International Conference on Fluid and Thermal Engineering
(ICFTE-2026)

19th – 21st January, 2026

BOOK OF ABSTRACT

Organized by

Department of

Mechanical Engineering

and

Clean Energy Research Lab,

BITS Pilani, Pilani Campus,

Rajasthan, India.

EDITOR :

Prof. Srinibas Tripathy

Prof. Yogesh Singh

Prof. Suvanjan Bhattacharyya

International Conference on Fluid and Thermal Engineering

(ICFTE) 2026

19 -21 January 2026

Book of Abstracts

Organized by

Department of Mechanical Engineering

and

Clean Energy Research Lab

Birla Institute of Technology and Science (BITS) Pilani Pilani, Campus, Pilani, Rajasthan-333031, India.

First impression: Publication 2026

International Conference on Fluid and Thermal Engineering (ICFTE 2026)

Disclaimer

The authors are solely responsible for the contents of the papers compiled in this volume. The publishers or editors do not take any responsibility for the same in any manner. Errors, if any, are purely unintentional, and readers are requested to communicate such errors to the editor or publishers to avoid discrepancies in the future.

Published at:

Department of Mechanical Engineering

Birla Institute of Technology and Science (BITS) Pilani, Pilani-333021, India

Typeset & Design by:

Book of Abstracts Committee ICFTE 2026

MESSAGE FROM CONFERENCE CHAIR

Prof. Srikanta Routroy, Ph.D.,
Professor & Head, Mechanical Engineering Department

It is my great pleasure to welcome you to the International Conference on Fluid and Thermal Engineering (ICFTE) 2026, a global forum dedicated to advancing research, innovation, and collaboration in fluid mechanics, heat transfer, and thermal engineering. ICFTE 2026 is envisioned as a common platform that brings together leading researchers, academicians, industry professionals, and young scientists from around the world to exchange knowledge and discuss emerging challenges and future directions in these vital engineering domains.

Fluid and thermal engineering play a central role in addressing critical global challenges such as energy efficiency, clean and renewable energy technologies, sustainable thermal systems, advanced manufacturing, and biomedical applications. In this context, ICFTE 2026 promotes in-depth technical discussions across a wide range of topics, including advanced heat transfer, multiphase and turbulent flows, hydrogen and alternative fuels, thermal management of electronics and batteries, micro- and nano-fluidics, biomechanics, and state-of-the-art computational and experimental techniques.

The conference program has been carefully designed to offer a stimulating and enriching experience through keynote lectures by distinguished international experts, invited talks, technical paper presentations, and interactive sessions. Special emphasis is placed on interdisciplinary research and strengthening academia–industry collaboration to translate fundamental research into practical engineering solutions.

ICFTE 2026 also provides an excellent opportunity for young researchers and doctoral students to showcase their work, receive expert feedback, and build lasting professional networks. I sincerely thank the organizing committee, advisory board members, reviewers, sponsors, and participants for their invaluable support, and I look forward to welcoming you to ICFTE 2026.

MESSAGE FROM ORGANISING SECRETARY

Prof. Suvanjan Bhattacharyya, Ph.D.,
Mechanical Engineering Department

It is my great pleasure to welcome you to the International Conference on Fluid and Thermal Engineering (ICFTE) 2026, a global forum dedicated to advancing research, innovation, and collaboration in fluid mechanics, heat transfer, and thermal engineering. ICFTE 2026 is envisioned as a common platform that brings together leading researchers, academicians, industry professionals, and young scientists from around the world to exchange knowledge and discuss emerging challenges and future directions in these vital engineering domains.

Fluid and thermal engineering play a central role in addressing critical global challenges such as energy efficiency, clean and renewable energy technologies, sustainable thermal systems, advanced manufacturing, and biomedical applications. In this context, ICFTE 2026 promotes in-depth technical discussions across a wide range of topics, including advanced heat transfer, multiphase and turbulent flows, hydrogen and alternative fuels, thermal management of electronics and batteries, micro- and nano-fluidics, biomechanics, and state-of-the-art computational and experimental techniques.

The conference program has been carefully designed to offer a stimulating and enriching experience through keynote lectures by distinguished international experts, invited talks, technical paper presentations, and interactive sessions. Special emphasis is placed on interdisciplinary research and strengthening academia–industry collaboration to translate fundamental research into practical engineering solutions.

ICFTE 2026 also provides an excellent opportunity for young researchers and doctoral students to showcase their work, receive expert feedback, and build lasting professional networks. I sincerely thank the organizing committee, advisory board members, reviewers, sponsors, and participants for their invaluable support, and I look forward to welcoming you to ICFTE 2026.

CONTENTS

ABOUT THE CONFERENCE

As one of India's premier institutions of higher learning, BITS Pilani has consistently championed excellence in engineering, science, and innovation. Continuing this tradition, the Institute is proud to host the International Conference on Fluid and Thermal Engineering (ICFTE), a distinguished global forum dedicated to advancing knowledge in fluid mechanics and thermal sciences and engineering. This conference will bring together leading researchers, scientists, engineers, and scholars from around the world to present pioneering research, exchange innovative ideas, and explore emerging trends shaping the future of fluid and thermal engineering.

ICFTE provides a unique opportunity to foster meaningful academic and industry collaborations, offering participants a platform to engage in stimulating discussions, share best practices, and address critical challenges in the field. With its emphasis on both fundamental research and industrial applications, the conference promises to be a vibrant exchange of knowledge and innovation.

ABOUT THE BITS PILANI

Birla Institute of Technology & Science, BITS Pilani, is an all-India Institute for higher education. Founded with strong technical collaboration with MIT (USA) under the Ford Foundation Grant, BITS Pilani has evolved into India's leading institute of higher education, distinguished by its illustrious legacy, modern campuses, and exceptional placement records. The institute's commitment to excellence, adherence to merit, transparency, innovation, and enterprise has been the hallmark of its journey. BITS Pilani has got campuses in Dubai, Hyderabad, Goa and Mumbai in addition to Pilani. The past six decades has been exceptional for BITS Pilani, marked by groundbreaking achievements in research, teaching and fostering entrepreneurship. Key achievements and milestones include: 1. Institute of Eminence, 2. BITS BioCyTiH Foundation, 3. Center for Research Excellence in Semiconductor Technologies (CREST), 4. BITS-Technology Enabling Centres (TEC), 5. Center for Research Excellence in National Security (CRENS), 6. BITS GATI (Gender Advancement for Transforming Institutions), 7. SATHI Project Award.

ABOUT THE MECHANICAL ENGINEERING DEPARTMENT

The Mechanical Engineering Department at BITS Pilani is spread across its four campuses of Pilani, Dubai, Goa and Hyderabad. It offers undergraduate, postgraduate, and PhD programs, focusing on a combination of theoretical learning and hands-on experience. Established as early as 1946, the department is recognised for its research in fields such as Design, Manufacturing, Thermo-fluids, and Engineering Management. Our students explore specialized areas such as Robotics, Mechatronics, Energy Management, 3D Printing, Battery Technology, etc. and are trained in software tools like AutoCAD, CATIA, COMSOL Multiphysics, MATLAB, Fusion 360, etc. Faculty members are deeply involved in cutting-edge research, addressing both fundamental and applied problems, with strong links to industry, academia and research. With a focus on critical thinking, innovation, and problem-solving, we contribute to technological advancements while addressing real-world challenges.

ABOUT THE SPARC

The Scheme for Promotion of Academic and Research Collaboration (SPARC), an initiative of Ministry of Human Resource Development, Government of India, aims at improving the research ecosystem of India’s higher educational institutions by facilitating academic and research collaborations between top ranked Indian Institutions and globally ranked Foreign Institutions, through Joint Research Projects involving mobility of students and faculty. The SPARC Scheme is expected to have a major impact in providing the best international expertise to address major national problems, expose Indian academicians to the best collaborators abroad, enable international faculty to stay in India for a longer duration, provide Indian students an opportunity to work in the world class laboratories, to develop strong bilateral relationships in research, and improve the international ranking of Indian Institution.

COMMITTEES

CHIEF PATRON

Prof. V. Ramgopal Rao, Vice-Chancellor, BITS Pilani

PATRON

Prof. Sudhirkumar Barai, Director, BITS Pilani, (Pilani campus)

CONFERENCE CHAIR

Prof. Srikanta Routroy, Dept. of Mechanical Engineering, BITS Pilani, Pilani Campus.
Prof. Josua Meyer, Dept. of Mechanical and Mechatronics Engineering, Stellenbosch University.
Prof. Tunde Bello-Ochende, Dept. of Mechanical Engineering, University of Cape Town.

ORGANIZING SECRETARY

Prof. Suvanjan Bhattacharyya, Dept. of Mechanical Engineering, BITS Pilani, Pilani Campus.

CONVENER

Prof. Nirmalendu Biswas, Dept. of Power Engineering, Jadavpur University.
Prof. Saket Verma, School of Energy Science and Engineering, IIT Guwahati.
Prof. Ranjan Dey, Dept. of Chemistry, BITS Pilani, KK Birla Goa Campus.

TREASURER

Prof. Jitendra S. Rathore, Dept. of Mechanical Engineering, BITS Pilani, Pilani Campus.
Prof. A. R. Harikrishnan, Dept. of Mechanical Engineering, BITS Pilani, Pilani Campus.
Prof. Yogesh Singh, Dept. of Mechanical Engineering, BITS Pilani, Pilani Campus.

SPONSORSHIP

Prof. Raganayakulu C., Dept. of Mechanical Engineering, BITS Pilani, Pilani Campus.
Prof. Biswadip Shome, Dept. of Mechanical Engineering, BITS Pilani, Pilani Campus.
Prof. Shyam S Yadav, Dept. of Mechanical Engineering, BITS Pilani, Pilani Campus.
Dr. Devendra Kr. Vishwakarma, Dept. of Mechanical Engineering, Manipal University Jaipur.

ADVISORY COMMITTEE

Prof. Gautam Biswas, Dept. of Mechanical Engineering, BITS-Pilani, KK Birla Goa Campus.
Prof. A. C. Benim, Duesseldorf University of Applied Sciences, Germany.
Prof. Rachid Bennacer, Dept. of Civil and Envn. Engineering, ENS Paris Saclay, France.
Prof. Himadri Chattopadhyay, Dept. of Mechanical Engineering, Jadavpur University.
Prof. Amaresh Dalal, Dept. of Mechanical Engineering, IIT Guwahati.
Prof. M. S. Dasgupta, Dept. of Mechanical Engineering, BITS Pilani, Pilani Campus.
Prof. P. Srinivasan, Dept. of Mechanical Engineering, BITS Pilani, Pilani Campus.
Prof. Manoj Kr. Soni, Dept. of Mechanical Engineering, BITS Pilani, Pilani Campus.
Prof. Arun Kr. Jalan, Dept. of Mechanical Engineering, BITS Pilani, Pilani Campus.
Prof. Sharad Shrivasthva, Dept. of Mechanical Engineering, BITS Pilani, Pilani Campus.

SPEAKERS

Prof. Gautam Biswas

BITS Pilani

Prof. Tunde Bello Ochende

University of Cape Town, South Africa

Prof. John Abraham

St. Thomas University, USA

Prof. Ali Cemal Benim

Duesseldorf Universitty of Applied Sciences, Germany

Prof. A. E. Abed

ENS - Paris Saclay, Paris, France

Prof. Mohsen Sharifpur

University of Pretoria, South Africa

Prof. Mohammad M. A.

