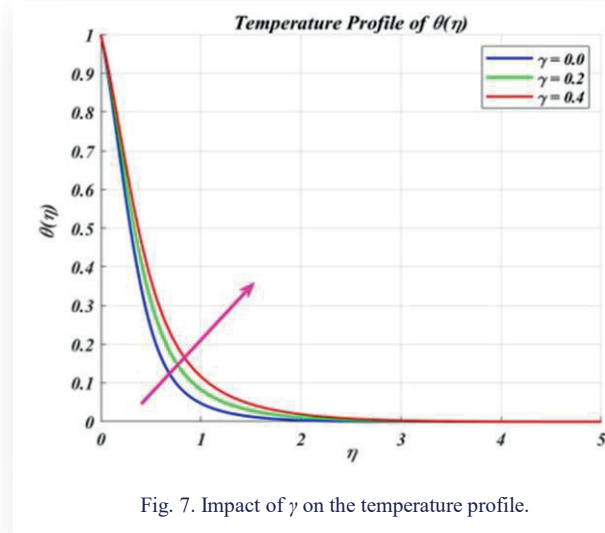


Fig. 7. Impact of γ on the temperature profile.

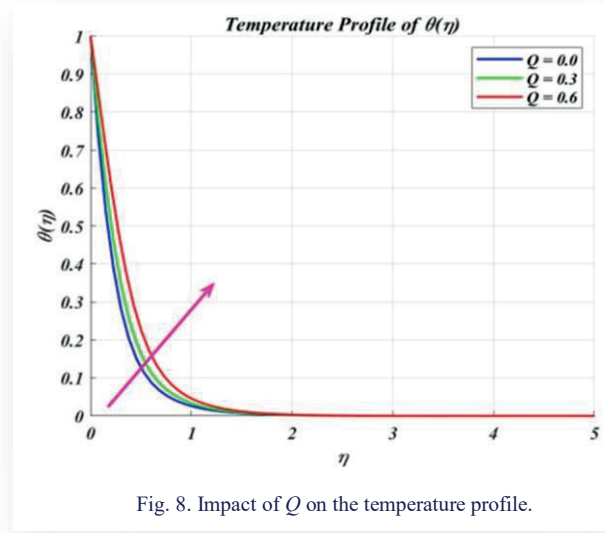


Fig. 8. Impact of Q on the temperature profile.

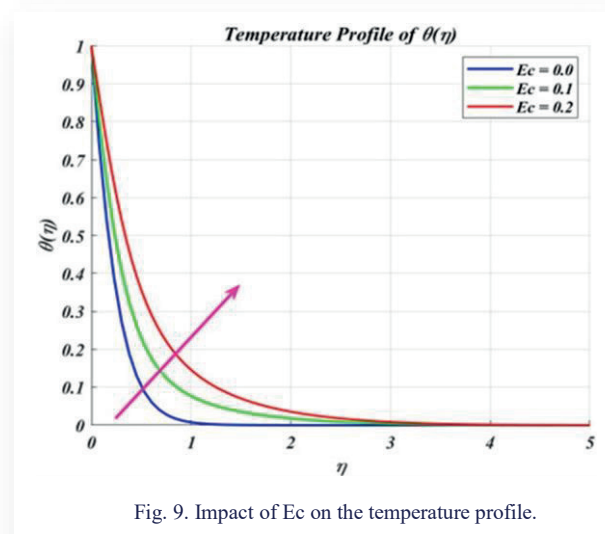


Fig. 9. Impact of Ec on the temperature profile.

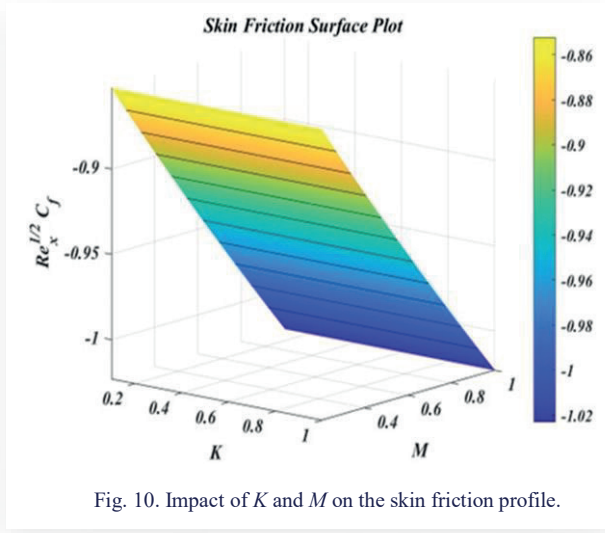


Fig. 10. Impact of K and M on the skin friction profile.

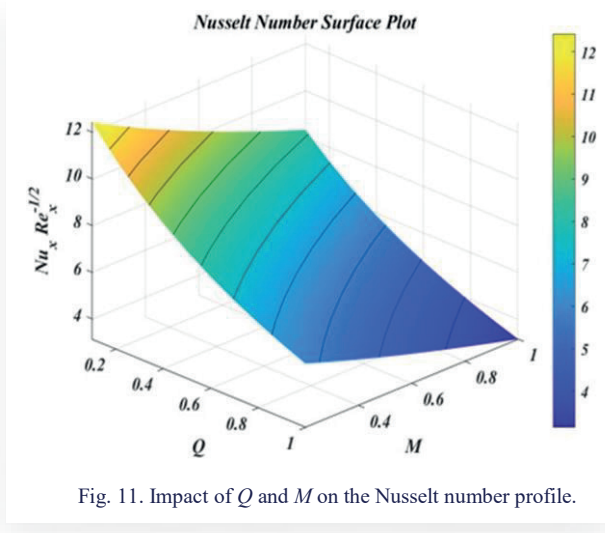


Fig. 11. Impact of Q and M on the Nusselt number profile.

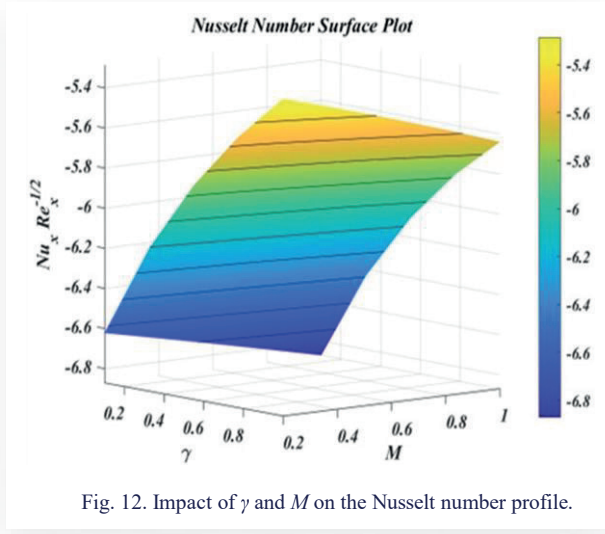


Fig. 12. Impact of γ and M on the Nusselt number profile.

fluid. Consequently, stronger magnetic fields contribute to a rise in the temperature profile.

Figure 7 shows that the curvature parameter increase leads to a noticeable rise in the fluid temperature. This behaviour arises due to the geometric effects associated with curved surfaces, where the boundary layer becomes thinner and convective heat loss is reduced. As a result, thermal energy accumulates more effectively near the surface. In addition, the presence of hybrid nanoparticles (Au-Ag) enhances heat conduction, further amplifying the temperature within the curved stenotic artery. Thus, curvature plays a critical role in regulating heat transfer characteristics in biomedical flows. Figure 8 shows a rise in the heat source parameter (Q), which leads to a noticeable enhancement in the temperature profile. This is because positive values of Q stand for the fluid's internal heat generation, which contributes additional thermal energy to the system. As a result, the thermal boundary layer becomes thicker, and the overall temperature distribution increases.

Figure 9 shows that the Eckert number (Ec) enhances viscous dissipation, converting more thermal energy from kinetic energy. This raises the fluid temperature, resulting in a thicker thermal boundary layer.

Figure 10 reveals that the skin friction coefficient decreases with increasing permeability and magnetic field strength. Higher porosity facilitates fluid seepage through the medium, lowering

wall shear. Meanwhile, the Lorentz force due to magnetic effects suppresses flow velocity near the surface, further reducing frictional resistance. From Fig. 11, it is seen that a rise in either magnetic parameter or the heat source parameter cuts the Nusselt number (Nu); added internal heating flattens the wall's temperature gradient, while the Lorentz force from a stronger

magnetic field slows the fluid. Both effects curb convective heat transfer. Figure 12 shows that as the radius of curvature increases, the Nusselt number also increases, indicating improved heat transfer near the artery wall; this happens because a smoother curvature allows better fluid flow. On the other hand,

increasing M lowers the Nusselt number, since the magnetic force slows down the fluid, lessening the heat it can transfer from the wall.

In Fig. 13, it has been observed that the streamline plot for $K = 2.0$ illustrates how the fluid flows through the porous artery.

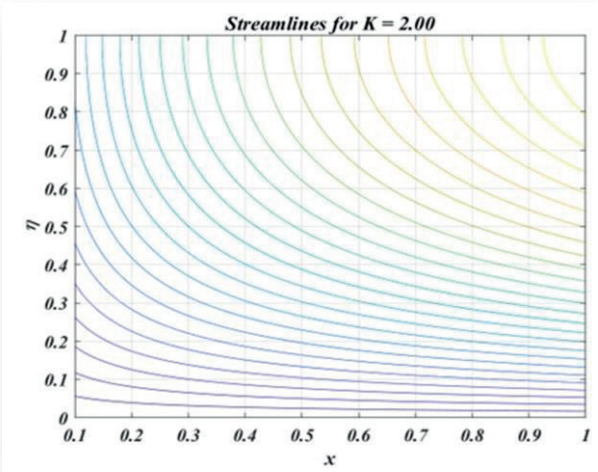


Fig. 13. Streamlines for $K = 2.0$.

The streamlines are denser near the wall, showing stronger velocity gradients in that region. Farther from the wall, the flow becomes more uniform. This behaviour reflects how moderate porosity influences the blood flow, allowing movement but still offering resistance. Figure 14 shows how the fluid flows under the influence of a magnetic field with $M = 1.0$; the streamlines

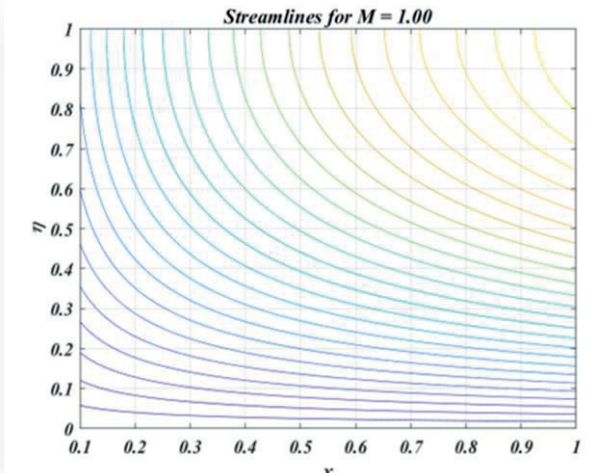


Fig. 14. Streamlines for $M = 1.0$.