University of Staffordshire, UK

Prof. Smith Eiansa-ard

Mahanakorn University, Thailand

Prof. Aliashim Albani

University of Malaysia, Terengganu, Malaysia

Prof. Bahni Ray

IIT Delhi

Prof. Amaresh Dalal

IIT Guwahati

Prof. Sudeshna Mukherjee

BITS Pilani

Dr. Arun Choudhary

MNRE, New Delhi

Prof. Arindam Bit

IIT Guwahati

Dr. Sumit Kumar

Georgia Tech, USA

Prof. Syamantak Majumder

BITS Pilani

Dr. Gaurav Tomar

IISc Bengaluru

Dr. Vivek Kumar Singh

Indian Space Research Organization

Prof. Pradyumna Ghosh

IIT BHU

Prof. Himadri Chattopadhyay

Jadavpur University

Prof. Venugopal Arumuru

Indian Institute of Technology Bhubaneswar

Prof. Manabendra Pathak

Indian Institute of Technology Patna

Prof. Andrea Vallati

Sapienza University of Rome, Italy

Prof. Celestino Rodrigues Ruivo

University of Algarve, Portugal

Prof. Sandip Sarkar

Jadavpur University

ABSTRACTS

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

MAGNETO-CONVECTIVE HEAT TRANSFER OF CuO-WATER NANOFLUID IN A WAVY-WALLED POROUS ENCLOSURE WITH DISTRIBUTED HEATING UNDER THERMAL RADIATION

Sayandeep Sain[1], Nirmalendu Biswas[2*], Dipak Kumar Mandal[3] and Nirmal K. Manna[4]

[1]Department of Chemical Engineering, Jadavpur University, Kolkata 700032, India
[2]Department of Power Engineering, Jadavpur University, Kolkata 700106, India
[3]Department of Mechanical Eng., Government Eng. College Samastipur, Bihar, India
[4]Department of Mechanical Engineering, Jadavpur University, Kolkata 700032, India
[*]corresponding author email id: biswas.nirmalendu@gmail.com

ABSTRACT

Natural convection in nanofluid-saturated porous media under magnetic field influence represents a critical multi-physics problem relevant to thermal management in electronics, renewable energy systems, biomedical devices, and industrial processes. This study numerically investigates the coupled thermo-magneto-convective-radiative heat transfer of CuO-water nanofluid in a sinusoidal porous enclosure with distributed heating using the finite element method. The research examines the combined effects of magnetic field strength (Ha = 0–70), buoyancy forces (Ra = 10^3–10^6), porous media permeability (Da = 10^{-3}–10^{-1}), thermal radiation (Rd = 0–2), and enclosure orientation (λ = 30°–150°) on flow patterns, temperature distributions, heat transfer rates, and entropy generation. The Darcy-Brinkman-Forchheimer model describes flow through porous media [1–3], while the Rosseland diffusion approximation captures radiative effects [4]. The governing equations incorporating continuity, momentum (with Lorentz force terms), energy (with radiation source), and entropy generation are solved using Galerkin finite element method with adaptive meshing. Grid independence studies ensure numerical accuracy. Copper nanoparticles (2% concentration) enhance the base fluid thermal conductivity following established correlations [5]. Flow signature analysis (as shown in Fig. 1) reveals maximum velocity decreasing from 332 to 145 as Ha increases from 0 to 70 (Da = 0.1), demonstrating effective magnetic suppression of convection. Higher permeability (Da = 0.1) intensifies convective circulation compared to lower Da values. Heat transfer performance, quantified by Nusselt number, increases substantially from 3.81 to 10.7 as Ra increases from 10^3 to 10^6, indicating convection-dominant regimes at high buoyancy. Thermal radiation participation enhances Nu from 5.50 to 10.4 (Rd: 0→2 at Ra = 10^6), providing additional energy transport pathways. However, magnetic field application reduces Nu from 8.75 to 6.66 (Ha: 0→70 at high Ra) due to flow suppression. Streamline and heatline analyses reveal transition from conduction-dominated symmetric patterns at low Ra to complex multi-cellular structures at high Ra, with intermediate Da producing the most complex flow regimes. Comprehensive entropy generation analysis identifies viscous, thermal, and magnetic irreversibility sources. Total irreversibility remains relatively uniform (0.050–0.129×10^{-3}), indicating thermodynamically efficient operation. Viscous irreversibility dominates at lower Ra and higher Da, while thermal irreversibility becomes significant at elevated Ra. Magnetic irreversibility contributes substantially at higher Ha values through Joule heating effects. The sinusoidal geometry creates secondary vortices enhancing mixing compared to regular cavities, while distributed heating simulates realistic multi-source configurations.

PULSATILE HEMODYNAMICS IN HEALTHY AND DISEASED CORONARY ARTERIES: A CFD STUDY

Khemraj Deshmukh[1*], Gaurav Singh[2], Ravi Nigam[3]

[1] Department of Biomedical Engineering, Mody University of Science and Technology, Sikar Rajasthan.
[2] Department of Pharmaceutical science, Mody University of Science and Technology, Sikar Rajasthan.
[3] Department of Mechanical Engineering, Mody University of Science and Technology, Sikar Rajasthan.

ABSTRACT

Post-COVID-19 complications have emerged as major contributors to severe cardiovascular events, including myocardial infarctions, cerebrovascular accidents, and venous thromboembolism (VTE), leading to substantial morbidity and mortality across populations. Thrombosis formation within coronary arteries is strongly governed by local rheological behavior, which directly affects blood flow dynamics and clot progression. This study presents a novel comparative rheological evaluation of coronary artery flow under healthy (S1) and thrombosed (S2) conditions using Computational Fluid Dynamics (CFD) in COMSOL 5.6. The uniqueness of this work lies in its integrated assessment of five key rheological metrics pressure, axial velocity, shear rate, deformation gradient, and displacement gradient across 24 patient-specific geometries, under combined sinusoidal and pulsatile non-Newtonian flow with slip-wall boundary conditions, a configuration rarely reported in existing literature. Rheological behavior was analyzed at five strategic points (P1–P5) throughout the cardiac cycle (T). Quantitatively, healthy S1 models demonstrated more coherent shear-rate progression, with early peaks occurring before T/3 and stabilization approaching T. In contrast, thrombosed S2 geometries exhibited irregular shear-rate spikes and elevated pressure variability. The maximum shear rate reached 4,272.11 s^{-1} in the healthy non-Newtonian model, while thrombosed cases showed up to 38–52% higher local pressure fluctuations and 22–35% reductions in axial velocity, confirming significant hemodynamic disruption due to obstruction. Additionally, deformation and displacement gradients increased by 1.4– 1.7 times in S2 models, indicating intensified mechanical stress around the thrombus. These findings highlight the essential role of advanced rheological characterization in identifying thrombosis-induced flow abnormalities. The study's novel multi-parameter rheological framework offers a significant step toward improving prediction, early diagnosis, and clinical management of post-COVID thrombotic complications

Key words: *Thrombosis, uterine arteries, CFD analysis, rheological parameters, non-Newtonian*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

MAGNETO-CONVECTIVE HEAT TRANSFER OF CuO-WATER NANOFLUID IN A PLUS-SHAPED POROUS ENCLOSURE UNDER THERMAL RADIATION

Tushti Thakur[1], Sayandeep Sain[2], Nirmalendu Biswas[3*], Dipak Kumar Mandal[4] and Nirmal K. Manna[1]
[1]Department of Mechanical Engineering, Jadavpur University, Kolkata 700032, India
[2]Department of Chemical Engineering, Jadavpur University, Kolkata 700032, India
[3]Department of Power Engineering, Jadavpur University, Kolkata 700106, India
[4]Department of Mechanical Eng., Government Eng. College Samastipur, Bihar, India [*]corresponding author email id: *biswas.nirmalendu@gmail.com*

ABSTRACT

Cruciform cavities with perpendicular extended arms create distinct flow and thermal characteristics compared to standard enclosures, relevant to electronics cooling, biomedical microdevices, and industrial thermal systems. This investigation examines magnetohydrodynamic convection of CuOwater nanofluid in a plus-shaped porous cavity subjected to thermal radiation using finite element analysis. The work systematically evaluates effects of Rayleigh number ($Ra = 10^3–10^6$), Hartmann number (Ha = 0–70), radiation parameter (Rd = 0–2), and Darcy number (Da = 0.001–0.1) on flow structure, temperature distribution, heat transfer performance, and entropy production. The crossshaped configuration generates flow bifurcation at the central junction with extended heat pathways through orthogonal branches. Governing equations incorporating Lorentz forces, porous resistance via Darcy-Brinkman model, and radiative transport through Rosseland approximation are discretized using Galerkin finite element technique. Copper nanoparticles at 2% volume fraction modify base fluid properties following established correlations. Flow analysis indicates maximum velocity reaching 177 at $Ra = 10^6$ and Da = 0.01, with circulation exhibiting complex interactions between horizontal and vertical arms. The cruciform geometry produces primary vortices in each arm plus junction recirculation zones where perpendicular flows converge. Heat transfer quantification yields Nu = 6.86 at reference conditions, reflecting geometric effects on convective transport. Streamline patterns (ψ = -12.7 to 17.1) reveal asymmetric circulation from branch interactions and thermal boundary conditions. Heatline distributions (Π = -5.94 to 9.35) demonstrate energy flux pathways through perpendicular arms toward cooler regions. Viscous irreversibility ($NS_{v,max}$ = 1.96E-6) concentrates at junction corners and branch entrances where flow redirection creates elevated shear. Results guide design strategies for electronics cooling with plus-shaped heat spreaders channeling thermal loads through orthogonal branches to multiple sinks, microfluidic devices utilizing cruciform channels for sample mixing with magnetic flow control, and building ventilation systems employing plus-shaped manifolds for balanced air distribution. Parametric examination confirms plus-shaped geometry substantially alters convective behavior through branch coupling, with circulation complexity arising from orthogonal arm interactions.

Keywords: *Natural convection, MHD, Nanofluid, Porous media, Thermal radiation, Entropy generation.*

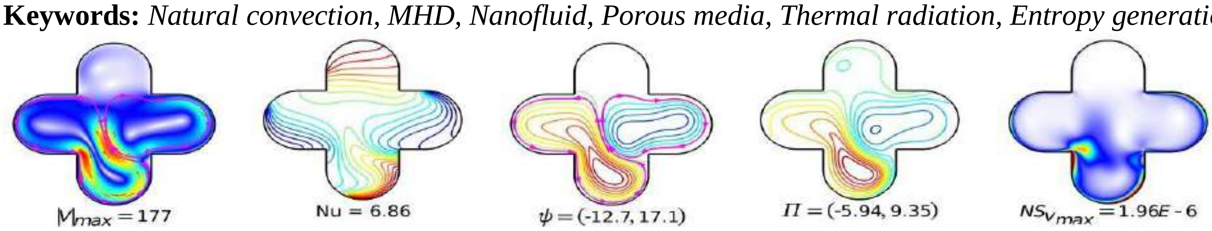


Fig. 1. Flow signature, isotherms, streamlines, heatlines and Viscous Irreversibility ($Ra = 10^6$, Rd = 1, Ha = 30, λ = 90° and Da = 0.01)

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

A COMPARATIVE STUDY ON PULSE LASER JOINING USING DIFFERENT COMBINATIONS OF Al- AND Mg-ALLOYS

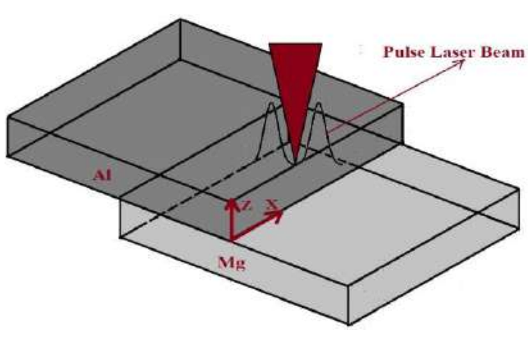
Asish Tripathy, Sayantan Sinha Biswas, Himadri Chattopadhyay and Nilkanta Barman
Department of Mechanical Engineering, Jadavpur University, Kolkata, India - 700032
drnilkantabarman@gmail.com, atripathy955@gmaial.com

ABSTRACT

A numerical investigation of transport phenomena during pulse laser joining of different Al- and Mg-alloys combinations is presented, where two thin sheets, each 100 mm × 50 mm × 2 mm, are placed in a lap configuration as shown in Figure 1. The mass, momentum and energy conservative equations, along with appropriate boundary conditions, are used to model the process. An argon laser beam of 1900 W in pulse form is considered for their joining, which is represented as a volumetric heat source in the model. The enthalpy update scheme is incorporated to account for the simultaneous occurrence of melting and solidification phenomena, and to identify the weld pool during the joining process. The set of governing equations is discretized using the finite volume method (FVM) and the SIMPLER algorithm. A FORTRAN-based numerical code is developed to solve the obtained linear algebraic equations using the TDMA. Introducing entropy generation and thermal stress development in the model to identify the zone of crack propagation and bending is a new approach in this study. A summary of the predicted results is presented below.

- After a suitable grid-independent study, the present prediction considering a set of 5A06-Al and AZ31B-Mg alloys is validated with an existing experiment, showing a good agreement.
- A maximum temperature of 720 °C is found with thermal stress and entropy distributions as shown in Figure 2 at t = 13 s.
- A dynamic weld pool is observed for pulse laser application.
- Different Al-alloys (2024Al, 5083Al and 5454Al) are combined with the AZ31B Mg-alloy to investigate the effect of thermal properties on the transport phenomena.

Key Words: *Pulse laser joining, a set of dissimilar materials, modelling, weld pool, HAZ*



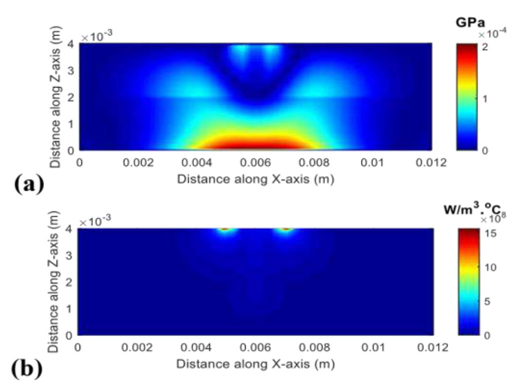


Figure 1: A physical problem

Figure 2: (a) thermal stress and (b) entropy

INFLUENCE OF MAGNETIC FIELD ON THE IMPINGEMENT DYNAMICS OF FERROFLUID DROPLETS ON HEATED HYDROPHOBIC SURFACES

Ram Krishna Shah[1]
[1]Mechanical Cluster, School of Advanced Engineering, UPES, Dehradun - 248007, India
*Email: *ramk.sha@ddn.upes.ac.in*; *rammax8@gmail.com*

ABSTRACT

Droplet interactions with solid surfaces are central to a wide range of natural and engineered thermal processes. Although the impingement dynamics of conventional aqueous droplets are well documented, the behaviour of ferrofluid droplets subjected to non-uniform magnetic fields during impact on heated hydrophobic substrates remains insufficiently explored. This study presents a computational framework that couples magnetic, hydrodynamic, and thermal fields to analyse ferrofluid droplet impingement under externally applied magnetic forces, a capability not addressed comprehensively in earlier works. The results show that the magnetic field significantly modifies interfacial evolution and heat transfer by intensifying inward magnetic body forces during the spreading process. Across the range of Weber numbers investigated (10 – 50), the magnetic field increases the maximum spreading diameter by up to 35%, suppresses recoil even on highly hydrophobic surfaces (θ = 135° – 150°), and enhances the timeaveraged wall heat transfer by as much as 75% due to increased contact area and longer interaction time. The study further quantifies the relationship between magnetic field strength and spreading behaviour, demonstrating a consistent increase in the maximum spreading coefficient with stronger field gradients. The novelty of this work lies in (i) the integration of a non-uniform magnetic field into droplet impact simulations on heated substrates, (ii) the quantitative assessment of coupled magneto-hydrodynamic and thermal effects on droplet spreading, and (iii) the development of generalised scaling behaviour relevant for magnetic actuation in thermal management systems. These findings offer new insights into the tunability of ferrofluid droplet dynamics, underscoring the potential of magnetic fields for adaptive cooling and enhanced heat transfer [1].

Key Words: *Magnetic field, ferrofluids, Droplet dynamics, Thermal transport*

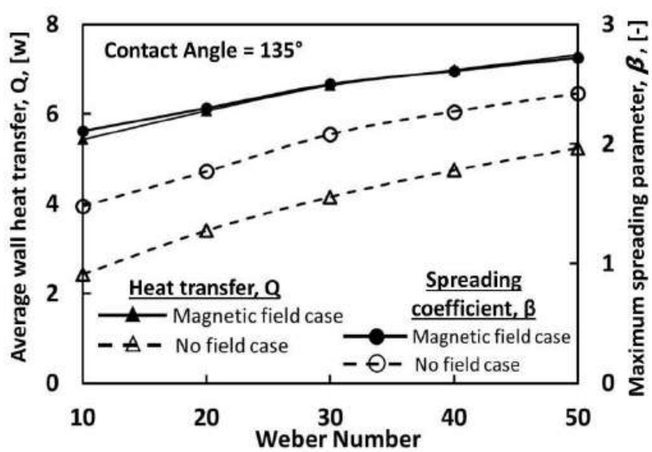


Figure 1: Comparison of wall heat transfer (Q) and maximum spreading parameter ($\beta = D/D_o$) for magnetic and no field cases at different Weber numbers for a static contact angle of 135°

CFD ANALYSIS OF DYNAMIC PRESSURE-INDUCED STIFFNESS IN A PEDIATRIC MAGNETICALLY LEVITATED VENTRICULAR ASSIST DEVICE

Khushi Rele[1] and Tadahiko Shinshi[2]

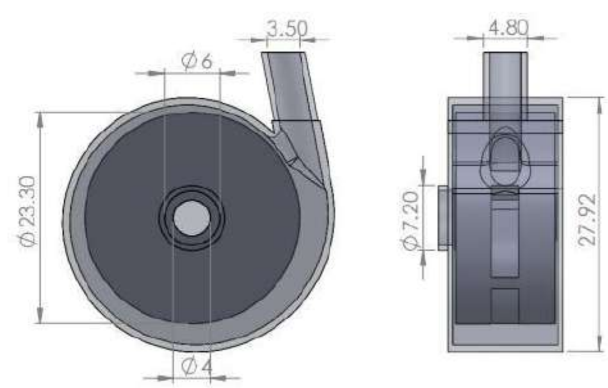
[1]Mechanical Engineering, Indian Institute of Technology Madras - 600036, me22b145@smail.iitm.ac.in

[2]Institute of Integrated Research, Institute of Science Tokyo - 226-8501, shinshi.t.ab@m.titech.ac.jp

ABSTRACT

Magnetically levitated ventricular assist devices (VADs) for infants require compact designs. They also demand a highly stable rotor suspension to minimise blood damage and mechanical wear. The passive magnetic stiffness in ultra-small devices remains limited. Consequently, it is critical to determine whether hydrodynamic forces stabilise or destabilise impeller motion during levitation. This study presents a computational fluid dynamics (CFD) analysis of a 23.3-mm intracorporeal pediatric centrifugal blood pump. Fluid-induced radial and tilt stiffness were quantified to calibrate magnetic bearing forces. These forces counteract flow-driven disturbances. A SOLIDWORKS model was simulated in ANSYS Fluent. The multiple reference frame (MRF) approach was used at 4000 rpm and 2 L/min. Forces and moments were calculated for eccentricities up to 0.3 mm and tilt angles of ±1° to obtain stiffness tensors. The simulated pressure head (80–95 mmHg) matched the experimental measurements, validating the model. The results demonstrate a key finding. Contrary to the common assumption that the working fluid provides a stabilising effect, intra-pump hydrodynamic forces in this pediatric VAD generated mainly **negative radial stiffness** (K_{xx} = –1.76 N/mm; K_{yy} = –1.35 N/mm). These forces drew the impeller toward the casing, increasing the risk of contact. Implementing a doublevolute geometry improved force balance. It reduced destabilising stiffness by approximately 32%. Tilt simulations indicated minor restoring stiffness (0.1–1 mNm/°) compared to magnetic stiffness (12–14 mNm/°). This confirms that magnetic control is the primary stabilising mechanism. These findings provide quantitative evidence of flow-driven destabilisation in intracorporeal pediatric VADs. They directly inform the optimisation of magnetic forces for safe, reliable long-term implantation.

Key Words: *Pediatric Ventricular Assist Device, Magnetic Levitation, CFD, Pressure-Induced Stiffness, Volute Geometry, Impeller Stability.*



INFLUENCE OF PAIR CORRELATION FUNCTION ON LATENT HEAT AND EVAPORATION DYNAMICS OF SUB-MICRON DROPLET

Rahul Bhattacharjee[1] and Anirudh Singh Rana[1]

[1]Department of Mathematics, Birla Institute of Technology and Science Pilani, Rajasthan, 333031,
anirudh.rana@pilani.bits-pilani.ac.in

ABSTRACT

We investigate the evaporation of sub-micron droplets using a thermodynamically consistent Navier-Stokes-Korteweg (NSK) model derived from the Enskog-Vlasov (EV) equation. The model incorporates finite molecular size and long-range attractive forces, allowing a unified description of liquid, vapor, and the diffuse interface without empirical evaporation coefficients. The central focus of this work is to assess how different pair-correlation functions Y(n) influence the latent heat of vaporization and, consequently, the evaporation rate. Three closures are examined—van der Waals, generalized Lennard-Jones, and classical .Carnahan-Starling pair correlations functions (CCSpc). We introduce a modified CarnahanStarling pair correlation (MCSpc), with a fitted parameter β, that corrects the entropy contribution in the EV framework. This modification enables the model to reproduce NIST latentheat values for neon, argon, and xenon with <0.1% error, outperforming the other pair-correlation forms which deviate by 25-55% at subcritical temperatures. A key result is shown in Fig. 1, where the evaporation rate predicted by the NSK-EV model with MCSpc is compared against molecular dynamics (MD) data for a 1.68 nm argon droplet. The proposed model agrees with MD within ~5%, whereas the CCSpc form shows deviations exceeding 20%, highlighting the necessity of the fitted correlation.

Key Words: *Navier-Stokes-Korteweg, Carnahan-Starling pair correlation, d^2-law, Enskog-Vlasov, Korteweg stress tensor.*

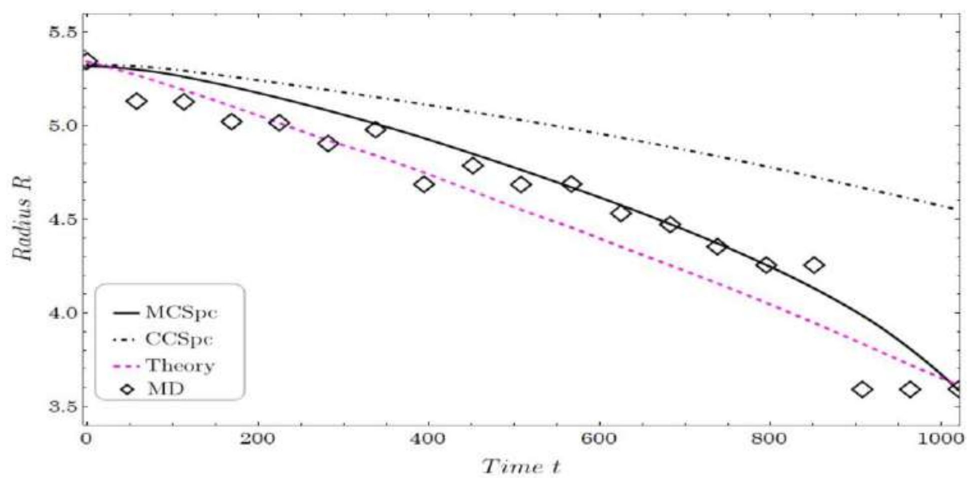


Figure 1: Comparison of the evaporation rate of an argon droplet predicted by the NSK-EV model with the MCSpc closure (black solid line) against MD simulation data (black diamonds), CCSpc (black dashed line) and the Knudsen aerosol theory (magenta dashed line) reported by Long et al.