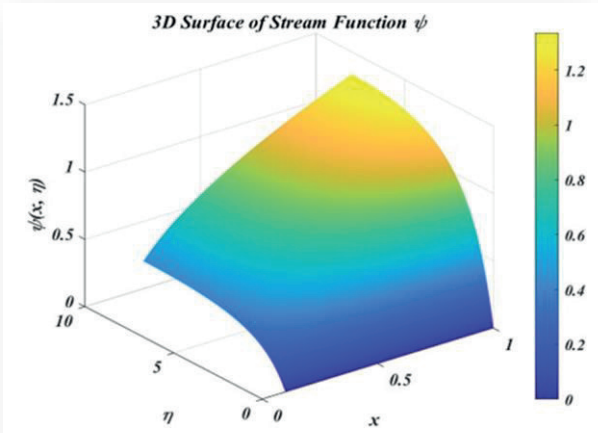


Fig. 15. 3D surface plot of stream function.

are tightly packed near the wall, indicating strong velocity gradients there. The magnetic field slows the flow by opposing motion, especially in the central region, leading to more stable and controlled stream behaviour across the artery. The 3D surface plot in Fig. 15 shows how the stream function changes with position, reflecting the strength and direction of blood flow. The flow starts slowly near the wall and upstream, then becomes stronger as it moves away from the wall and further downstream. This gives a clear picture of how the velocity field evolves through the stenotic artery.

Table 2 confirms the validity of the present numerical method by comparing it with earlier results from Waqas et al. [40]. The strong consensus between both sets of values demonstrates the accuracy of the current model and solution technique.

Table 2. Comparison of results for $C_f Re_r^{1/2}$ for various values of γ and ϕ_1 .

Sarwar and Hussein [58]			Present outcomes (bvp4c)
γ	ϕ_1	$C_f Re_r^{1/2}$	$C_f Re_r^{1/2}$
0.10	0.01	0.939968	0.939925
0.12	0.01	0.924794	0.924651
0.14	0.01	0.911311	0.911306
0.10	0.05	1.329552	1.329549

6. Conclusions

The numerical investigation of Casson-based hybrid nanofluid flow through a porous, magnetised stenotic artery using the bvp4c MATLAB solver has provided detailed insights into the flow and thermal behaviour of blood with embedded gold-silver nanoparticles. The model revealed critical dependencies of velocity, temperature, wall shear, and heat transfer on parameters such as the magnetic parameter, permeability and curvature parameter, contributing valuable knowledge towards biomedical fluid dynamics and therapeutic control in stenotic conditions:

- 1) A rise in magnetic field intensity leads to reduced blood velocity due to the opposing Lorentz force. Simultaneously, it raises the fluid temperature, as magnetic resistance contributes to internal heat build-up (Joule heating) and weakens the transport of heat by convection.
- 2) As the permeability of the artery increases, fluid velocity and wall shear stress decrease; this is attributed to higher flow resistance within the porous structure, which dissipates energy more effectively.
- 3) Higher curvature values result in increased velocity and temperature. This is due to geometric changes in the artery that affect both flow distribution and heat transfer behaviour, especially in narrowed (stenotic) regions.
- 4) Larger values of the Casson parameter increase flow velocity by reducing the fluid's yield stress. This change enables the fluid to behave more like a Newtonian fluid, thereby improving its ability to move through irregular arterial structures.
- 5) An increase in internal heat source strengthens the temperature field; this causes thickening of the thermal boundary layer and decreases the Nusselt number due to reduced temperature gradient.

- 6) Higher Eckert numbers increase the temperature of the nanofluid by converting kinetic energy into thermal energy. This enhances viscous dissipation and leads to greater heat accumulation in the fluid.

This work shows a comprehensive theoretical framework for analysing bio-magnetic Casson hybrid nanofluid blood flow through a stenosed artery with internal heat generation or absorption. The results demonstrate that:

1. Casson rheology significantly influences velocity and shear stress distribution, reinforcing the need for non-Newtonian modelling in stenotic arteries.
2. Magnetic fields reduce flow velocity but enhance temperature due to Joule heating, revealing both therapeutic potential and safety considerations.
3. Heat source/sink effects strongly modulate temperature, suggesting applications in hyperthermia and inflammatory heat regulation.
4. Hybrid nanoparticles, especially of cylindrical shape, improve thermal conductivity and heat transport but must be carefully controlled to avoid excessive thermal loads.
5. The combined effects of rheology, stenosis geometry, magnetic field, and nanoparticle-enhanced thermal conduction create complex hemodynamic behaviours relevant to cardiovascular pathology and advanced biomedical engineering.

Acknowledgements

We are grateful for the open access funding provided by Rastriya Uchchattar Shiksha Abhiyan (RUSA 2.0), Ministry of Human Resource Development, Department of Higher Education, Government of India, India.

The funding bodies had no role in the design of the study, collection, analysis, interpretation of data, or in writing the manuscript.

References

[1] Chakraborty, S. (2019). Electrokinetics with blood. *Electrophoresis*, 40(1), 180–189. doi: 10.1002/elps.201800353

[2] Vasu, B., Dubey, A., & Bég, O.A. (2020). Finite element analysis of non-Newtonian magneto-hemodynamic flow conveying nanoparticles through a stenosed coronary artery. *Heat Transfer – Asian Research*, 49(1), 33–66. doi: 10.1002/hjt.21598

[3] Mukhopadhyay, S. (2013). Effects of thermal radiation on Casson fluid flow and heat transfer over an unsteady stretching surface subjected to suction/blowing. *Chinese Physics B*, 22(11), 114702. doi: 10.1088/1674-1056/22/11/114702

[4] Jaffrullah, S., Wuriti, S., Ganugapati, R.G., & Talagadadevi, S.R. (2023). Numerical comparison of stagnation point Casson fluid stream over flat and cylindrical surfaces with Joule heating and chemical reaction impacts. *Frontiers in Heat and Mass Transfer*, 21(1), 407–426. doi: 10.32604/fhmt.2023.043305

[5] Akbar, N.S., Akram, J., Ijaz, S., Jahan, Z., Muhammad, T., Habib, M.B., & Farooq, M. (2025). High accuracy computational approach to study Casson hybrid nanofluid flow under induced magnetic field inside wavy walls configuration having peristaltic motion with entropy generation. *Dynamics of Atmospheres and Oceans*, 112, 101609. doi: 10.1016/j.dynatmoce.2025.101609

[6] Mehdi, I., Abbas, Z., & Hasnain, J. (2023). Entropy generation effects with thermal radiation on MHD Casson fluid flow in porous micro-annulus. *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering*, 239(1), 434–444. doi: 10.1177/09544089231189564

[7] Islam, M.R., Reza-E-Rabbi, S., Yousuf Ali, M., Rasel, M.M.H., & Ahmmed, S.F. (2023). Numerical simulation of mass and heat transport phenomena of hydromagnetic flow of Casson fluid with sinusoidal boundary conditions. *Engineering Reports*, 5(11), e12659. doi: 10.1002/eng2.12659

[8] Dubey, A., Vasu, B., Bég, O.A., Gorla, R.S.R., & Kadir, A. (2020). Computational fluid dynamic simulation of two-fluid non-Newtonian nano hemodynamics through a diseased artery with a stenosis and aneurysm. *Computer Methods in Biomechanics and Biomedical Engineering*, 23(8), 345–371. doi: 10.1080/10255842.2020.1729755

[9] Tripathi, J., Vasu, B., Bég, O.A., Mounika, B.R., & Gorla, R.S.R. (2022). Numerical simulation of the transport of nanoparticles as drug carriers in hydromagnetic blood flow through a diseased artery with vessel wall permeability and rheological effects. *Microvascular Research*, 139, 104241. doi: 10.1016/j.mvr.2021.104241

[10] Liu, B., & Tang, D. (2000). A numerical simulation of viscous flows in collapsible tubes with stenosis. *Applied Numerical Mathematics*, 32(1), 87–101. doi: 10.1016/s0168-9274(99)00015-x

[11] Young, F.D., & Tsai, F.Y. (1973). Flow characteristics in model of arterial stenosis – I. Steady flow. *Journal of Biomechanics*, 6(4), 395–410. doi: 10.1016/0021-9290(73)90099-7

[12] Pincombe, B., & Mazumdar, J.N. (1997). The effects of post-stenotic dilatations on the flow of a blood analogue through stenosed coronary arteries. *Mathematical and Computer Modelling*, 25(6), 57–70. doi: 10.1016/s0895-7177(97)00039-3

[13] Pincombe, B., Mazumdar, J., & Hamilton-Craig, I. (1999). Effects of multiple stenoses and post-stenotic dilatation on non-Newtonian blood flow in small arteries. *Medical & Biological Engineering & Computing*, 37, 595–599. doi: 10.1007/bf02513353

[14] Wong, K.K.L., Tu, J., Mazumdar, J., & Abbott, D. (2010). Modelling of blood flow resistance for an atherosclerotic artery with multiple stenosis and post-stenotic dilations. *ANZIAM Journal*, 51(EMAC2009), C66–C82. doi: 10.21914/anziamj.v51i0.2587

[15] Akbar, N.S., Maraj, E.N., Noor, N.F.M., & Habib, M.B. (2022). Exact solutions of an unsteady thermal conductive pressure-driven peristaltic transport with temperature-dependent nanofluid viscosity. *Case Studies in Thermal Engineering*, 35, 102124. doi: 10.1016/j.csite.2022.102124

[16] Bali, R., & Awasthi, U. (2011). Mathematical model of blood flow in small blood vessel in the presence of magnetic field. *Applied Mathematics*, 2(2), 264–270. doi: 10.4236/am.2011.22031

[17] Jain, M., Sharma, G.C., & Singh, R. (2010). Mathematical modeling of blood flow in a stenosed artery under MHD effect through porous medium. *International Journal of Engineering*, 23(3–4), 243–252.