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

ENHANCING BOILING HEAT TRANSFER ON AN ULTRAFINE NANOFIBROUS COATED SUPERHEATED SURFACE USING ETHANOL/SWCNT NANOFLUID

Ravi Pippal[1], Pushpendra Kumar Shukla[1] and Sumit Sinha-Ray[1,2]

[1]Department of Textile and Fibre Engineering, Indian Institute of Technology – Delhi, Hauz Khas, New Delhi 110016, India, ravimecher15@gmail.com, pkshukla.manit@gmail.com, and ssinharay@textile.iitd.ac.in

[2]Department of Mechanical and Industrial Engineering, University of Illinois at Chicago, Chicago, IL 60607-7022, USA

ABSTRACT

This study investigated saturated pool boiling of ethanol and single-walled carbon nanotube (SWCNT) based nanofluid (NF) (0.01 wt%) on bare and nanotextured (NT) copper surfaces, aiming at solving the thermal management crisis of microelectronics. The nanotextured surfaces were created via supersonic solution-blown ultrafine polymeric nanofibers. The nanotexturing was optimized based on coating thickness and its performance in boiling conditions. The key parameters analyzed include critical heat flux (CHF), boiling heat transfer coefficient (BHTC), overheating factor (β), and bubble dynamics parameters (such as bubble departure frequency [f_b]) across all the fluid-surface combinations. CHF represents the maximum heat removal capacity before dry-out, serving as a thermal safety threshold. BHTC quantifies the effectiveness of heat transfer per unit temperature difference, reflecting the system's cooling efficiency. The overheating factor indicates the degree of superheat at which vapor bubbles form and depart, influencing bubble nucleation intensity. Bubble departure frequency measures the rate of bubble release from the surface, correlating with the latent heat removal rate during boiling. The results showed that the average CHF of NT surfaces, NT/45, NT/90, and NT/135 (the numerical indicating coating deposition time in seconds) improved by approximately 11.3%, 54.8%, and 12.9%, respectively, compared to bare surface with ethanol, whereas for nanofluid the increments were 27.4%, 85.5%, and 33.9%, respectively. CHF increased from 654.26 kW/m^2 for Bare+Ethanol to 1213.55 kW/m^2 for NT/90+NF, showing an 85.5% rise. Correspondingly, BHTC showed a 244% improvement, peaking at 59.1 kW/m^2·K for NT/90+NF from 17.2 kW/m^2·K for Bare+Ethanol, indicating significant thermal energy transfer. The overheating factor rose up to 0.89 for NT/90+NF as compared to 0.5 for Bare+Ethanol, enabling stronger bubble nucleation at lower surface superheat, optimizing boiling inception at about 10.9 °C for NT/90+NF surface superheat compared with 12.8 °C for Bare+Ethanol. Surface nanotexture improved bubble departure frequency from 75 Hz for Bare+Ethanol to 253 Hz for NT/90+NF, signifying improved dynamic vapor removal and heat flux sustainability. These improvements resulted from increased active nucleation sites, higher surface wettability due to nanofiber coatings, and enhanced thermal conductivity afforded by CNT deposition on the NT surface. The fibrous network also disrupted large bubble coalescence, producing smaller and more frequent bubbles, which amplified heat transfer rates. This synergistic effect of increased CHF, BHTC, overheating factor, and bubble departure frequency highlights the role of surface engineering combined with nanofluids in pushing the limits of pool boiling performance. Such strategies hold substantial promise for advanced microelectronics thermal management and other heat-intensive applications.

Key Words: *Ultrafine nanofiber, Nanofluid, ONB, Critical heat flux, Macro-layer*

NUMERICAL ANALYSIS OF Fe_3O_4–H_2O NANOFLUID-BASED COLD PLATE FOR ENHANCED THERMAL MANAGEMENT UNDER HIGH HEAT FLUX CONDITIONS

Amish Samaiya[1], Chandra Tivrakrishna Panday[1], Somenath Gorai[1] and Devendra Kumar Vishwakarma[1]
[1]Department of Mechanical Engineering, Manipal University Jaipur, Jaipur-303007, India,
devendra.vishwakarma@jaipur.manipal.edu

ABSTRACT

Efficient heat removal is crucial for next-generation electronic and energy systems operating under high thermal loads. Cold plates with pin fins are widely adopted for such applications, where cooling effectiveness depends strongly on fin arrangement and geometry. This study focuses on improving the cooling performance of high heat flux generating electronic devices, as processing units are becoming compact, faster, and more powerful. As these compact devices handle heavier workloads, the heat generation increases significantly, demanding more efficient cooling methods. To address this challenge, Fe_3O_4 + H_2O nanofluid with a 2% volume concentration were used in micro pin-fin heat sinks to enhance heat transfer and temperature control. Numerical simulations were performed under a constant heat flux of 37,200 W/m² and an inlet temperature of 298 K, with Reynolds numbers ranging from 250 to 550 to represent different flow conditions. The results showed that nanofluid significantly improves heat transfer compared to conventional coolants due to its superior thermal properties and enhanced fluid mixing. It was also observed that flow distribution, particularly near the initial rows of fins, plays a crucial role in determining overall cooling efficiency. Overall, the combination of micro-scale heat sink design and advanced nanofluid demonstrates a promising approach to achieving efficient thermal management in compact, high-performance electronic devices. The present work aims to enhance the cooling performance in modern systems, where the processing units of high performing devices are getting smaller in size and to come along with it we have proposed the innovative heat transfer method in the existing technology.

Key Words: *Cold Plate, Nanofluid, Heat Transfer Enhancement, Micro-Pin Fins, and Electronic Cooling*

HYDROTHERMAL CHARACTERISTICS AND SECOND LAW ANALYSIS OF MAGNETO-NANOFLUID FLOW IN AN OCTAGONAL POROUS ENCLOSURE FITTED WITH FINS

Anubhav Chowdhury[1], Nirmalendu Biswas[2*], Dipak Kumar Mandal[3] and Suvanjan Bhattacharyya[4], and Nirmal K. Manna[5]

[1]Department of Civil Engineering, Jadavpur University, Kolkata 700032, India
[2]Department of Power Engineering, Jadavpur University, Kolkata 700106, India
[3]Department of Mechanical Eng., Government Eng. College Samastipur, Bihar, India
[4]Department of Mechanical Eng., Birla Institute of Technology and Science Pilani, Rajasthan, India
[5]Department of Mechanical Engineering, Jadavpur University, Kolkata 700032, India
[*]corresponding author email id: biswas.nirmalendu@gmail.com

ABSTRACT

The present investigation explores the hydrothermal behavior of CuO-water nanofluid in a porous octagonal cavity, containing a conducting block fitted with fins. The applications of fins have attracted significant attention due to their ability to enhance thermal performance, which is relevant to engineering applications such as automobile radiators, hydrogen fuel cells, biomedical microdevices, industrial thermal systems, etc. Unlike previous works that focused mainly on simple geometries or non-porous enclosures, the present work introduces an octagonal porous enclosure with an internal finned conducting block under a discrete sidewise heating-cooling configuration, and a vertically applied magnetic field, providing a novel configuration for enhanced thermal efficiency. The impact of numerous control parameters is evaluated systematically, including the effects of the modified Rayleigh number ($10 \leq Ra_m \leq 10^4$) and Hartmann number ($0 \leq Ha \leq 75$), while maintaining fixed values of radiation parameter ($Rd = 1$), Darcy number ($Da = 10^{-3}$), porosity ($\varepsilon = 0.8$), and nanofluid volume fraction ($\phi = 0.02$). The governing equations are formulated using the Brinkman-Forchheimer porous media model with the Boussinesq approximation and solved in their non-dimensional form through the Galerkin finite element method. The flow configuration, heat transfer characteristics, and irreversibility analysis are depicted through the resulting streamline, isotherm, heatline, and respective entropy generation distributions. The maximum rate of heat transfer is achieved at $Ra_m = 10^4$ in the absence of the Lorentz force interferences, with Nu reaching up to 24.06. This trend extends to the quantitative magnitudes of stream function, heat function, and total entropy generation as well. The hydrodynamic analysis suggests that the fluid circulation intensifies with increasing Ra_m. Consequently, the vortices begin to stretch towards the periphery, leading to the formation of a single, clockwise recirculation pattern within the enclosure, with streamline magnitudes intensifying from $\psi = (0, -1.15)$ to $\psi = (0, -68.77)$. Conversely, as Ha increases, the convective heat transfer weakens, giving rise to conduction-dominated transport, marked by a decline of the average Nu from 11.53 to 8.60 for $Ra_m = 10^3$. The Lorentz force redistributes thermal energy circulation patterns with heatline modifications from $\Pi = (13.99, 35.46)$ to $\Pi = (11.70, -29.35)$. These results contribute to the optimization of thermal management systems employing octagonal enclosures with internal fins. Such configurations enhance thermal transport and fluid circulation by increasing the effective heat transfer area and guiding flow circulation along finned surfaces, making them suitable for applications in electronics cooling, microchannel heat sinks, and thermal energy storage systems.

EFFECT OF RBC ON THE THERMAL TRANSPORT IN CAPILLARY BLOOD FLOW IN RESPIRATORY SYSTEM

Abhijit Dutta[1*], Subhajit Banerjee[2,3], Sudhir Chandra Murmu[3] and Himadri Chattopadhyay[3]

[1]Department of Mechanical Engineering, MCKV Institute of Engineering, Howrah, West Bengal, India, abhijit_me2005@yahoo.co.in
[2]Department of Mechanical Engineering, Greater Kolkata College of Engineering and Management, Baruipur, West Bengal, India, bsubhajit15@gmail.com
[3]Department of Mechanical Engineering, Jadavpur University, Kolkata, West Bengal, India, murmusudhir@gmail.com, himadri.chattopadhyay@jadavpuruniversity.in

ABSTRACT

Heat and mass transfer through the human respiratory capillary is a complex phenomenon through which oxygen uptake and de-carboxygenation take place. During oxygen uptake, the Red Blood Cells (RBCs) are saturated and produce oxy-haemoglobin (HbO_4). This process is exothermic and releases heat typically around 31.34 kJ/ gm-molecule of Hb. On the other hand, the velocity of the capillary flow varies between 0.0005 to 0.0015 m/s, which is very low. Therefore, the objective of this work is to study the thermo-fluid behaviour of the capillary flow. A 3D model has been developed in a solid modelling software environment based on available dimensions in open literature, and then it was imported to the COMSOL Multiphysics software to observe the thermofluidic nature. This study employs finite element analysis to model capillary blood flow in microcirculatory systems. Computations have been performed with meshes of 8,20,117, 6,51,343, 3,73,561 and 2,15,714 nodes for obtaining grid independence where the x- directional velocity (u) was compared. Finally, the mesh with 3,73,561 elements is chosen for computation. For this purpose, the grid-independent value was extrapolated from values obtained at four different grid levels which differ by only approximately less than 1% from the chosen mesh. The simulation integrates heat transfer and fluid dynamics, providing insights into the complex interactions between apparent viscosity, shear rate, friction factor, and entropy generation under different physiological conditions. The fluid characteristics are considered as non-Newtonian, where the Casson and Power law models have been considered. The Bejan number (Be), a dimensionless parameter, quantifies the ratio of heat transfer irreversibility to total (heat transfer and frictional) irreversibility in thermal systems. However, the Brinkman number (Br) is also a dimensionless number that signifies heat generation in a viscous fluid to the external heat supplied. The distribution of Be and Br along the length has been observed. The present work assesses the Bejan number to evaluate entropy generation and flow irreversibility in capillary blood flow and oxygenation. The Fåhræus–Lindqvist effect has been observed where apparent viscosity drops around the RBC for both models. The friction factor around the RBC also changes due to the change in velocity magnitudes.

Key Words: *Heat Transfer, Finite Elements, Entropy Generation, Bejan Number, Fåhræus–Lindqvist effect.*

HYBRID MAGNETIC NANOFLUID COOLING WITH VIRTUAL VORTEX GENERATION FOR ENHANCED BATTERY THERMAL MANAGEMENT IN ELECTRIC VEHICLES

Chandra Tivrakrishna Panday[1], Amish Samaiya[2] and Devendra Kumar Vishwakarma[1,2,*], Suvanjan Bhattacharyya[3]

[1]Department of Mechanical Engineering, Manipal University Jaipur, Jaipur-Ajmer Express Highway, Jaipur 303 007, Rajasthan, India.

[2]Multiscale Simulation Research Center (MSRC), Manipal University Jaipur, Jaipur 303 007, Rajasthan, India

[*]E-mail: devendra.vishwakarma@jaipur.manipal.edu

[3]Department of Mechanical Engineering, Birla Institute of Technology and Science, Pilani, Pilani Campus, Pilani 333 031, Rajasthan, India

E-mail: suvanjan.bhattacharyya@pilani.bits-pilani.ac.in

ABSTRACT

In the present study, a novel vortex generator-based hybrid heat transfer technique has been introduced, combining the effects of channel surface roughness, magnetic nanofluid, and externally applied magnetic fields for efficient battery thermal management. The cooling channel designed for this study features a diverging–converging geometry with an inlet width of 3 mm, an outlet width of 2 mm, a maximum width of 4 mm, and a total channel length of 40 mm. The simulations are performed under laminar flow conditions with Reynolds numbers ranging from 100 to 500 and magnetic field intensities varying from 800 G to 2000 G. The working fluid is an Fe_3O_4–water (H_2O) magnetic nanofluid with a 2% volume concentration. The magnetic field is applied at four locations along the channel, where the resulting magnetic forces induce synthetic (non-physical) vortices within the flow field. These vortices enhance fluid mixing and thermal disturbances, leading to improved convective heat transfer performance of the system.

Key Words: *Hybrid Cooling Techniques, Heat Transfer Enhancement, Magnetic Field, Nanofluids, Battery Thermal Management*

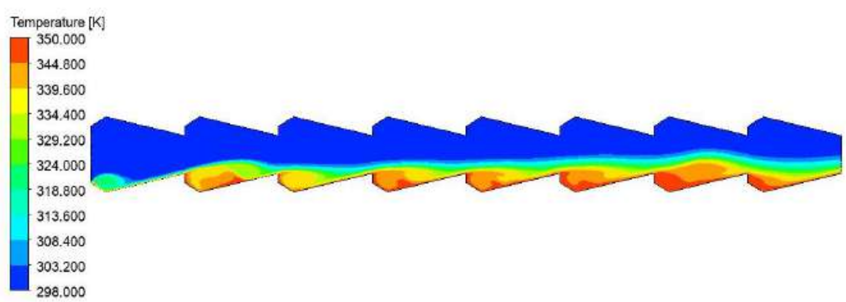


Figure 1: Minichannel showing temperature distribution

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

NUMERICAL INVESTIGATION OF CAMBER MORPHING AERODYNAMICS FOR UNMANNED AERIAL VEHICLE APPLICATION

Satvik Sandeep Sarode[1], and Manabendra Manindrakumar De[2]

[1]BITS Pilani, Pilani Campus, satvik.sarode@gmail.com (Presenting author)

[2]CSIR-National Aerospace Laboratories, Bengaluru, manav.nal@csir.res.in (corresponding author)

ABSTRACT

Unmanned Aerial Vehicles (UAVs) demand optimal aerodynamic forces during various phases of flight and manoeuvres. Conventional approach to cater to these requirements is through deflection of discrete control surfaces. However, this approach comes at a cost of aerodynamic losses. Natural flyers like birds have addressed this issue by adopting active morphing of wing. With the advancement in materials science, control algorithms, and compliant structures, researchers across the globe are making efforts to mimic the bioinspired morphing approach for UAVs. Literature have indicated that camber morphing is preferred for UAV application. However, to effectively utilize the morphing approach, it is important to understand the finer details of the underlying fluid dynamics. In line with this, the present paper reports the effort to gain an insight into the transient aerodynamics of global camber morphing from -5° to +35° angles of attack at a chord-based Reynolds number of 6.5 x 10^5. Conventionally, quasi-steady simulations have been used for similar studies and vorticity contours have been employed for flow visualization. The novelty of present study lies in the fact that 2D time-resolved simulations using Dynamic meshing technique have been implemented and Q criteria based flow visualization has been used for understanding the aerodynamics behind the hysteresis loops observed in the force patterns (Fig. 1 a,d) during various instances of morphing for different angles of attack (Fig. 1 b,c,e,f). Open-source high fidelity numerical solver OpenFOAM was used for carrying out the time-resolved simulations.

Key Words : *Global camber morphing, Unmanned Aerial Vehicle, Dynamic Meshing, Q-criteria*

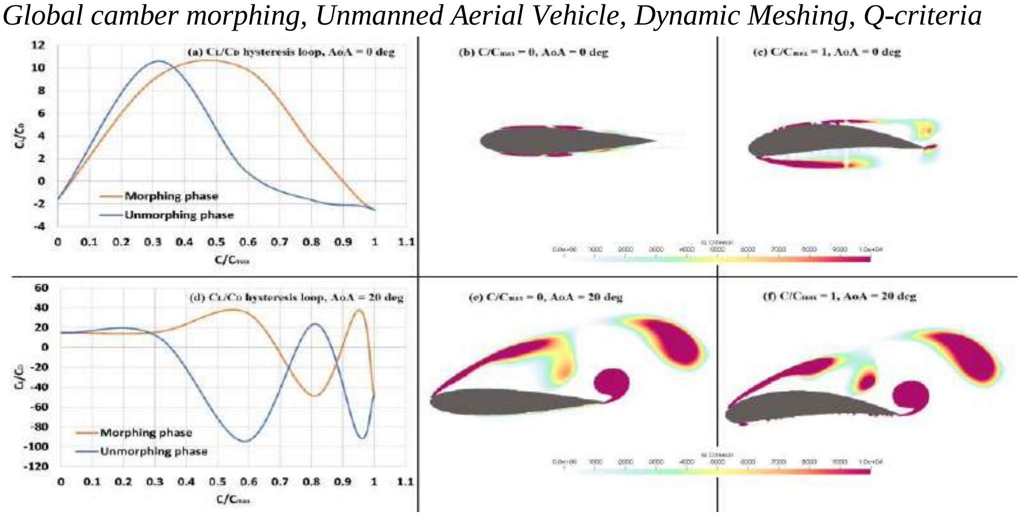


Figure 1: Force patterns and flow visualization of global camber morphing aerofoil

A NOVEL RING-WITH-FINS AND PLATE DESIGN FOR EFFICIENT HEAT DISSIPATION IN AIR-COOLED LITHIUM-ION BATTERY PACKS

Ram Babu Gupta[1*], Rahul Ranjan[1], Devesh Kumar[1], Tanmay Dutta[1]
[1]Department of Mechanical Engineering, IIT(ISM) Dhanbad, Dhanbad-826004, Jharkhand, India
*Email – ram.gupt95@gmail.com

ABSTRACT

An efficient Battery Thermal Management System (BTMS) is essential for maintaining the safety, lifespan, performance, and reliability of lithium-ion battery packs—particularly as the global shift toward cleaner energy sources continues to reduce environmental pollution and dependency on fossil fuels. Lithium-ion batteries is the primary power source for electric vehicles, offering a sustainable alternative to traditional internal combustion engines which run by fossil fuels. However, effective thermal regulation is important to preserving battery performance and preventing safety risks such as thermal runaway, which may result from excessive heat generation during charging and discharging cycles of the battery pack(BP).Conventional natural or forced aircooling BTMS configurations often face several limitations, including elevated cell temperatures, uneven temperature distribution across the BP, localized hot spots near the terminals, and inefficient heat dissipation from the central region of the BP. Previous studies have mainly addressed only one or two of these issues simultaneously and commonly employed fins to enhance conductivity. However, such fins are integrated at the individual-cell level and improve heat dissipation locally rather than uniformly throughout the BP. To tackle these challenges, this study proposes a novel aircooled BTMS design that enhances heat removal through a synergistic combination of conduction and convection approaches. The proposed system incorporates highly conductive rings with fins and plates within the BP. The rings with fins are tightly fitted around the cell surfaces, while the plates are positioned above the top busbars and below the bottom busbars. Ring with fins positioned between adjacent cells, which are perpendicular to the height of the cells, ensuring tight surface contact for improving the heat conduction. This design innovation improves the heat dissipation not only at the individual cell level but also across the whole BP. Additionally, an extra air inlet is introduced at the bottom of the battery pack, while 2 mm extra gap provided in the central row of the cells to improve the air flow and heat dissipation rate. Numerical investigation were done for various BTMS configurations under a 4C discharge rate. The results reveal that the optimized model—featuring fin plates with rings, plates, and an additional air inlet—achieves superior thermal performance. Compared to a standard battery pack (STBP), the proposed system reduces the maximum temperature (Tmax) by 23 % and minimizes the temperature difference (ΔT) to 2.8°C, demonstrating its effectiveness in achieving uniform temperature distribution and efficient heat dissipation.

Key Words: *Battery thermal management, Air cooling, Heat transfer enhancement, Computational Fluid Dynamics.*

DRYING CHARACTERISTICS OF CLUSTER FIGS (FICUS CARICA L.) USING EVACUATED TUBE SOLAR DRYER WITH AND WITHOUT THERMAL ENERGY STORAGE

Ankit Kumar Agarwal[1,2], K.B. Rana[1], B. Tripathi[1]

[1]Department of Mechanical Engineering, Rajasthan Technical University, Kota – 324010, India,
akagarwal.87@gmail.com

[2]Department of Mechanical Engineering, Swami Keshvanand Institute of Technology, Management and Gramothan, Jaipur – 302017, India

ABSTRACT

This research investigates the drying characteristics of cluster figs using an indirect evacuated tube solar dryer, aiming to extend shelf life, enhance product quality, and minimize post-harvest losses. Fresh figs deteriorate rapidly due to their high moisture content, making effective drying methods essential for long-term preservation. In this study, a portable evacuated tube indirect solar dryer was designed to harness solar radiation efficiently. The system operated by heating air inside the evacuated tube collector, with a blower circulating the warm air through the drying chamber to ensure uniform heat transfer and controlled dehydration.

Drying experiments were performed at air mass flow rates of 0.01, 0.02, and 0.03 kg/s in both the indirect solar dryer and traditional open sun drying. Thermal energy storage (TES) using pebble beds was incorporated to maintain higher temperatures during off-sunshine periods, thereby stabilizing the drying process. The dryer demonstrated a notable temperature increase of 15.2°C above ambient air under solar radiation of 883 W/m², confirming its strong thermal performance. Significant improvements in drying efficiency were observed across multiple configurations. The highest drying rate of 0.282 g/g dm-h was recorded in the indirect solar dryer with thermal storage and quenching (ISDTESQ), followed by 0.231 g/g dm-h in ISD, 0.216 g/g dm-h in ISDTES, and 0.178 g/g dm-h in open sun drying. Drying time was reduced by 48% in ISDTESQ, 38% in ISDTES, and 26% in ISD compared to open sun drying. These reductions are attributed to sustained higher temperatures within the drying chamber and the formation of micro-cracks during the quenching process, which facilitated faster moisture diffusion. Earlier findings also indicated a 32% reduction in drying time at 0.03 kg/s airflow with pebble bed storage, further supporting the dryer's superior performance. To evaluate the drying kinetics, experimental data were fitted to eleven thinlayer models. The Page model provided the best fit, exhibiting the highest coefficient of determination (R^2 = 0.999), and the lowest standard error values (SSD = 0.0019, RMSE = 0.0074), confirming its suitability for predicting the moisture behavior of cluster figs. The overall thermal efficiency of the dryer was determined to be 36%.