[18] Tripathi, D. (2012). A mathematical model for blood flow through inclined arteries under the influence of inclined magnetic field. *Journal of Mechanics in Medicine and Biology*, 12(3), 1250033. doi: 10.1142/s0219519411004812

[19] Eldesoky, I.M. (2012). Slip effects on the unsteady MHD pulsatile blood flow through porous medium in an artery under the effect of body acceleration. *International Journal of Mathematics and Mathematical Sciences*, 2012, 860239. doi: 10.1155/2012/860239

[20] Khan, U., Shafiq, A., Zaib, A., Sherif, E.-S.M., & Baleanu, D. (2020). MHD radiative blood flow embracing gold particles via a

- slippery sheet through an erratic heat sink/source. *Mathematics*, 8(9), 1597. doi: 10.3390/math8091597
- [21] Nisar, K.S., Khan, U., Zaib, A., Khan, I., & Morsy, A. (2020). Magnetohydrodynamic radiative blood flow embracing nanoparticles. *The European Physical Journal Plus*, 135(10), 842. doi: 10.1140/epjp/s13360-020-00830-w
- [22] Akbar, N.S., Habib, M.B., Rafiq, M., Muhammad, T., & Alghamdi, M. (2024). Biological structural study of emerging shaped nanoparticles for the blood flow in diverging tapered stenosed arteries to see their application in drug delivery. *Scientific Reports*, 14, 1475. doi: 10.1038/s41598-024-51848-4
- [23] Gonzalez, C., Rosas-Hernandez, H., Ramirez-Lee, M.A., Salazar-García, S., & Ali, S.F. (2016). Role of silver nanoparticles (AgNPs) on the cardiovascular system. *Archives of Toxicology*, 90(3), 493–511. doi: 10.1007/s00204-014-1447-8
- [24] Talapko, J., Matijević, T., Juzbašić, M., Antolović-Požgain, A., & Škrlec, I. (2020). Antibacterial activity of silver and its application in dentistry, cardiology and dermatology. *Microorganisms*, 8(9), 1400. doi: 10.3390/microorganisms8091400
- [25] Hussain, A., Sarwar, L., Rehman, A., Akbar, S., Gamaoun, F., Copan, H.H., Almaliki, A.H., & Alqurashi, M.S. (2022). Heat transfer analysis and effects of (silver and gold) nanoparticles on blood flow inside arterial stenosis. *Applied Sciences*, 12(3), 1601. doi: 10.3390/app12031601
- [26] Hussain, A., Riaz, M.B., Dar, M.N.R., Kanwal, R., Sarwar, L., & Jhangeer, A. (2025). Computational study of effect of hybrid nanoparticles on hemodynamics and thermal transfer in ruptured arteries with pathological dilation. *Scientific Reports*, 14, 13400. doi: 10.1038/s41598-024-64356-2
- [27] Zaman, A., Ali, N., & Sajid, M. (2016). Slip effects on unsteady non-Newtonian blood flow through an inclined catheterized overlapping stenotic artery. *AIP Advances*, 6(1), 015118. doi: 10.1063/1.4941358
- [28] Eldesoky, I.M., Abdelsalam, S.I., El-Askary, W.A., & Ahmed, M.M. (2019). Concurrent development of thermal energy with magnetic field on a particle-fluid suspension through a porous conduit. *Bionanoscience*, 9, 186–202. doi: 10.1007/s12668-018-0585-5
- [29] Mishra, S.R., Hoque, M.M., Mohanty, B., & Anika, N.N. (2019). Heat transfer effect on MHD flow of a micropolar fluid through porous medium with uniform heat source and radiation. *Nonlinear Engineering*, 8(1), 65–73. doi: 10.1515/nleng-2017-0126
- [30] Rehman, K.U., Malik, M.Y., Al-Mdallal, Q.M., & Zahri, M. (2019). On both magnetized and non-magnetized dual stratified medium via streamlines topologies: A generalized formulation. *Scientific Reports*, 9, 6306. doi: 10.1038/s41598-019-42726-5
- [31] Jena, S., Mishra, S.R., & Pattnaik, P.K. (2020). Development in the heat transfer properties of nanofluid due to the interaction of inclined magnetic field and non-uniform heat source. *Journal of Nanofluids*, 9(3), 143–151. doi: 10.1166/jon.2020.1749
- [32] Pattnaik, P.K., Moapatra, D.K., & Mishra, S.R. (2021). Influence of velocity slip on the MHD flow of a micropolar fluid over a stretching surface. In *Recent Trends in Applied Mathematics: Select Proceedings of AMSE 2019* (pp. 307–321). doi: 10.1007/978-981-15-9817-3_21
- [33] Priyadharsini, M., Gururaj, D.M., Ismail, M., & Sheremet, M. (2023). Numerical and sensitivity study on the heat transfer due to bioconvection on unsteady radiative MHD blood flow over a permeable artery with chemical reaction effects. *International Communications in Heat and Mass Transfer*, 147, 106981. doi: 10.1016/j.icheatmasstransfer.2023.10698
- [34] Ali, B., Jubair, S., Fathima, D., Akhter, A., Rafique, K., & Mahmood, Z. (2023). MHD flow of nanofluid over moving slender needle with nanoparticles aggregation and viscous dissipation effects. *Scientific Progress*, 106(2), 00368504231176151. doi: 10.1177/00368504231176151
- [35] Ali, B., Liu, S., Liu, H.J., & Siddiqui, M.I.H. (2025). Magnetohydrodynamics tangent hyperbolic nanofluid flow across a vertical stretching surface using Levenberg-Marquardt backpropagation artificial neural networks. *Numerical Heat Transfer, Part A: Applications*, 86(20), 7116–7138. doi: 10.1080/10407782.2024.2348121
- [36] Ali, B., Jubair, S., Mahmood, Z., Koka, N.A., & Gani, A.H. (2024). Analysis of unsteady MHD fluid flow across two parallel discs with uniform fluctuation subject to modified Hall and activation energy. *International Journal of Modern Physics B*, 38(31), 2450424. doi: 10.1142/s0217979224504241
- [37] Ali, B., Liu, S., & Liu, H. (2024). An application of artificial neural network toward the mathematical modeling of MHD tangent hyperbolic nanofluid across a vertical stretching surface. *Journal of Porous Media*, 27(10), 49–72. doi: 10.1615/jpormedia.2024051939
- [38] Jalali, S., Jalali, S., & Barati, E. (2024). Pulsatile blood flow simulations of magnetic drug targeting (MDT) for drug particles dispersion through stenosis asymmetric and symmetric vessels. *Journal of Magnetism and Magnetic Materials*, 590, 171649. doi: 10.1016/j.jmmm.2023.171649
- [39] Akbar, N.S., Maraj, E.N., Ijaz, S., Muhammad, T., Hussain, M.F., Habib, M.B., & Farooq, M. (2025). Artificial neural network analysis for entropy-driven electroosmotic blood cell transport with drug-loaded ZnO nanoparticles in a stenosed tapered artery. *International Communications in Heat and Mass Transfer*, 169(D), 109786. doi: 10.1016/j.icheatmasstransfer.2025.109786
- [40] Waqas, H., Farooq, U., Liu, D., Alghamdi, M., Noreen, S., & Muhammad, T. (2022). Numerical investigation of nanofluid flow with gold and silver nanoparticles injected inside a stenotic artery. *Materials & Design*, 223, 111130. doi: 10.1016/j.matdes.2022.111130
- [41] Ramasekhar, B., Jakeer, S., Reddy, S.R.R., Alkarni, S., & Shah, S.A. (2024). Biomedical importance of Casson nanofluid flow with silver and Fe₂O₃ nanoparticles delivered into a stenotic artery: Numerical study. *AIMS Mathematics*, 9(8), 23142–23157. doi: 10.3934/math.20241125
- [42] Tanveer, A., Ul-Haq, S., Ashraf, M.B., Ashraf, M.U., & Nawaz, A.R. (2024). Optimisation of MHD flow within trapezoidal cavity containing hybrid nanofluid by artificial neural network. *International Journal of Computational Methods*, 34(6), 2455–2479. doi: 10.1108/hff-01-2024-0058
- [43] Islam, T., Gama, S., & Martins Afonso, M. (2025). Artificial neural network and response surface methodology-driven optimization of Cu–Al₂O₃/water hybrid nanofluid flow in a wavy enclosure with inclined periodic magnetohydrodynamic effects. *Mathematics*, 13(1), 78. doi: 10.3390/math13010078
- [44] Zaman, A., Ali, N., & Khan, A.A. (2020). Computational biomedical simulations of hybrid nanoparticles on unsteady blood hemodynamics in a stenotic artery. *Mathematics and Computers in Simulation*, 169, 117–132. doi: 10.1016/j.matcom.2019.09.010
- [45] Majeed, A.H., Ahmed Zubair Jan, A.Z., Alamri, A.M., AlQahtani, S.A., Ali, M.R., & Hendy, A.S. (2024). Recent developments in the Darcy-Forchheimer model of magnetized tetra hybrid nanofluid activation energy/joule heating in a stenotic artery. *Case Studies in Thermal Engineering*, 59, 104346. doi: 10.1016/j.csite.2024.104346
- [46] Tang, T.-Q., Rooman, M., Vrinceanu, N., Shah, Z., & Alshehri, A. (2022). Blood flow of Au-nanofluid using Sisko model in stenotic artery with porous walls and viscous dissipation effect. *Micromachines*, 13(8), 1303. doi: 10.3390/mi13081303
- [47] Jain, R.K. (1987). Transport of molecules in the tumor interstitium: A review. *Cancer Research*, 47(12), 3039–3051. https://aacrjournals.org/cancerres/article/47/12/3039/491490/Transport-of-Molecules-in-the-Tumor-Interstitium-A
- [48] Stylianopoulos, T., & Jain, R.K. (2013). Combining two strategies to improve perfusion and drug delivery in solid tumors. *Proceedings of the National Academy of Sciences*, 110(46), 18632–18637. doi: 10.1073/pnas.1318415110
- [49] Sajid, M., Wasim Jamshed, Mohamed R. Eid, Gilder Altamirano, C., Aslam, F., Alanzi, A.M., & Abd-Elmonem, A. (2023). Magnetized Cross tetra hybrid nanofluid passed a stenosed artery with nonuniform heat source (sink) and thermal radiation: Novel tetra hybrid Tiwari and Das nanofluid model. *Journal of Magnetism and Magnetic Materials*, 569, 170443. doi: 10.1016/j.jmmm.2023.170443
- [50] Shah, S.Z.H., Ayub, A., Khan, U., Darvesh, A., Sherif, E.S., & Pop, I. (2024). Thermal transport exploration of ternary hybrid nanofluid flow in a non-Newtonian model with homogeneous heterogeneous chemical reactions induced by vertical cylinder. *Advances in Mechanical Engineering*, 16(5), 168781322412522294. doi: 10.1177/16878132241252229
- [51] Liu, K.C., & Chen, T.M. (2021). Comparative study of heat transfer and thermal damage assessment models for hyperthermia treatment. *Journal of Thermal Biology*, 98, 102907. doi: 10.1016/j.jtherbio.2021.102907
- [52] Liang, A., Munier, S., & Danish, S. (2020). NIMG-68. Mathematical modeling of thermal damage estimate volumes in magnetic resonance-guided laser interstitial thermal therapy. *Neuro-Oncology*, 22(Suppl. 2), ii163. doi: 10.1093/neuonc/noaa215.681
- [53] Sarwar, L., Hussain, A., Fernandez-Gamiz, U., Akbar, S., Rehman, A., & Sherif, E.M. (2022). Thermal enhancement and numerical solution of blood nanofluid flow through stenotic artery. *Scientific Reports*, 12(1), 17419. doi: 10.1038/s41598-022-20267-8
- [54] Ramasekhar, G., Shaik, J., Reddisekhar Reddy, S.R., Divya, A., Jawad, M., & Ali Yousif, B.A. (2026). Numerical investigation of Casson fluid flow performance of blood containing gold and Fe₃O₄ nanofluid injected into a stenotic artery. *Numerical Heat Transfer, Part A: Applications*, 87(1), 2372465. doi: 10.1080/10407782.2024.2372465
- [55] Ramasekhar, G., Divya, A., Selvi, P.D., Narayanagari, R., Sekhar Reddy, R.C., Jakeer, S., Reddisekhar Reddy, S.R., Suneetha, S., Shah, N.A. & Ahmed, S.F. (2025). Exploring the role of Casson hybrid nanofluid flow through a stretching cylinder with a significant impact of heat source/sink: A computational analysis. *AIP Advances*, 15(3), 035120. doi: 10.1063/5.0254849
- [56] Kumar, M.D., Upadhya, M.S., Yook, S.J., Shah, N.A., & Raju, C.S.K. (2025). Spectral quasi-linearization and response optimization on magnetohydrodynamic flow via stenosed artery with hybrid and ternary solid nanoparticles: Support vector machine learning. *Nanotechnology Reviews*, 14(1), 20250188. doi: 10.1515/ntrev-2025-0188
- [57] Shahzad, F., Jamshed, W., Aslam, F., Bashir, R., Tag El Din, E.S.M., Khalifa, H.A.E.W., & Alanzi, A.M. (2022). MHD pulsatile flow of blood-based silver and gold nanoparticles between two concentric cylinders. *Symmetry*, 14(11), 2254. doi: 10.3390/sym14112254
- [58] Sarwar, L., Hussain, A. (2021). Flow characteristics of Au-blood nanofluid in stenotic artery. *International Communications in Heat and Mass Transfer*, 127, 105486. doi: 10.1016/j.icheatmasstransfer.2021.105486



Co-published by
Institute of Fluid-Flow Machinery
Polish Academy of Sciences
Committee on Thermodynamics and Combustion
Polish Academy of Sciences

Copyright©2026 by the Authors under licence CC BY-NC-ND 4.0

<http://www.imp.gda.pl/archives-of-thermodynamics/>



Production and characterisation of pyrolysis oil from discarded medical plastic (syringe) waste with common rail direct injection engine assessment

Rajkumar Thamizhvel*, Natarajan Vaithianathan, Narayanan Sethuraman,
Subramanian Vinodraj, Lenin Ambetharasu, Palani Surendar

Indo French Educational Trust (IFET) College of Engineering, Gangarampalayam, Villupuram 605108, Tamil Nadu, India

*Corresponding author email: rthamizhvel@gmail.com

Received: 13.09.2025; revised: 11.02.2026; accepted: 25.02.2026

Abstract

This study explores the potential of converting syringe plastic waste into an alternative fuel through pyrolysis. Since syringes are primarily made of polypropylene, they were thermally decomposed in a batch reactor, producing a maximum pyrolysis oil yield of 76% at 400–500°C. The produced syringe plastic oil (SPO) was characterised for key fuel properties and blended with diesel at 20%, 30%, and 40% proportions. The performance of the engine and the emission characteristics of these blends were tested experimentally on a common rail direct injection diesel engine. Of the fuels tested, the D80SPO20 mixture proved to be as efficient as diesel with the brake thermal efficiency of 26.9% and brake specific fuel consumption of 0.27 kg/kWh at full load. This mixture was also found to result in a significant reduction in carbon monoxide, hydrocarbons and smoke emission with a slight increment in NO_x relative to diesel. On the whole, the findings show that low-percentage SPO30, especially D80SPO20, provide a favourable combination between engine efficiency and emission reduction, proving the potential of the syringe plastic pyrolysis oil as a sustainable alternative of diesel fuel in part.

Keywords: Pyrolysis; Medical plastic; CRDI engine; Performance; Emission

Vol. 47(2026), No. 2, 201–210; doi: 10.24425/ather.2026.158687

Cite this manuscript as: Thamizhvel, R., Vaithianathan, N., Sethuraman, N., Vinodraj, S., Ambetharasu, L., & Surendar, P. (2026). Production and characterisation of pyrolysis oil from discarded medical plastic (syringe) waste with common rail direct injection engine assessment. *Archives of Thermodynamics*, 47(2), 201–210.

1. Introduction

The high consumption of plastics around the world has caused a high level of plastic waste accumulation, which has been rather threatening to not only the environment but also human health [1]. Of these wastes, medical plastics, with single-use syringes being one exceptional and increasing proportion, are a major cause of concern. Polymers (polypropylene and polyethylene) are the most commonly used materials in the production of syringes because they are mechanically robust, have chemical resistance, and are inexpensive, which are required for use in healthcare practices [2]. However, these properties cause poor biodegradation, which causes long-term pollution of the environment.

The control of medical plastic waste has continued to become increasingly difficult with the growing healthcare services and emergency medical interventions, which are highly dependent on disposable plastic products. The traditional methods of disposal, such as incineration and landfill, cannot be considered sustainable methods anymore. Incineration can also release toxic chemicals like dioxins and furans, and landfilling can also consume unused land resources and contaminate the soil and groundwater [3]. In addition, medical plastic waste may pose serious biological risks when mis-handled or improperly disposed of. Consequently, there is an urgent requirement for an alternative mode of waste management that would limit environmental degradation but allow the reuse of useful resources in medical plastic waste [4].

Nomenclature

CO – carbon monoxide
HC – hydrocarbon
NO_x – oxides of nitrogen
SO_x – oxides of sulphur

Abbreviations and Acronyms

BSFC – brake specific fuel consumption
BTE – brake thermal efficiency
CRDI – common rail direct injection
PM – particulate matter
SPO – syringe plastic oil

A lot of focus has been on bio-derived oxygenated fuels, bio-derived biodiesel, and products of thermochemical conversion, as they are renewable and have the potential to lower the environmental effects. Some of the studies highlight the nature of biodiesel and oxygenated fuels' performance and combustion in CI engines. The experiment reported in [5] showed that bio-diesel mixtures have a higher combustion efficiency because they contain built-in oxygen, which results in increased brake thermal efficiency under increased loads. Nevertheless, bio-diesel had a lower calorific value, which meant that it consumed more brake-specific fuel than diesel.

The same tendencies could be found in [6], where optimisation of the injection parameters was used to overcome the performance penalties related to the use of biodiesel. These are the findings that point to the significance of engine calibration to reap the full advantages of oxygenated fuels.

Research concerns on engine emissions indicate uniform emissions of carbon monoxide (CO), unburned hydrocarbons (HC) and particulate matter (PM) that are reduced with the use of biodiesel or bio-derived production. The paper [7] has obtained significant reductions in smoke opacity and PM emissions, which were attributed to better oxidation in combustion. Nevertheless, the rise in nitrogen oxides (NO_x) emissions is still an issue that is here to stay, as research findings have established in [7]. Further combustion plans and post-treatment devices are thus required to deal with this trade-off. It has also discussed the environmental and health effects of exhaust emissions in detail. Another extensive study [8] assessed the oxidative potential and toxicity of the biodiesel-powered engine particulate emissions. The research inference was that biodiesel particulates have a lower potential to cause adverse health effects due to their lower oxidative stress potential than diesel PM. This is in line with the larger environmental analyses that focus on the use of biodiesel in reducing air pollution in urban environments. Renewable fuel production pathways are critically evaluated in terms of their sustainability and life cycle. The paper [9] compared life-cycle greenhouse gas emissions of biodiesel and concluded that the feedstock choice is a very important part of the total sustainability. It was demonstrated that waste-derived and non-edible feedstocks have a huge impact in lowering carbon footprints as compared to the traditional sources of edible oils. To further support this, paper [10] emphasised the potential of using agricultural waste and oils from production processes to generate biofuels, thereby reducing environmental costs and resource loss.

Recent studies talk about the advances in technologies of biomass conversion. A study [11] examined catalytic and thermochemical conversion pathways in the production of biofuels and found that the quality of fuels produced was better and conversion efficiencies were increased. These findings were also confirmed by [12], which showed that optimising the process can

help to increase the stability of the fuel and the engine's compatibility. Most recently, the paper [11] provided promising findings on an integrated bioenergy system with a focus on techno-economic feasibility and scalability.

Thermochemical conversion of plastic wastes by the pyrolysis process is one prospective method. One of these techniques is pyrolysis, which involves decomposition of waste plastics at high temperatures in the absence of oxygen [12]. Through this process, long plastic chains are reduced into smaller parts, which are useful materials such as pyrolysis oil, syngas and char. Of these, pyrolysis oil has attracted much attention because of its prospects of being used as a liquid fuel. It has properties similar to those of diesel, which provide the chance to completely or partially replace standard petrol engines.

Current advancements in fuel technology involve the use of alternative forms of fuel in contemporary diesel motors [13]. The common rail direct injection (CRDI) engine is one of them. Fuel atomisation, accurate injection timing and injected pressures are also some of the factors that make common rail direct injection engines achieve better combustion efficiency and lower exhaust emissions [14]. With these merits, CRDI engines are specifically better applied in the experimental studies of using alternative fuels, even waste-based fuels, like plastic pyrolysis oil. Past experimental research on the utilisation of plastic-derived pyrolysis oil as a diesel engine fuel has recorded promising findings [15]. Studies have also proved that the pyrolysis oil can be mixed with standard diesel in any quantity and used successfully in a diesel engine without significant operational problems. The blended fuels have shown satisfactory performance qualities and, in some instances, enhanced thermal efficiency of the brakes. In addition, it has been found that there will be decreases in exhaust pollutants like unburned hydrocarbons and carbon monoxide at specific ratios of blends. Nevertheless, under certain operating conditions, nitrogen oxide emissions are inclined to go up, which shows that additional optimisation and emission control measures are needed.

Although significant advances have been achieved in the recent past regarding the use of plastic waste to obtain energy via its conversion into energy, little attention has been given to the actual application of medical plastic waste and specifically discarded syringes in the pyrolysis oil production [17]. This is a significant gap in the study since, besides being easily accessible, syringe plastics are also high-quality polymers that can undergo thermal degradation [18]. The advantages of converting this form of waste into fuel are also twofold since it not only solves this pressing concern of medical waste disposal but also helps in the energy recovery process. The paper [19] examines the possibility of using the pyrolysis oil by using the old medical syringe plastic as an alternative fuel in a common rail direct injection diesel engine.

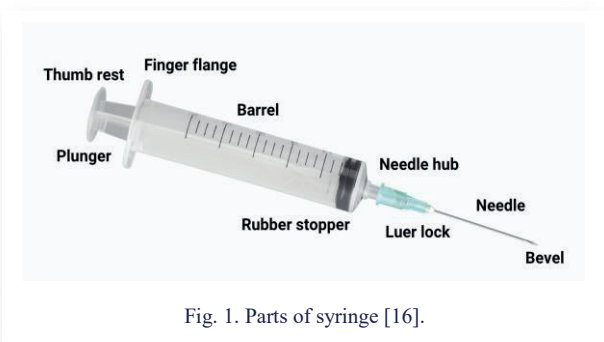


Fig. 1. Parts of syringe [16].