Key Words: *Indirect solar dryer, Evacuated tube, Cluster figs, Ficus Carica, Thermal energy storage, Pebble bed*

NUMERICAL ANALYSIS OF HYDROGEN MICRO-MIX COMBUSTOR

Varun Dinachandra[1], Siddharth Satheesan[1] and Sumer B Dirbude[1]
[1]Department of Mechanical Engineering, NIT Calicut-673601, sumer@nitc.ac.in

ABSTRACT

Hydrogen is a promising fuel if used in the conventional gas turbine using a micro-mix concept. However, the effect of the equivalence ratio (as it implies lean or rich conditions) on flame stability, NOx production, etc., plays a very crucial role in determining the operating conditions of the combustor. While micro-mix hydrogen combustor studies exist in the open literature, the stability behaviour under changing equivalence ratios has not been clearly understood due to multi-physics, and small-scale (<1mm) phenomena. Also, the majority of the work focuses on NOx and emission aspects and completely ignores the equivalence ratio effect on the stability behaviour. In this work, therefore, a high-fidelity 3D CFD model (ANSYS Fluent) is used in analysing flame stability, asymmetry, and instability, revealing stability limits. It further identify an optimal operating window for low-NO_x micro-mix combustion. Here, SST k-ω and the eddy dissipation concept (EDC) are combined with detailed hydrogen-air reaction mechanisms, respectively, to include turbulence and combustion. The simulation results—after grid independence and model validation—show that flame stability, temperature distribution, and NOx formation are strongly related to the equivalence ratio (ϕ). At a lean condition $\phi = 0.3$, the flame temperature is low, leading to negligible NO_x formation. When the equivalence ratio reaches ϕ to 0.4, the flame temperature rises approximately to ~2200 K, resulting in more stable and efficient combustion and a corresponding moderate increase in NO_x. Further increasing the equivalence ratio to $\phi = 0.7$ intensifies combustion with peak temperatures exceeding ~2390 K. Under these conditions, NO_x formation increases sharply, and signs of flame instability and asymmetry emerge. This work highlights the increasing NO_x emissions trend in practical hydrogen-air combustion with the increase in flame temperature. While higher temperatures enhance combustion efficiency, the high diffusivity and low thermal inertia can induce flame instability, restricting the optimal operating range $\phi = 0.3$–0.4, where emission levels and the energy output are more favourably balanced. The results are valuable for the design of hydrogen-based, low-emission propulsion systems.

Key Words: *Micro-mix combustor; Hydrogen combustion, NOx predictions, Computational fluid dynamics*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

FLUID FLOW AND HEAT TRANSFER OF AN IMPINGING SWEEPING JET IN ENHANCED SURFACE FLUIDIC OSCILLATOR

Devendra Kumar Vishwakarma[1,2] and Suvanjan Bhattacharyya[3]

[1]Department of Mechanical Engineering, Manipal University Jaipur, Jaipur-Ajmer Express Highway, Jaipur 303 007, Rajasthan, India.

[2]Multiscale Simulation Research Center (MSRC), Manipal University Jaipur, Jaipur 303 007, Rajasthan, India.

[3]Department of Mechanical Engineering, Birla Institute of Technology and Science, Vidya Vihar, Pilani, Rajasthan 333031, India.

E-mail: devendra.vishwakarma@jaipur.manipal.edu

ABSTRACT

The study focused on improving the performance of a fluidic oscillator through systematic modification of the Coandă surface geometry. The work primarily aimed to increase the oscillation frequency and jet deflection angle, followed by a detailed assessment of the thermal behaviour of the optimized designs in comparison with a reference smooth oscillator. A two-dimensional CFD model was developed to quantify the influence of the selected parameters on individual performance metrics. The oscillator configurations exhibiting superior flow characteristics demonstrated an increase in oscillation frequency and an enhancement in jet deflection angle relative to the baseline case. In the numerical formulation, a pressure inlet and pressure outlet were imposed as the flow boundary conditions, with air used as the working fluid and an inlet temperature of 300 K. The target wall was maintained at a constant temperature of 400 K, while all remaining solid walls were treated as isothermal at 300 K. The distance between the exit nozzle and the target surface was varied at 40 mm, 50 mm, and 60 mm to assess its influence on jet impingement and heat transfer behaviour. The optimized oscillator configurations exhibited improved flow performance, reflected by higher oscillation frequencies and increased jet deflection angles compared to the reference design. Thermal analysis indicated significant enhancement of in the Nusselt number relative to the smooth oscillator.

Key Words: *Fluidic Oscillator, Sweeping Jet, Heat Transfer, Fluid Flow, Nusselt Number*

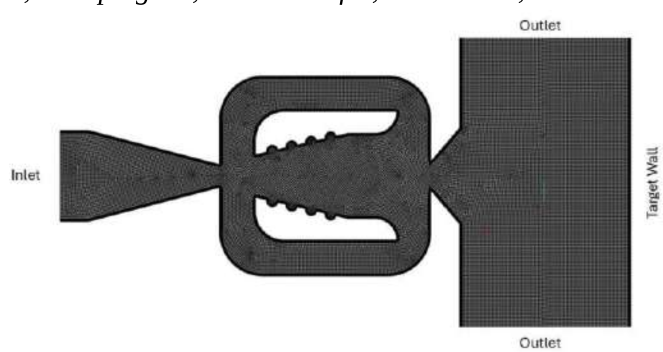


Figure 1: Enhanced surface fluidic oscillator

ENTRAINMENT IN A TURBULENT JET: A NEW APPROACH

Sachin Yashavant Shinde[1] and Prasanth Prabhakaran[1]

[1]Indian Institute of Technology, Kanpur, Uttar Pradesh, India, 208016, E-mail: sachin@iitk.ac.in

ABSTRACT

The mechanism by which ambient non-turbulent fluid is entrained into a turbulent free shear flow has been a subject of interest for a long time. Recent research on free shear flows has focused on whether the dominant mechanism is engulfment or nibbling. We report here results from a DNS study of a round turbulent jet at a Reynolds number of 2400 (based on source diameter *d* and exit velocity *Uo*) with emphasis on the near-field flow in the neighbourhood of the edge of the jet. Figure 1 shows an axial section of the jet with contours of vorticity modulus |□| down to 0.1 (nondimensionalized with mean centerline velocity and local jet half-velocity width). It is found necessary to distinguish between turbulent/non-turbulent (T/NT) and rotational/irrotational (R/IR) interfaces, the two being close together or well-separated at different times and places depending on the location of high |□| regions in the core of the jet (see Figure 1). The organization evident in the near-field velocity can often be correlated with vorticity elements in the coherent structures within the core of the jet: see e.g., sub-regions 2 and 4 in Figure 1, which are at the diametral ends of the toroidal vortex that forms the base of a coherent structure extending over 26 □ *z* □ 38. Entrainment and detrainment are not uniformly distributed in time or space and are found to be best described as a series of intermittent events with a lifetime of order 50 units and spatial extent of about 2/3rd of a quadrant in the diametral plane. An 'inrush event' is generated by the induced velocity field of vorticity element(s) in a nearby coherent structure and culminates in crossing into the turbulent core across a T/NT interface.

Key Words: *Turbulent Jet, Entrainment, Engulfment, Nibbling, Coherent Structure*

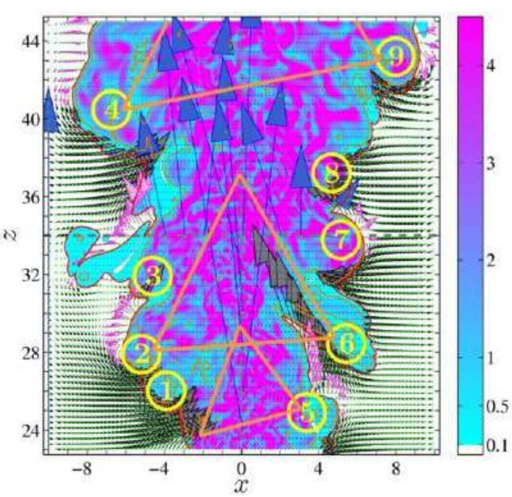


Figure 1: Instantaneous axial section of turbulent jet showing velocity and total vorticity modulus fields. velocity 0.31 *Uo*.

AN INVESTIGATION OF THE SEPARATED FLOW OVER A BLUNT CIRCULAR DISK

Yasar Arafath. U[1], Rajat Kumar Sahu[1], A. C. Mandal[1]

[1]Department of Aerospace Engineering, IIT Kanpur, U.P. 208016, uyasararafath@gmail.com

ABSTRACT

We have carried out a time-resolved particle image velocimetry measurements and large eddy simulations (LES) over a blunt circular disk to study the separated flow characteristics at different circumferential positions. The blunt circular disk was placed parallel to the flow direction, as shown in figure 1. The flow was found to separate at the leading edge of the disk as expected. We focus on the separated flow at the leading edge. From the preliminary analysis, we find that the length of separated regions over the circular disk changes as we move along the circumference, as seen in figure 2. The separated flow reattaches over the surface of the disk, and some elongated streaky patterns are seen in the attached flow region. The proper orthogonal decomposition (POD) in the wall-normal plane reveals dominant shed vortices in the separated regimes. Spatial structures of the first nine modes highlighting the evolution of vortical structures and the dominant energy-containing regions in the flow field. The first 9 modes corresponding to the dominant shedding frequency were extracted from Spectral POD and analyzed. Each frequency corresponds to a specific type of coherent motion, allowing clear identification of energetic flow structures such as vortex shedding, shear-layer oscillations, or recirculation dynamics. The regions of high modal intensity near the disc and in the wake indicate areas of strong flow oscillations and energy concentration

Key Words: *Blunt body, Flow separation, Particle Image Velocimetry, LES.*

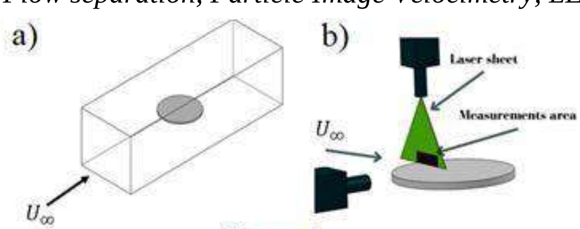


Figure 1

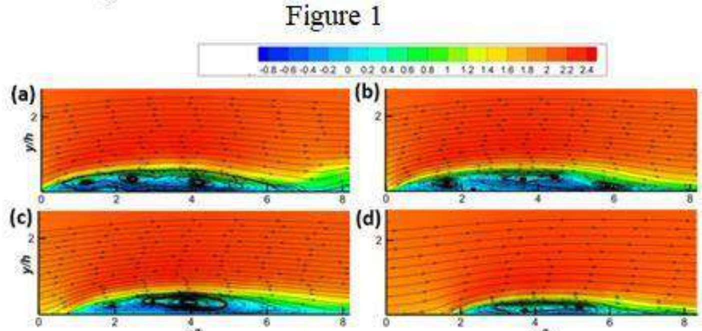


Figure 2

NUMERICAL ANALYSIS OF A VARIABLE PITCH VERTICAL AXIS WIND TURBINE

Nishant Mahamuni[1], Pramath Bhat[1], Sathyabhama A[1] and Ramakrishna N Hegde[2]

[1]Department of Mechanical Engineering, NITK Surathkal, Mangaluru-575025, India, sathyabhama@nitk.edu.in
[2]Srinivas University, Institute of Engineering and Technology, Srinivasnagar, Mukka, Mangalore-574146, rkhegderk@gmail.com

ABSTRACT

Traditional Vertical Axis Wind Turbine (VAWT) offers several advantages for urban environments, including simple design, omni-directional operation, and low cut-in wind speeds. However, they commonly exhibit low starting torque and poor self-starting behavior. Blade pitch angle is one parameter that may help improve VAWT performance. Although previous studies have examined the influence of pitch angle, they have typically relied on lower-fidelity models such as Double Multiple Stream Tube model (DMST) and focused on high wind speeds. Hence, in the present work, CFD simulations are conducted to evaluate the effect of preset blade pitch angles on the performance of the VAWT and to determine the optimal pitch angle. A three-bladed H-Darrieus VAWT with a NACA 0021 airfoil at zero pitch angle is used as the baseline configuration. Simulations are performed for pitch angles of 14°, 16°, and 18°, across tip speed ratios (TSR) ranging from 1.5 to 3.5, at a wind speed of 10 m/s. The numerical simulations revealed that 16° pitch amplitude produced better results in terms of Power Coefficient (C_P) and Torque Coefficient (C_T) as depicted in figures 1 and 2. The maximum C_P of the VAWT with blade pitching was observed to be 0.325 at a TSR of 2.5, with an overall average increase in efficiency of 23.6% compared to the VAWT without blade pitching. The aerodynamic gains were further supported by velocity contour and streamline visualizations, which showed less wake asymmetry, improved flow attachment, and lower separation losses for the variable pitch case.

Keywords: *VAWT, Variable Pitch, Power coefficient, Torque Coefficient*

NUMERICAL STUDY OF LID DRIVEN CAVITY WITH SLIP BOUNDARIES USING FINITE VOLUME METHOD

Likhitha A, Greeshma P K, Suyash Bhadoyuria and Ranjith Maniyeri*
Biophysics Lab, Department of Mechanical Engineering, National Institute of Technology Karnataka (NITK), Surathkal, Mangalore- 575 025, Karnataka, India * Corresponding author: mranji1@nitk.edu.in

ABSTRACT

The lid driven cavity is a classical benchmark problem in fluid dynamics. However, at the microscale the conventional no-slip boundary condition no longer provides an accurate description of wall-fluid interactions. This work provides detailed comparisons of single and dual lid cavity configurations under Maxwell's slip model to capture effects of microscale conditions, supported by a comprehensive accommdation coeffient (σ)–Reynolds number (Re) parametric study, offering new benchmark data for slip flow at Knudsen number 0.01. In this study, the square lid driven cavity flow problem has been simulated with slip boundaries using an explicit scheme based finite volume method (FVM) by developing a code in MATLAB. The code is validated for the case of no- slip condition. The model is then extended to incorporate Maxwell's slip boundary condition, where the tangential velocity at the wall is related to the velocity gradient through σ and the Knudsen number (Kn). The value of σ is varied between 0.01 and 1 to represent different degrees of slip. Simulations are performed for Re ranging from 1 to 300 for both single and dual lid driven cavity configurations. In the dual lid cavity case, both parallel and antiparallel lid motions are analysed to study the combined influence of wall motion and slip effects on the flow structure. The results show a strong dependence of flow characteristics on σ. As σ decreases, the secondary vortices near the corners weaken, and the primary vortex core shifts toward the geometric centre of the cavity. The overall circulation strength, peak velocity and wall shear reduce correspondingly. For a typical value of Re = 10, reducing σ from 1 to 0.01 lowers the maximum u velocity from 1(no-slip case) to
0.075 (single lid), 1 to 0.067 (parallel), and 1 to 0.094 (anti-parallel). The single lid vortex shifts 0.17 toward the geometric centre, the parallel lid vortex shifts 0.11 toward its half cavity centre, while in the anti parallel case the two recirculation zones collapse into a single symmetric vortex at the centre. This demonstrates the significant role of wall slip in microfluidic flows. The study confirms that even small degrees of slip can substantially alter internal flow patterns, emphasizing the importance of incorporating Maxwell's slip boundary condition when modelling microscale cavity flows.

Key Words: *Accommodation coefficient, Finite volume method, Lid driven cavity, Reynolds number, Slip boundary.*

STUDIES ON RESPONSE DELAY FOR THERMOCOUPLE MOUNTED THERMOWELL IN NON-ISOTHERMAL SODIUM FLOW

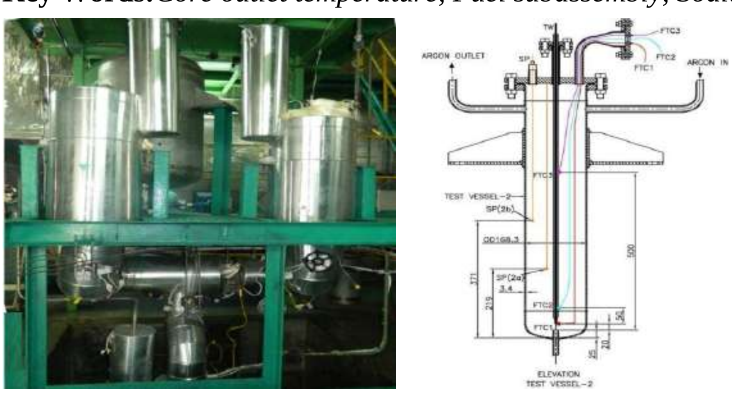
Ch.S.S.S.Avinash*, Avinash Kumar Acharya, B.Malarvizhi, E.Hemanth Rao, Sanjay Kumar Das and S. Chandramouli

Safety Engineering Division, Fast Reactor Technology Group,IGCAR,Kalpakkam-603102, *chavin@igcar.gov.in

ABSTRACT

In a typical sodium cooled fast reactor (SFR), the core outlet temperature is monitored by a set of thermowell probes positioned at the outlet of each fuel sub-assembly (FSA).The information is fed to the Core Temperature Monitoring Systemfor initiation of safety action, in case of abnormality.In general, to have redundancy and to avoid common mode failure, each such probe consists of two thermocouples (PTCs) inserted into a thermowell positioned at ~ 120 mm above the corresponding fuel subassembly head. However, in case of incomplete insertion of the probe into the thermowell, the response time of the PTCs may be affected, resulting in delayed indication for sudden temperature transients. Towards this, for the first time an experimental study has been taken up at IGCAR, to determine the response time of the thermocouple probe with reference to gap between the PTC and thermowell tip in a dynamic sodium loop. Experiments were carried out for three cases (0, 5 & 10 mm gap) for various temperature transients. This was achieved by sending liquid sodium at higher temperature to create transients at the probe tip, which is initially at ~ 550 ºC. Upstream sodium flow velocity at the tip of the thermocouple probe was maintained at around 3m/s similar to that of a typical SFR FSA outlet velocity by constricted passage and differential pressure technique, while maintaining the flow for about 35s to 40s. The response of PTCs was compared with 'K' type reference TC positioned adjacent to the probe for determining the delay in the probe, which was observed to be within the allowable limits for all the gap configurations. Details of the experiment along with important results are discussed in this work.

Key Words:*Core outlet temperature, Fuel subassembly, Sodium cooled fast reactor*



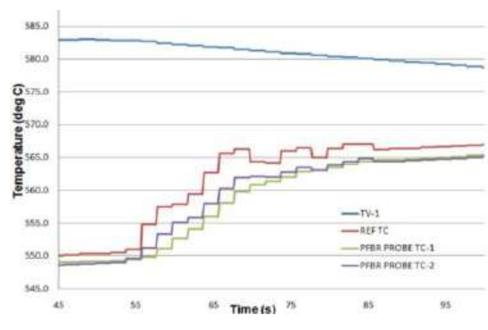


Figure1: Experimental facility with internal details recorded during the flow across the probe

Figure2: Typical temperature plot of probe inserted test vessel

INVESTIGATION OF IMMERSION COOLING EFFECTS ON THE THERMAL BEHAVIOUR OF HIGH-DISCHARGE LITHIUM-ION BATTERY PACKS

Swagata Gupta[1,2], Bittagopal Mondal[1,2], Chanchal Loha[1,2], Ishita Sarkar[1,2]

[1]Energy Research and Technology Group CSIR-Central Mechanical Engineering Research Institute, Durgapur-713209, India

[2]Academy of Scientific and Innovative Research Ghaziabad-201002, India

ABSTRACT

Effective thermal management is essential for ensuring the performance, safety, and longevity of lithium-ion batteries, whose electrochemical behavior is highly sensitive to temperature. Immersion cooling has recently gained attention as a superior alternative to conventional air and indirect liquid cooling due to its high heat-removal capability, low noise, and excellent temperature uniformity. This study experimentally evaluates the thermal performance of a 20 Ah battery module composed of 21700 cylindrical lithium-ion cells under three cooling strategies: natural air cooling and singlephase immersion cooling using two dielectric fluids—natural ester and silicone oil. Discharge rates of 0.5C, 1C, 1.5C, and 2C were tested, with systematic variation in coolant viscosity and flow rate. Key performance indicators included maximum cell temperature, temperature distribution, and spatial temperature gradients. Optimal flow rates were identified by minimizing coolant temperature rise between inlet and outlet. The results show that both immersion-cooling approaches significantly outperform natural air cooling, with silicone oil providing the greatest thermal benefit. Relative to air cooling, silicone oil reduced the maximum cell temperature by 25.22%, 31.08%, 33.92%, and 30.58% at 0.5C, 1C, 1.5C, and 2C, respectively, and improved temperature uniformity by 27.20% compared with air cooling and 30.97% compared with natural ester. Overall, the findings demonstrate that silicone-oil immersion cooling is a highly effective strategy for enhancing thermal safety and performance in high-power lithium-ion battery systems.

Key Words: Immersion Cooling, 21700 Li-Ion Cell, Dielectric liquid, Flow Rate optimization, Natural Ester

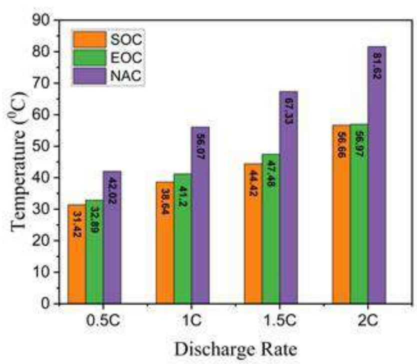


Figure 1. Maximum temperature of the battery pack at various discharge rates under different cooling techniques

COUPLED NUMERICAL AND ANN-BASED FRAMEWORK FOR ENTROPY ANALYSIS IN MHD FLOW OF HYBRID NANOFLUIDS THROUGH POROUS MEDIA

[1]D. Krishnan, and [1]P. Durgaprasad
[1,2]Department of Mathematics, SAS, Vellore Institute of Technology, Chennai-600127, India.
Corresponding author: durgaprasad.p@vit.ac.in

ABSTRACT

The work studies how the magnetohydrodynamic (MHD) boundary layer flow and entropy generation are carried out in monofluid (Au/water) and hybrid nanofluid (Au–TiO_2/water) over a wedge embedded in a Darcy–Forchheimer porous medium under linear thermal radiation. The main equations are simplified to a set of linked ordinary differential equations through similarity transformations and are later solved numerically using MATLAB's bvp4c solver. It is found that the increase in magnetic field strength causes the fluid velocity to rise and due to the Lorentz force, the momentum boundary layer to become thicker. With the help of enhanced permeability, flow resistance is reduced. The temperature drops with increased magnetic effects, but it goes up with higher Brinkman numbers and radiation intensities, thus, allowing the viscous heating and radiative flux to play their roles. As compared to the monofluid, the hybrid nanofluid exhibits better thermal performance and lesser entropy generation, due to the higher effective thermal conductivity. The Bejan number gets lower as the viscous and magnetic effects become stronger, thus, indicating that there is a move from thermal to frictional irreversibility. Sensitivity analysis reveals that magnetic field strength and permeability are the main factors that have a major influence on the skin friction coefficient and Nusselt number, whereas radiation and Brinkman effects have a moderate influence.

Novelty: The idea of combining MHD flow, porous media resistance, thermal radiation, and machine learning into one model is a major novelty of this work. An artificial neural network (ANN) driven by data and trained on numerical results efficiently accomplishes the task of wall heat transfer and shear stress parameters, thus, it is not necessary to perform a large number of computationally expensive simulations. This study not only provides a single framework for comparing mono and hybrid nanofluids under the same conditions but also gives practical insights into the enhanced thermal management in porous geometries, which is totally different from previous research that investigates these effects separately.

Keywords: *Hybrid nanofluid; MHD; Porous medium; Darcy–Forchheimer flow; Thermal radiation; Entropy generation; ANN model.*

ARTIFICIAL NEURAL NETWORK ANALYSIS OF ENTROPY GENERATION IN EYRING-POWELL TERNARY HYBRID NANOFLUID OVER A MAGNETISED POROUS VERTICAL CONE

[1]M. Bhuvana and [1*]P. Durgaprasad
[1,2]Department of Mathematics, SAS, Vellore Institute of Technology, Chennai-600127, India ,
Corresponding author: durgaprasad.p@vit.ac.in

ABSTRACT

This study investigates the magnetohydrodynamic boundary-layer transport of an Eyring–Powell ternary hybrid nanofluid, with blood as the base fluid, over a porous vertical cone, taking into account buoyancy, thermal radiation, and internal heat generation or absorption. Similarity variables reduce the model to ordinary differential equations; MATLAB's bvp4c solver then handles the numerical solution. We examine the Bejan number and entropy generation using 3D surface plots. We also show the skin-friction coefficient and the local Nusselt number for a range of physical parameters. The thermal and momentum aspects are strongly dependent on physical parameters, as shown in the numerical results. While the porosity raises the momentum layer by about 31%, an increase in the magnetic parameter causes the velocity to drop by 21% for HNF and 23% for THNF. The heat source of the thermal field, radiation, is increasing the temperature by 17.6% (HNF) and 15.1% (THNF). The heat-transfer rate increases by 23-24% due to magnetic forces, and the thermal boundary layer thickens due to increased heat generation, resulting in a temperature rise of 27-29%. A similar heat sink, however, decreases the temperature by 18-19%. Entropy generation increases with the Brinkman number, while the Bejan number decreases, confirming that viscous dissipation becomes the dominant contributor to thermodynamic irreversibility. THNF consistently shows a marginally stronger thermal response than HNF, driven by enhanced ternary nanoparticle synergy. A feed-forward Artificial Neural Network (ANN) is trained on the numerical dataset to predict the skin-friction coefficient and local Nusselt number. The ANN achieves $R \approx 0.999$ with very low mean-square errors, demonstrating excellent agreement with the numerical solutions and reliable nonlinear mapping capability. The findings are beneficial for biomedical applications, nanofluidic cooling, and compact energy conversion technologies.