In the work [20], the collection and preparation of medical plastic waste is followed by the pyrolysis oil production, in which controlled thermal decomposition takes place, and the characterisation of the received fuel in greater detail. The properties of important fuels such as viscosity, density, calorific value and flash point are discussed and compared with those of conventional diesel to determine their suitability to be applied in engines [21]. Using a CRDI engine, the engine performance and emission tests with different proportions of pyrolysis oil and diesel have been carried out [22]. The performance parameters that are measured to understand how the engine works include brake thermal efficiency, brake specific fuel consumption, and the in-cylinder exhaust gas temperature and pressure [23]. Moreover, exhaust emissions, such as hydrocarbons, carbon monoxide, nitrogen oxides and smoke opacity, are quantified to evaluate the environmental effect of the alternative fuel blends [24].

This paper gives useful information on the combustion behaviour and performance characteristics, as well as the emission profile of syringe plastic-derived pyrolysis oil in a modern CRDI diesel engine. The study reveals that waste-based fuels are practically applicable by using an innovative method of medical plastic waste treatment in order to generate energy. The research results provide evidence-based information that proves the feasibility of this fuel and can be used to guide future policy, technological advancement and manufacturing to curb plastic waste and to improve energy sustainability. The novelty of the work is the use of the discarded plastic medical syringe as feedstock and a systematic assessment of syringe plastic oil (SPO) in a CRDI engine under controlled experimental conditions. Also, the study underscores the correlation of pyrolysis temperature, properties of fuels, and performance and emissions of the engine, which gives a holistic picture of SPO as an alternative fuel.

2. Materials and methods

2.1. Syringe waste

Syringe plastic waste was selected due to its abundance, consistent polymer composition and environmental concerns. Made mainly of polypropylene and polyethylene, it is ideal for pyrolysis to produce fuel-rich oil. Utilising this waste helps reduce biomedical pollution, avoids harmful incineration by-products, and supports sustainable energy recovery from non-biodegradable plastics, in line with eco-friendly waste management prac-

tices [25]. The part of the syringe shown in Fig. 1 and Table 1 represents the materials of syringe components.

Table 1. Materials and properties of syringe components.

Syringe component	Material used
Barrel	Polypropylene (PP)
Plunger	Polypropylene (PP)
Plunger seal (rubber stopper)	Synthetic rubber (e.g., polyisoprene or butyl rubber)
Finger flange	Polypropylene (PP)
Syringe tip (luer lock/slip)	Polypropylene (PP)
Needle hub	Polypropylene (PP) or polyethylene (PE)
Needle	Stainless steel (Grade 304 or 316)
Needle bevel (tip)	Stainless steel
Needle cap	Polyethylene (PE)

2.2. Pyrolysis process

Pyrolysis is a process where syringe plastic waste is heated in the absence of oxygen, breaking it down into liquid oil, gases, and char. The photographic view of the experimental setup is shown in Fig. 2. In a batch reactor, oil is produced by heating waste to temperatures ranging from 300°C to 500°C, which starts the breakdown of polymers. This process efficiently turns non-biodegradable plastic into useful fuel [26], providing a long-term solution for energy recovery and biomedical waste management [27]. The temperature in the reactor first increases at a rate of 8°C/min and then decreases to 3°C/min towards the end of pyrolysis.



Fig. 2. Pyrolysis setup.

3. Testing and analysis

3.1. Proximate analysis

The proximate analysis (Table 2) of medical syringe waste reveals a high volatile matter content of 99.86%, indicating excellent potential for pyrolysis-based fuel recovery. As the moisture content is 0.00%, there is no water interference in thermal conversion [28]. Ash content is minimal at 0.05%, while fixed car-

Table 2. Proximate analysis of medical syringe waste.

Proximate analysis	wt.%
Volatile matter	99.86
Fixed carbon	0.09
Ash	0.05
Moisture	0.00

bon is 0.09%, confirming the material’s clean and energy-rich nature, ideal for thermal degradation [29].

3.2. Ultimate analysis

The ultimate analysis (Table 3) shows that medical syringe waste contains a high carbon content of 84.71% and hydrogen at 13.81%, indicating strong potential as a hydrocarbon fuel source. The absence of nitrogen (0.00%) and very low sulphur content (0.07%) suggest minimal emission of harmful gases, making it suitable for clean pyrolysis-based energy recovery applications [30].

Table 3. Ultimate analysis of medical syringe waste.

Elements analysis	wt.%
Carbon (C)	84.71
Hydrogen (H)	13.81
Nitrogen (N)	0.00
Sulphur (S)	0.07

3.3. Thermogravimetry analysis

A thermogravimetric analysis (TGA) was performed on medical syringe plastic waste to assess its thermal decomposition characteristics, as shown in Fig. 3. The sample showed a steady weight loss when heated at a rate of 10°C per minute from room temperature to 800°C in a nitrogen atmosphere. The breakdown started around 250°C, with major weight loss occurring between

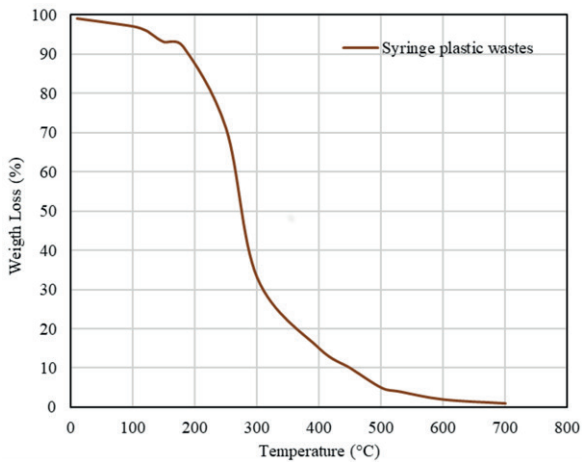


Fig. 3. Thermogravimetric analysis curve.

300°C and 500°C, suggesting the main thermal decomposition of plastics like polypropylene and polyethylene. The maximum rate of degradation occurred around 400°C, with a steep decline in weight. By 600°C, over 90% of the sample mass had decomposed. A stable residual mass of approximately 5% remained beyond 700°C, likely representing char and inorganic fillers. The TGA curve confirms that syringe waste undergoes major thermal degradation within a specific range, making it suitable for pyrolysis processes targeting liquid fuel recovery while minimising solid residue [31]. This thermal behaviour guides the selection of optimal pyrolysis conditions.

3.4. Pyrolysis oil yield rate

As seen in Fig. 4, more oil is obtained when using a syringe plastic waste at higher temperatures. At 300°C, the liquid production is more or less 71.2%, the gas and char are 20% and 8.8%, respectively. With a rise in temperature to 400°C, the liquid production increases to 76.4%, and the gas and char fractions decrease to 17% and 6.6%, respectively. Further increase in temperature to 500°C yields the same oil yield of 76.5%, as well as gas and char fractions of 17% and 6.5%, respectively. This shows that the optimum temperature at which the highest output of oil is achieved is between 400–500°C. The oil yield levels off beyond 400°C, giving little indication of further benefits due to high temperature. The selected temperature profile was informed by the preliminary experiments and literature on plastic pyrolysis to maximise the oil production and reduce a lot of excessive degradation by the thermal process.

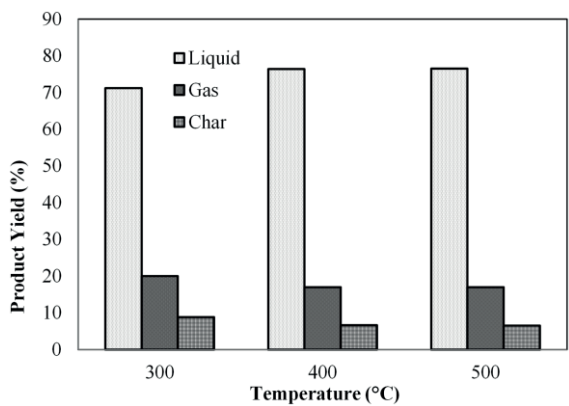


Fig. 4. Pyrolytic product yield rate.

3.5. Gas chromatography – mass spectrometry

To identify the chemical composition of the pyrolysis oil of discarded plastic waste derived from medical syringes, gas chromatography-mass spectrometry (GC-MS) analysis (Fig. 5) was conducted. The chromatogram indicates a wide distribution of hydrocarbon compounds in a retention time span of 0–40 min. The existence of light hydrocarbons, predominantly short-chain alkanes and alkenes (C5–C7) in low-intensity peaks at low retention times (0.05 min (5.10%), 0.15 min (7.12%), and 1.32 min (7.12%)) at early retention times (< 2 min) shows that early polymer chain scission occurred during pyrolysis. The promi-

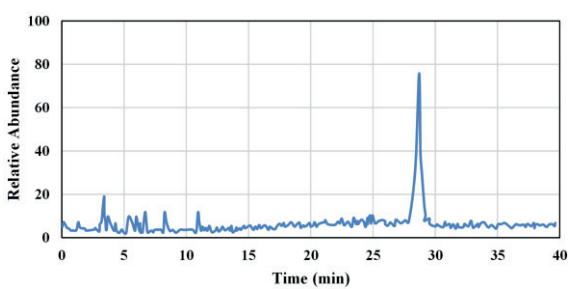


Fig. 5. Gas chromatography-mass spectrometry analysis of syringe plastic pyrolytic oil.

nent peaks at 3.30 min (14.06%), 3.41 min (18.98%), and 3.70 min (9.69%) in the low-to-mid retention time range (3–5min) represent the presence of the light olefinic and branched hydrocarbons (octane, nonane and their unsaturated derivatives). The medium-chain hydrocarbons (decane, undecane, dodecane, etc.) have a boiling point in the gasoline and light diesel ranges (6–10 minutes’ retention time) and yield moderate-intensity peaks at 6–12 min (6.69 min (11.71%), 8.26 min (11.71%), and 10.94 min (11.71%)). Higher aliphatic hydrocarbons such as tridecane, tetradecane, pentadecane and cycloalkanes are present, as indicated by the presence of peaks with relative abundances of between 5% and 9% in the mid-to-high retention time region (15 min to 23 min), such as tridecane at 17.83 min (6.72%), tetradecane at 18.43 min (7.09%) and pentadecane at 21.10 min. A large mass cluster is found around the range of 28 min to 29 min with a major peak at 28.71 min indicating the highest relative abundance of 75.77% and the next few masses of 28.46 min (37.75%), 28.81 min (39.02%), 28.91 min (30.83%), which are related to long-chain hydrocarbons, including eicosane, heneicosane and other polypropylene derivatives aliphatic compounds. The heavier hydrocarbons (C20–C30) above 30 min correspond to heavier hydrocarbons that add to the boiling range and viscosity of the oil. In general, the GC-MS data prove that the pyrolysis oil is mainly composed of the alkanes and alkenes of light to heavy fractions, which evidences that this oil can be used as an alternative fuel after proper blending.

3.6. Properties of syringe plastic oil

Syringe plastic oil (SPO) (Fig. 6), produced through pyrolysis, shows great potential as an alternative source due to its favourable properties (Table 4). It has a higher energy content of 39.5 MJ/kg, with a density of 0.89 g/cm³ and viscosity of 1.92 mm²/s. Its flash point is 39°C and fire point is 48°C. These features make SPO comparable to conventional diesel, indicating its suitability for energy applications [32].

4. Experimentation

4.1. Common rail direct injection engine specification

The engine to be considered in this study is a common rail direct injection (CRDI) engine (Fig. 7), because it has accurate control of the fuel injection, better combustion efficiency, and reduced



Fig. 6. Syringe plastic oil from pyrolysis process.

Table 4. Physical properties of pyrolysis oil from syringe plastics.

Property	Syringe plastic oil	Standard method
Lower heating value (MJ/kg)	39.5	ASTM D240
Density (g/cm ³ at 15°C)	0.89	ASTM D4052
Kinematic viscosity (mm ² /s at 40°C)	1.92	ASTM D445
Flash point (°C)	39	ASTM D93
Fire point (°C)	48	ASTM D92
Cetane number	46	ASTM D613
Lubricity (μm, HFRR*)	450	ASTM D6079
Cold filter plugging point (°C)	–4	ASTM D6371

*HFRR – high frequency reciprocating rig.

emissions. The following conditions were used to operate the engine: injection pressure – 2000–2300 bar; injection timing – 230° before top dead centre (BTDC); injection strategy – multiple injections per cycle; compression ratio – 17.5:1. To measure the combustion and performance characteristics of alternative fuels, such as syringe plastic pyrolysis oil and mixtures with diesel, these settings were selected. High injection pressure, optimised timing and multiple injection strategy enable the engine



Fig. 7. Common rail direct injection engine setup.

Table 5. Engine specifications.

Parameter	Specification
Engine type	4-cylinder, 4-stroke, double overhead camshaft (DOHC), water-cooled
Engine capacity	1248 cc
Bore × stroke	69.6 mm × 82 mm
Compression ratio	17.5:1
Rated power	74 BHP at 4000 rpm
Maximum torque	190 Nm at 2000 rpm
Fuel injection system type	Common rail direct injection (CRDI)
Injection pressure	1400–1600 bar
Injection timing	23° BTDC (typical for CRDI diesel)
Rated engine speed	4000 rpm
Operating speed range	1000–4000 rpm

to achieve effective fuel atomisation and full combustion, making the engine ideal in the study of effects of pyrolysis oil on efficiency, emissions and combustion behaviour. The engine specifications are provided in Table 5.

4.2. Uncertainty analysis

The accuracy of experimental measurements of the data is highly reliant on an appropriate uncertainty analysis, which is necessary to determine the accuracy of the instrumentation and measurements. An appropriate choice of instruments, proper operating conditions, proper configuration of experiment parameters, and proper carrying out of calibration, observation, and testing procedures can help in minimising measurement errors [33].

The root sum square (RSS) method was used to determine the total experimental uncertainty in the current experiment by examining each of the uncertainties related to brake power (BP) and specific fuel consumption (BSFC), brake thermal efficiency (BTE) and exhaust emissions (CO, HC, and NO_x). It is determined by the uncertainties of the pressure transducer (δP), the crank angle encoder ($\delta \theta$), the exhaust emission measurements of nitrogen oxides (δNO_x), hydrocarbons (δHC) and carbon monoxide (δCO), the fuel consumption measurement (δFC) and the manometer (δM):

Total uncertainty of the experiment = $\sqrt{(\delta P)^2 + (\delta \theta)^2 + (\delta \text{NO}_x)^2 + (\delta \text{HC})^2 + (\delta \text{CO})^2 + (\delta \text{FC})^2 + (\delta M)^2}$

Substituting the relevant numbers into the formula using consistent units, the following is obtained:

Total percentage uncertainty = $\sqrt{(1.2)^2 + (1.5)^2 + (2.0)^2 + (1.8)^2 + (1.0)^2 + (1.5)^2 + (1.8)^2}$

According to this analysis, the overall uncertainty of the experiment readings was determined as being within the range of about ±4%.

5. Results and discussion

The generation of bio-oil from medical plastic waste through pyrolysis is an innovative and environmentally significant process with the potential to address waste management and energy needs for the CRDI engine.

5.1. Performance characteristics

5.1.1. Brake thermal efficiency

Figure 8 shows how the brake thermal efficiency (BTE) changes with brake power in a CRDI engine with diesel-based and SPO-diesel blends. In all test fuels, the BTE increases with the load, which can be explained by the increased combustion efficiency at the higher brake power as a result of reduced relative heat losses and an increased use of the provided fuel energy. Diesel has the highest BTE of 27.2, with D80SPO20 having a nearly similar value (26.9%) and D70SPO30 (25.7%). The marginal decrease in BTE with an increase in the proportion of SPO is mainly due to the lower calorific value and increased viscosity of the syringe plastic oil relative to diesel that can negatively influence the atomisation of the fuel and the air-fuel mixture, mainly at reduced loads. Nevertheless, the fact that the D80SPO20 blend is nearly diesel highlights that at lower SPO concentrations, the increased volatility and oxygenated character of the blend fosters increased combustion during CRDI injection conditions [34]. This is because the negative effects of SPO properties are further reduced by the high injection pressure and the enhanced spray properties of the CRDI system, allowing efficient combustion at higher loads.

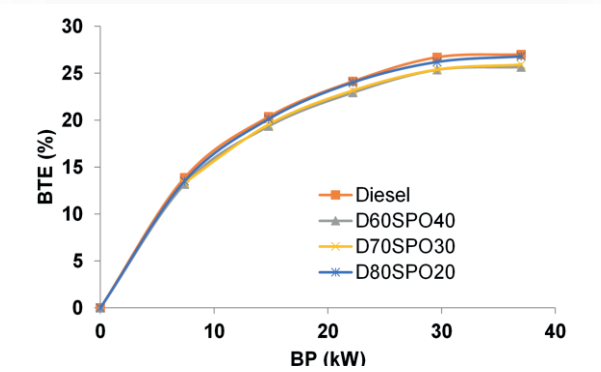


Fig. 8. Brake thermal efficiency vs. load for diesel and SPO-diesel blends.

These trends are in line with the previous studies of plastic pyrolysis oil-diesel blends in CRDI engines [35], where even blends with as much as 20% pyrolysis oil had a BTE value similar to diesel due to phasing up of combustion and low ignition lag at higher loads.

5.1.2. Brake specific fuel consumption

Figure 9 illustrates the change in the brake specific fuel consumption (BSFC) with the brake power of diesel and SPO-diesel blends in a CRDI engine. In all test fuels, we find that BSFC decreases significantly with increasing load, and this is mainly because of the better combustion efficiency and the low relative friction and heat loss at higher brake power. Diesel at low loads (10 kW) has a BSFC of approximately 0.512 kg/kWh, whereas higher values are observed in the SPO-diesel blends, with D60SPO40 and D70SPO30 actually being around 0.55 kg/kWh and 0.60 kg/kWh, respectively.

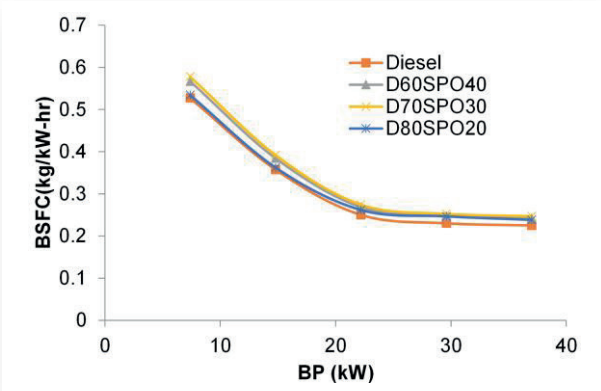


Fig. 9. Brake specific fuel consumption vs. load for diesel and SPO-diesel blends.

It can be explained by the higher BSFC of the blends with a high percentage of SPO at low loads, as the calorific value and viscosity of the syringe plastic oil negatively influence the process of spray atomisation, leading to incomplete combustion of the air-fuel mixture and increased fuel consumption. The in-cylinder temperature and pressure increase with the load and improve fuel evaporation and combustion performance of all blends [36]. Accordingly, when the loads increase (approaching 35 kW), the values of the BSFC of all fuels become almost equal to approximately 0.27 kg/kWh. D80SPO20, of the SPO blends, is also always found to have lower BSFC at all loads with an almost similar operation to that of the diesel.

5.2. Emission characteristics

5.2.1. Carbon monoxide

Figure 10 shows how carbon monoxide (CO) emissions depend on the brake power of the CRDI engine for diesel and SPO-diesel blends. The CO emissions of diesel are comparatively higher at low load states (approx. 5 kW) of approximately 0.10% vol., and this may be associated with incomplete combustion because of lower in-cylinder temperatures and the lack of oxidation of CO. Conversely, the D80SPO20 blend has the least CO emission of about 0.06% (vol.), stating that combustion behaviour is better even at light loads. This is reduced with the improved spray atomisation and oxidation properties that are generated by the fuel properties of the blended SPO at the high-pressure injection of the CRDI engine.

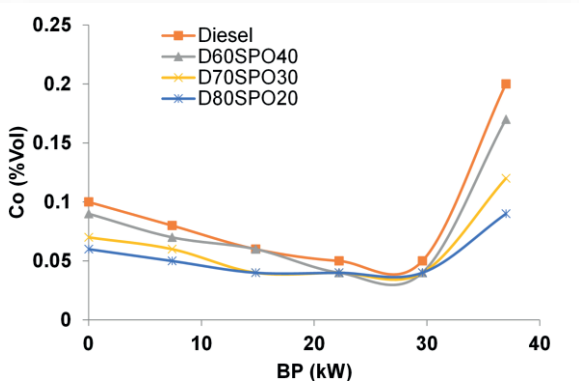


Fig. 10. Carbon monoxide emissions vs. load for diesel and SPO-diesel blends.

Increasing the brake power, the CO emissions of all test fuels decrease to about 25–30 kW as a result of high combustion temperatures, better air-fuel mixing, and a long effective oxidation period, which favour complete combustion [37]. But, over this range of loads, there is an abrupt rise in CO emissions of all the fuels. This increase with higher loads is primarily a result of a higher rate of fuel injection, locally rich mixtures of fuel and air, low oxygen supply, and the shortness of residence time of oxidation reactions [38]. Diesel has the highest CO emission at 35 kW of approximately 0.21% (vol.), and D80SPO20 has a relatively lower amount of about 0.13% (vol.). This lower CO emission of D80SPO20 was consistent with the trend of the study previously reported on plastic oil in CRDI engines because of its improved combustion efficiency.

5.2.2. Unburnt hydrocarbons

Figure 11 shows the change in the emission of hydrocarbon (HC) with the brake power of diesel and SPO-diesel mixtures used in a CRDI engine. At low loads (approx. 5 kW), diesel has a higher value of HC emissions of about 46 ppm (parts per million) and the lowest value of about 34 ppm at D80SPO20. This is mainly because of the incomplete combustion at the low in-cylinder temperatures, such that incomplete oxidation results in the persistence of unconsidered hydrocarbons. The comparatively reduced HC emissions of SPO blends, especially

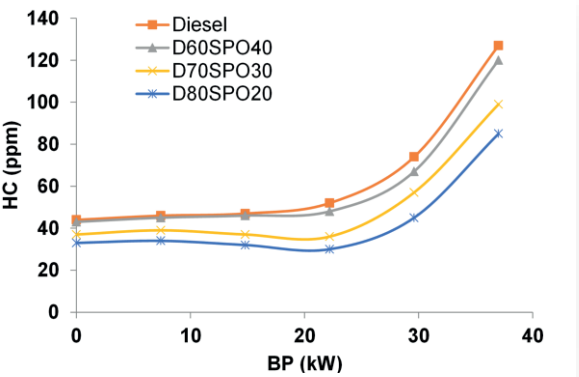


Fig. 11. Hydrocarbon emissions vs. load for diesel and SPO-diesel blends.

D80SPO20, are indicative of better combustion properties with better spray atomisation, as well as better mixing under high-pressure injection during CRDI.

The emissions of HC increase gradually with the brake power, and increase sharply after 30 kW, regardless of the test fuel. At higher loads, higher rates of fuel injection also result in locally rich fuel-air mixtures and re-shortened residence time available to allow complete oxidation and, thus, increasing HC formation. Diesel has the highest level of HC of approximately 130 ppm, whereas D60SPO40, D70SPO30 and D80SPO20 have lower values of about 120 ppm, 105 ppm and 95 ppm, respectively. The relatively smaller HC emissions of D80SPO20 indicate that an intermediate level of SPO as a blend would support further complete combustion even during the high load status. Other past research has reported similar HC emission trends in CRDI engines using plastic pyrolysis oil-diesel blends [28], where blends with up to 20% pyrolysis oil had lower unburned hydrocarbons as a result of enhanced combustion efficiency and a favourable fuel-air interaction [29].

5.2.3. Nitrogen oxides

Figure 12 shows that the emissions of nitrogen oxides (NO_x) depend on the brake power of diesel and SPO-diesel mixtures in a CRDI engine. The emissions of NO_x go proportionate to load in all test fuels because of the increased in-cylinder temperatures and the greater ability to form thermal NO_x with increased combustion intensity. At low loads, the level of NO_x is relatively low in all fuels, indicating low peak temperatures and low oxidation rates. The combustion is also conducive to NO_x formation, as the increase in brake power causes an increase in the quantity of fuel injection and efficiency of the combustion, which results in the enhancement of the flame temperature. The highest NO_x emission of around 850 ppm is generated by the D60SPO40 blend, followed by the emissions of D70SPO30 and D80SPO20 being around 750 ppm and 700 ppm, respectively, and the lowest emission level, at 640 ppm, recorded for diesels. This is because the increased NO_x emissions of SPO blends are due to the enhanced combustion properties like reduced ignition delay and increased rates of heat release, which elevate peak temperatures in the cylinder. Moreover, the availability of lighter hydrocarbon fractions and oxygenated compounds in syringe plastic oil in-

tensifies oxidation reactions, which further contributes to the formation of NO_x .

Despite the fact that SPO-diesel blends show better brake thermal efficiency and lower CO and HC emissions compared to diesel, the trade-off is, however, exhibited through higher emissions of NO_x [39]. D80SPO20 is the most desirable of the blends investigated, and it has significantly lower NO_x than higher SPO-percentage blends without compromising the advantages of the improved combustion [40].

5.2.4. Smoke emission

Figure 13 demonstrates the change in smoke emissions with brake power of diesel and SPO-diesel combinations in a CRDI engine. There is a steady increase in the smoke opacities with increasing load in all test fuels, primarily due to increased volumes of fuel injection, localised rich fuel-air blends and shorter time of oxidation at high brake power. Under low load levels, the smoke emissions are relatively low because of the adequate oxygen supply, as well as improved combustion conditions. With increased load, the increase in in-cylinder pressure and fuel supply causes incomplete combustion and increased quantity of soot formation, which causes more smoke. Diesel has the highest smoke emission at full load (35 kW), which is approximated to be about 72 HSU (Hartridge smoke units); D60SPO40, D70SPO30 and D80SPO20 have lower values of about 58 HSU. The relatively greater emission of smoke by diesel is linked with less oxidation of soot particles by fuel-rich conditions at high loads.

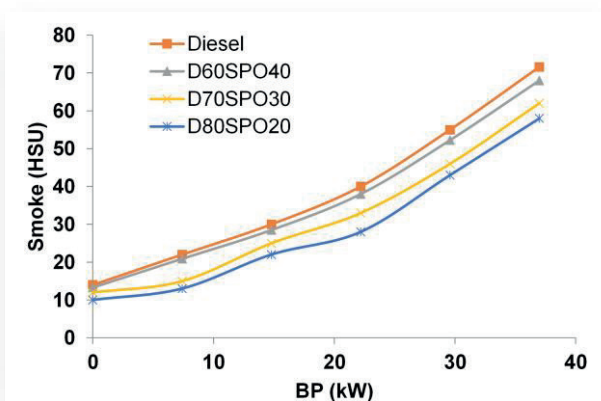


Fig. 13. Smoke opacity vs. load.

The lower levels of smoke emissions recorded with SPO-diesel blends, especially D80SPO20, can be explained by the fact that there were superior combustion properties during CRDI engine injection, which included better spray atomisation and better oxidation of the soot precursors. Moreover, the occurrence of lighter fractions of hydrocarbons in syringe plastic oil helps to promote more complete combustion, and thus, inhibits the development of particles [41]. D80SPO20 was selected as the best soot oxidation blend, showing the lowest smoke emissions at all loads, and, therefore, the best behaviour [42].

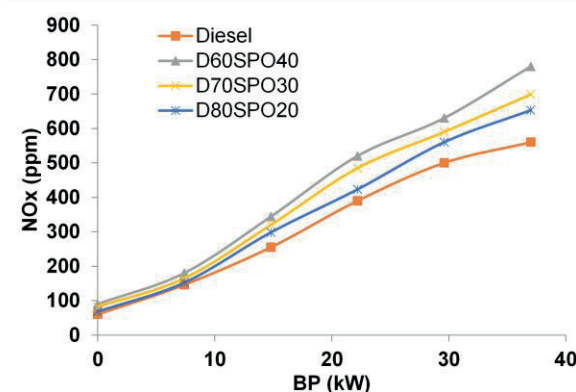


Fig. 12. Nitrogen oxides emission vs. load for diesel and SPO-diesel blends.

6. Conclusions

This study investigated the viability of syringe plastic oil (SPO) as an alternative fuel for a common rail direct injection (CRDI) diesel engine by characterising the properties of fuel, pyrolysing it, and assessing its performance and emissions. According to the results, the following conclusions are made:

- 1) Syringe plastic waste was found to contain a very high proportion of volatile matter (99.86%), which proved that it is very suitable for pyrolysis conversion. During pyrolysis at 400–500°C, the highest percentage of liquid oil produced was up to 76%, implying the efficient thermal breakdown of the substance within this temperature range.
- 2) The SPO that was obtained had a lower heating value of 39.5 MJ/kg, similar to diesel; its density (0.89 g/cm³) and kinematic viscosity (1.92 mm²/s) were within the tolerable blending ranges, and it was possible to operate the engine with it without alteration.
- 3) A test on engine thermal efficiency (BTE) showed that brake thermal efficiency increased with load, across all fuels. However, the D80SPO20 mixture produced BTE values at the level of 0.89% of those of diesel at the full load, indicating its ability to burn under the altered fuel characteristics through CRDI engine injection.
- 4) Brake specific fuel consumption decreased with load for all fuels, with D80SPO20 consistently showing lower BSFC than higher SPO-percentage blends, confirming improved fuel utilisation at moderate SPO proportions.
- 5) Emission measurements indicated that D80SPO20 cut down CO, HC and smoke emissions by an average of 35–45% as compared to diesel under heavy load conditions, which demonstrated improved oxidation and suppression of soot. Even though NO_x emissions at increased SPO fractions increased due to increased combustion temperatures, the D80SPO20 blend constrained the increase in NO_x compared with higher SPO concentrations.

Overall, the findings show that a 20% SPO blend offers the best balance between performance improvement and emissions. It is observed that the D80SPO20 blend is both technically feasible and eco-friendly as an alternative fuel in CRDI diesel engines, which justifies the successful use of medical plastic waste in the recovery of energy.

References

- [1] Ibn-Mohammed, T., Mustapha, K.B., Godsell, J., Adamu, Z., Babatunde, K.A., Akintade, D.D., Acquaye, A., Fujii, H., Ndiaye, M.M., Yamoah, F.A., & Koh, S.C.L. (2021). A critical analysis of the impacts of COVID-19 on the global economy and ecosystems and opportunities for circular economy strategies. *Resources Conservation and Recycling*, 164, 105169. doi: 10.1016/j.resconrec.2020.105169
- [2] Ahmed, M.M., Pali, H.S., & Khan, M.M. (2024). Allometry of nanoparticles in diesel-biodiesel blends for CI engine performance, combustion and emissions. *Archives of Thermodynamics*, 45(1), 99–108. doi: 10.24425/ather.2024.150442
- [3] Apeviyeneku, H., Ampofo, J., & Amoako, J.A. (2024). Recycling potential of medical plastic waste. *Discover Environment*, 2, 69. doi: 10.1007/s44274-024-00109-2

- [4] Senthil, R., Thamizhvel, R., Ashok, M., & Sudagar, S. (2026). Experimental investigation in CI engine using automobile plastics waste as bio oil (pyrolytic oil). In *Lecture Notes in Mechanical Engineering* (pp. 461–474). Springer, Singapore. doi: 10.1007/978-981-95-2239-2_40
- [5] Arefin, M.A., Nabi, M.N., Akram, M.W., Islam, M.T., & Chowdhury, M.W. (2020). A review on liquefied natural gas as fuels for dual fuel engines: Opportunities, challenges and responses. *Energies*, 13(22), 6127. doi: 10.3390/en13226127
- [6] Chowdhury, M.W., Nabi, M.N., Arefin, M.A., Rashid, F., Islam, M.T., Gudimetla, P., & Mueen, S.M. (2022). Recycling slaughterhouse wastes into potential energy and hydrogen sources: An approach for the future sustainable energy. *Bioresource Technology Reports*, 19, 101133. doi: 10.1016/j.biteb.2022.101133
- [7] Hassan, T., Rahman, M.M., Rahman, M.A., & Nabi, M.N. (2022). Opportunities and challenges for the application of biodiesel as automotive fuel in the 21st century. *Biofuels Bioproducts and Biorefining*, 16(5), 1353–1387. doi: 10.1002/bbb.2375
- [8] Hedayat, F., Stevanovic, S., Milic, A., Miljevic, B., Nabi, M.N., Zare, A., Bottle, S.E., Brown, R.J., & Ristovski, Z.D. (2016). Influence of oxygen content of the certain types of biodiesels on particulate oxidative potential. *Science of the Total Environment*, 545–546, 381–388. doi: 10.1016/j.scitotenv.2015.12.036
- [9] Hossain, F.M., Nabi, M. N., Rahman, M.M., Bari, S., Van, T.C., Rahman, S.M.A., Rainey, T.J., Bodisco, T.A., Suara, K., Ristovski, Z., & Brown, R.J. (2019). Experimental investigation of diesel engine performance, combustion and emissions using a novel series of diocetyl phthalate (DOP) biofuels derived from microalgae. *Energies*, 12(10), 1964. doi: 10.3390/en12101964
- [10] Hossain, F.M., Nabi, M.N., Rainey, T.J., Bodisco, T., Bayley, T., Randall, D., Ristovski, Z., & Brown, R.J. (2019). Novel biofuels derived from waste tyres and their effects on reducing oxides of nitrogen and particulate matter emissions. *Journal of Cleaner Production*, 242, 118463. doi: 10.1016/j.jclepro.2019.118463
- [11] Hossain, F.M., Nabi, M.N., Rainey, T.J., Bodisco, T., Rahman, M.M., Suara, K., Rahman, S.M.A., Van, T.C., Ristovski, Z., & Brown, R.J. (2017). Investigation of microalgae HTL fuel effects on diesel engine performance and exhaust emissions using surrogate fuels. *Energy Conversion and Management*, 152, 186–200. doi: 10.1016/j.enconman.2017.09.016
- [12] Islam, M.R., Nabi, M.N., & Islam, M.N. (2003). The fuel properties of pyrolytic oils derived from carbonaceous solid wastes in Bangladesh. *Jurnal Teknologi*, 38(A), 75–80. doi: 10.11113/jt.v38.484
- [13] Kannan, D., Nabi, M.N., & Hustad, J.E. (2009). Influence of ethanol blend addition on compression ignition engine performance and emissions operated with diesel and Jatropa methyl ester. *Powertrains, Fuels and Lubricants Meeting*, June 15, Florence, Italy, SAE Technical Papers. doi: 10.4271/2009-01-1808
- [14] Mitchell, B.J., Zare, A., Bodisco, T.A., Nabi, M.N., Hossain, F.M., Ristovski, Z.D., & Brown, R.J. (2017). Engine blow-by with oxygenated fuels: A comparative study into cold and hot start operation. *Energy*, 140(1), 612–624. doi: 10.1016/j.energy.2017.08.115
- [15] Verma, A., Budiya, L., Sanjay, M.R., & Siengchin, S. (2019). Processing and characterization analysis of pyrolyzed oil rubber (from waste tires)-epoxy polymer blend composite for lightweight structures and coatings applications. *Polymer Engineering and Science*, 59(10), 2041–2051. doi: 10.1002/pen.25204
- [16] Landry, J. (2026). *Parts of a syringe: Diagram and components*. Respiratory Therapy Zone. <https://www.respiratorytherapyzone.com/syringe-parts/> [accessed 10 Jan. 2026].
- [17] Singh, T.S., Rajak, U., Dasore, A., Muthukumar, M., & Verma, T.N. (2021). Performance and ecological parameters of a diesel engine fueled with diesel and plastic pyrolyzed oil (PPO) at vari-

- able working parameters. *Environmental Technology & Innovation*, 22, 101491. doi: 10.1016/j.eti.2021.101491
- [18] Singh, A.K., Pali, H.S., & Khan, M.M. (2024). Experimental investigations of CI engine performance using ternary blends of n-butanol/biodiesel/diesel and n-octanol/biodiesel/diesel. *Archives of Thermodynamics*, 45(1), 109–118. doi: 10.24425/ather.2024.150443
- [19] Senthil, R., Thamizhvel, R., Sudagar, S., & Sathiyaraj, P. (2026). A comparative analysis of bio-oil production from plastic and biomass with aid of nano particles through pyrolysis process. In *Lecture Notes in Mechanical Engineering* (pp. 423–435). Springer, Singapore. doi: 10.1007/978-981-95-2239-2_37
- [20] Dubba, S.K., Pisini, R., Kelli, D., Bammidi, R., Vanam, J.P., & Turali, N. (2025). Experimental investigation of the performance and emission characteristics of cotton seed biodiesel/diesel blends with titanium dioxide nanoparticles. *Archives of Thermodynamics*, 46(3), 151–161. doi: 10.24425/ather.2025.156587
- [21] Kumar, D., Kumar, N., & Chaudhary, R. (2024). Use of butanol, pentanol and diesel in a compression ignition engine: A review. *Archives of Thermodynamics*, 45(4), 27–35. doi: 10.24425/ather.2024.151994
- [22] Kaczmarczyk, T.Z. (2024). Experimental research of a pumping engine in a micro-ORC system with a low-boiling medium. *Archives of Thermodynamics*, 45(4), 125–140. doi: 10.24425/ather.2024.152002
- [23] Rai, I., & Kumar, R. (2025). Heat transfer analysis of a longitudinal fin attached to a beta-type Stirling engine. *Archives of Thermodynamics*, 46(3), 51–60. doi: 10.24425/ather.2025.156578
- [24] Kumar, A., Kumar, S., Verma, V.K., Singh, S., Shukla, R., Jain, A., Bist, A.S., Vishnoi, P., Kumar, V., & Bhandari, P., (2025). Experimental analysis of ethanol-diesel blends stabilised with Jatropa methyl ester: Engine Performance and emission characteristics. *Archives of Thermodynamics*, 46(3), 229–236. doi: 10.24425/ather.2025.156593
- [25] Banoth, K., Banoth, M., & Mukuloth, S. (2025). Investigating the impact of TiO₂ nanoparticles on waste plastic pyrolysis oil and DEE diesel blends for diesel engine performance and emission optimisation. *Archives of Thermodynamics*, 46(3), 237–245. doi: 10.24425/ather.2025.156594
- [26] Subramani, S., Kumar, P.D., Govindasamy, R., & Iyemperumal, M.S. (2025). Comparative analysis of cylinder-to-cylinder variation of parameters of an automotive CRDI engine. *Archives of Thermodynamics*, 46(4), 105–115. doi: 10.24425/ather.2025.156841
- [27] Senthil, R., Thamizhvel, R., Sudagar, S., & Ahamadu, B.A. (2025). Production of high-grade fuels from biomass and waste plastics and its influence in a CI engine. *Polymer Bulletin*, 82, 6095–6114. doi: 10.1007/s00289-025-05740-4
- [28] Ramalingam, S., Rajkumar, T., Subramanian, S., & Palani, S. (2024). Investigation of combustion, emission, and performance parameters of a natural antioxidant additives using hydrogen and biodiesel as dual fuel in CI engine operation. *International Journal of Hydrogen Energy*, 110, 44–54. doi: 10.1016/j.ijhydene.2025.01.425
- [29] Thamizhvel, R., Sethuraman, N., Sakthivel, M., & Prabhakaran R. (2020). Experimental investigation of diesel engine by using paper cup waste as the producer gas with help of down-draft gasifier. In *IOP Conference Series: Materials Science and Engineering*. 988, 012015. doi: 10.1088/1757-899X/988/1/012015
- [30] Senthil, R., Thamizhvel, R., & Sudagar, S. (2024). A comparative analysis of pyrolytic oil from medical waste and its suitability as fuel in a compression ignition engine. *Energy Sources Part A: Recovery Utilization and Environmental Effects*, 46(1), 2040–2058. doi: 10.1080/15567036.2024.2302369
- [31] Subramanian, S., Thamizhvel, R., Ashokan, A., Rajendran, S., Venkatesan, E.P., Alwetaishi, M., Hussain, F., & Keçebaş, A. (2026). Integrated NiO₂-catalyzed reforming of sesame stalk for hydrogen-enriched producer gas and dual-fuel CI engine performance evaluation. *International Journal of Hydrogen Energy*, 209, 153333. doi: 10.1016/j.ijhydene.2025.153333.
- [32] Thamizhvel, R., Naveen Raj, S., & Krishna Raj, S. (2025). Production of oil from plastic waste through thermal degradation process. *Turkish Journal of Engineering*, 9(3): 409–416. doi: 10.31127/tuje.1535115.
- [33] Thamizhvel, R., Manikandan, A., Praveen Kumar, S., & Sivasankaran, S. (2023). Potential use of cathode ray tube as an abrasive particle in abrasive jet machine. In *2023 International Conference on Energy, Materials and Communication Engineering (ICEMCE)* (pp. 1–6). Madurai, India. doi: 10.1109/ICEMCE57940.2023.10434107
- [34] Thamizhvel, R., Suryavarman, K., Velmurugan, V., & Sethuraman, N. (2021). Comparative study of gasification and pyrolysis derived from coconut shell on the performance and emission of CI engine. *Materials Today: Proceedings*, 47(4), 978–983. doi: 10.1016/j.matpr.2021.05.350
- [35] Sethuraman, N.K., Surya Varman, K., Venkatakrishnan, R., & Thamizhvel, R. (2021). An experimental investigation of crude glycerol into useful products by using IC engine in dual fuel mode. *Materials Today: Proceedings*, 44(5), 3914–3918. doi: 10.1016/j.matpr.2021.02.160
- [36] Kadupu, R.V., Priyadharsini, S., Ananthi, K., Thamizhvel, R., Sudagar, S., & Silambrasan, R. (2024). Assessing the thermal insulation properties of thermoplastic bricks for energy-efficient building solutions. *Matéria (Rio de Janeiro)*, 29(4). doi: 10.1590/1517-7076-rmat-2024-0583
- [37] Ramalingam, S., Thamizhvel, R., Sudagar, S., & Silambrasan, R. (2023). Production of third generation bio-fuel through thermal cracking process by utilizing Covid-19 plastic wastes. *Materials Today: Proceedings*, 72(3), 1618–1623. doi: 10.1016/j.matpr.2022.09.430
- [38] Sethuraman, N., Karthikeyan, D., Vinodraj, S., Thamizhvel, R., & Kumarasamy, G. (2023). Performance and emission characteristics on CI engine by using waste cooking oil and waste transformer oil as substitute fuel with coconut shell as a catalyst. *Materials Today: Proceedings*, 72(4): 2469–2475. doi: 10.1016/j.matpr.2022.09.457
- [39] Thamizhvel, R., Molly Irine, G.S., Vaithianathan, N., Ganesh, M. (2023). Evaluating the performance and emission characteristics of jackfruit seed as bio-oil in CI engine. *Materials Today: Proceedings*, 2214–2853. doi: 10.1016/j.matpr.2023.08.100
- [40] Narayanan, S., Vinodraj, S., Thamizhvel, R., & Vaithianathan, N. (2025). Machine learning optimisation of CI engine characteristics with WSA algorithm and Prosopis juliflora biofuel. *Archives of Thermodynamics*, 46(4), 117–125. doi: 10.24425/ather.2025.156842
- [41] Madhuvanesan, D.M., & Babu, J.M. (2024). Impact of nano cerium oxide addition with pyrolysis waste plastic oil on combustion, performance and emission characteristics of CRDI diesel engine. *Engineering Research Express*, 6, 045551. doi: 10.1088/2631-8695/ad923e
- [42] Khandal, S.V., Khan, T.M.Y., Badruddin, I.A., Kamangar, S., Saleel, C.A., Baig, M.A.A., & Ahmed, N.J.S. (2021). Performance and emission analysis of multicylinder common rail direct injection diesel engine powered with blends of tyre pyrolysis oil-ethanol diesel. *Journal of Engineering Research*, 10(1B), 167–192. doi: 10.36909/jer.8583