Keywords: *Eyring-Powell, ternary hybrid nanofluid, MHD, porous medium, thermal radiation, entropy generation, and ANN model.*

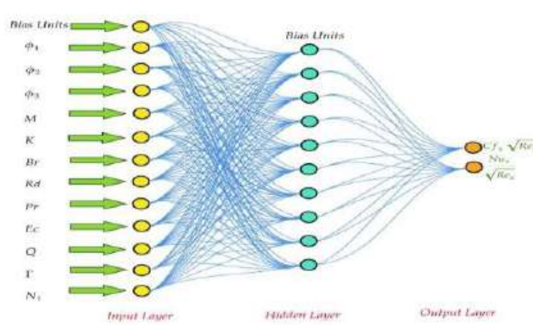


Figure 1: An ANN design with several layers for skin friction and Nusselt number

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

NUMERICAL ANALYSIS OF HEAT TRANSFER ENHANCEMENT IN MICROCHANNEL HEAT SINKS WITH CIRCULAR Y-CHANNEL RIBS

Mithun Das[1*], Nirmalendu Biswas[2], Dipak Kumar Mandal[3], Nirmal K. Manna[4], Aparesh Dutta[5], and Suvanjan Bhattacharyya[6]

[1]School of Nuclear Studies and Application, Jadavpur University, Kolkata 700106, India
mdas190@gmail.com
[2]Department of Power Engineering, Jadavpur University, Kolkata 700106, India
biswas.nirmalendu@gmail.com
[3]Department of Mechanical Engineering, Government Eng. College Samastipur, Bihar, India
dipkuma@yahoo.com
[4]Department of Mechanical Engineering, Jadavpur University, Kolkata 700032, India
nirmalkmannaju@gmail.com
[5]Department of Mechanical Engineering, NIT Durgapur, West Bengal, 713209, India
adatta96@gmail.com
[6]Department of Mechanical Eng., Birla Institute of Technology and Science Pilani, Rajasthan, India
suvanjan.bhattacharyya@pilani.bits-pilani.ac.in

ABSTRACT

This study numerically examines the thermal performance of a microchannel heat sink with circular Y-channel ribs under various geometric and operating conditions. The effects of three main parameters, namely the rib front opening angle (θ), rib leg angle (ϕ) (see **Fig. 1**), and applied heat flux, are analyzed using the finite volume method for Reynolds numbers ranging from 92 to 735. Results show that increasing θ while keeping ϕ = 90° reduces the friction factor by about 8 percent, with the optimal θ of 45° providing the highest thermal performance. The influence of ϕ indicates that both lower and higher leg angles result in reduced heat transfer enhancement, and the best performance occurs at ϕ = 90°. At this configuration (θ = 45°, ϕ = 90°), the maximum thermal performance factor of 1.20 is achieved at a heat flux (q_w) of 50 W/cm². Increasing heat flux further improves heat transfer but introduces higher pressure losses. Velocity and temperature contour analyses reveal stronger fluid mixing and vortex formation downstream of the rib legs. The results demonstrate that circular Y-channel ribs effectively enhance heat transfer, offering a promising approach for compact electronic cooling applications.

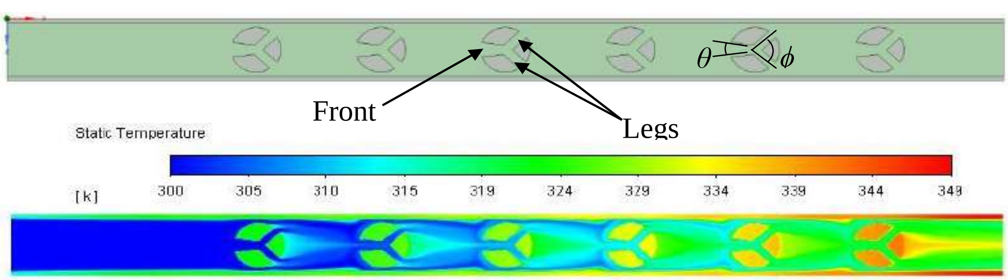


Figure 1: Schematic of the problem geometry and temperature contour at the mid-plane of the channel (θ = 15°, ϕ = 90°, Re = 459 and q_w = 50 W/cm²)

HEAT TRANSFER STUDY OF MAXWELL FLUID OVER A VERTICALLY MOVING CIRCULAR SURFACE USING ARTIFICIAL NEURAL NETWORK

Amit Yadav, Kushal Sharma

Department of Mathematics, MNIT Jaipur, Rajasthan, India, amit.ydv9097@gmail.com

ABSTRACT

The present study aims to investigate the slip flow of Maxwell fluid on a vertically moving rotating disk. The effect of activation energy, and viscous dissipation is also considered. The temperature-dependent viscosity is incorporated to more accurately capture the thermally driven variations in fluid rheology, ensuring a realistic prediction of flow. The governing nonlinear partial differential equations are transformed into the non-linear ordinary differential equations using the similarity transformations. The effect of physical parameters is analysed by plotting the graph of such parameters on the velocity, thermal, and concentration profiles. The artificial neural network (ANN) approach is employed to predict the drag and heat transfer over the disk. A feed forward neural network based on Levenberg-Marquardt algorithm is employed to develop a relation between input and output parameters. The results indicate a decrease in the radial and tangential velocities with the viscosity variation parameter and the reverse effect can be seen on axial velocity. The results predicted by ANN are also validated against the numerical results computed by BVP Midrich scheme in MAPLE software. The novelty of the current study is the Maxwell fluid flow with vertical moving, activation energy, and viscous dissipation effects on flow on rotating disks. To the best of our knowledge, no such consideration has been published in the literature.

Key Words: *Activation Energy, Viscous Dissipation, Artificial Neural Network.*

EVALUATION OF PRESSURE LOADS AND ACOUSTIC RESPONSE OF LAUNCH PAD STRUCTURES UNDER SUPERSONIC WALL JET IMPINGEMENT DURING LIFT-OFF

S. Sankaran[1], G. Venkatesh[1], M. Deepthi[1], B. N. V. S. Aditya[1], K. V. Kali Prasad[1], V. V. Ramakrishna[1]

[1]Satish Dhawan Space Centre (SDSC) SHAR, ISRO, Sriharikota, India - 524124

E-mail: kvkprasad@shar.gov.in

ABSTRACT

Launch vehicle lift-off generates extremely severe fluid-dynamic and acoustic environments that govern the design of launch pad and propellant servicing structures. With the reconfiguration of an operational launch vehicle, additional Propellant Service Structures (PSS) were introduced on the launch pedestal, necessitating a reliable estimation of pressure and acoustic loads during lift-off. Owing to the limitations of numerical simulations in capturing highly unsteady supersonic jet– structure interactions, an experimental investigation using geometrically scaled sub-scale models remains the most dependable approach. In the present study, a 1:100 geometrically scaled cold-flow experimental campaign is carried out to quantify wall-jet-induced pressure loads and lift-off acoustics on newly introduced PSS towers. Supersonic jet impingement corresponding to Mach 3.92 (core jet) and Mach 3.38 (strap-on jets) is simulated using cold nitrogen as surrogate gas while matching nozzle exit pressure and area ratio. Two realistic lift-off scenarios are examined: (i) core jet with both strap-on jets operating simultaneously, and (ii) strap-on jets operating alone. Wall static pressures are measured along the height of the PSS at a sampling rate of 500 Hz, while acoustic measurements are performed along the launch vehicle and umbilical tower to identify overall sound pressure levels (OASPL) and spectral characteristics with 40 kHz sampling rate. The results indicate that peak wall pressures occur within $L/De \approx 6–8$, with maximum nondimensional pressure coefficients being significantly higher (by approximately 25–35%) when the core jet is active due to enhanced entrainment and stronger wall-jet formation. The presence of the umbilical tower introduces asymmetry in the flow field, leading to measurable pressure differences between the PSS located toward and away from the tower. Cold-flow acoustic measurements reveal a dominant tonal component near 3 kHz, particularly in the vehicle base region. However, confirmatory 1:12.8 scale hot-flow tests, matching exhaust temperature and thermodynamic properties, demonstrate the absence of such tones, establishing that the observed tones are prevalent in cold-flow scale tests and do not exist for the actual flight condition. The novelty of this work lies in the first integrated experimental characterization of wall-jet pressure loads and lift-off acoustics on newly introduced propellant service structures under realistic multi-jet launch conditions. By simultaneously resolving pressure loading, acoustic response, and surface flow topology for core–strap-on jet combinations, and by validating cold-flow acoustic tonal observations through matched hot-flow testing, the study delivers a configuration-specific, experimentally verified load envelope that directly supports reliable and non-conservative launch pad structural design.

Key Words: *Supersonic jet impingement, Wall jet pressure, Lift-off acoustics, Launch pad structures, Sub-scale experiments*

THERMAL MANAGEMENT OF LITHIUM-ION BATTERIES USING COOLING PLATES

Purab Sharma, Ashwinkumar S. Dhoble, Someshwar S. Bhakre, Shriram S. Ughade, Akshay Shewalkar
Department of Mechanical Engineering, Visvesvaraya National Institute of Technology, Nagpur, Maharashtra, India, [1]purab2903@gmail.com

ABSTRACT

This study presents the design and evaluation of a cooling plate-based system for cylindrical 18650 Samsung 35E cells arranged in a 2p 4s configuration, to form a battery pack. A total of four hollow flat aluminium cooling plates (6 mm total thickness) are strategically positioned on either side of the cells, providing direct thermal contact via high-conductivity (12 W/mK) silicone pads. Three situations of cooling plates are tested without and with cooling the battery pack. The plates are internally filled with a coolant mixture consisting of 100% distilled water and another coolant combination of 90% distilled water and 10% ethylene glycol by volume, selected for its thermal properties, which helps in increasing boiling and freezing point of the coolant fluid. The mode of cooling is active as a pump is used for coolant circulation. The battery pack is subjected to different C-rating charging and discharging. The internal resistance of the cells led to a peak heat generation of approximately 1.16 W per cell during the 2C discharge rate. Mathematical modelling results indicate, that the cooled cells charged at 2C rate highest temperature is around 338 °K, whereas the uncooled cells highest temperature is around 340 °K for same C rate. An experimental setup is also prepared on the proposed cooling plate architecture. Results of this setup shows the peak temperature without cooling at 2C discharge rate around 344 °K, whereas in cooling case of distilled water peak temperature around 336 °K. These results also shows that the 100% distilled water compared to 90% distilled water and 10% ethylene glycol mixture temperature difference is nearly equal but the properties of ethylene glycol mixed with distilled water favour arid climate regions. Modular pack configurations that require tight temperature uniformity across cells are particularly well-suited for the design.

Key Words: *Aluminium Cooling Plate, Silicone Pad, Ethylene Glycol, Heat Transfer, Active Cooling*

ENTROPY GENERATION AND THERMODIFFUSION IN CATTANEO–CHRISTOV MHD BIOCONVECTIVE TANGENT HYPERBOLIC NANOFLUID FLOW WITH ACTIVATION ENERGY

Priyanka Yadav[1], Dr. Santosh Chaudhary[2]

[1]Department of Mathematics, Malaviya National Institute of Technology, Jawahar Lal Nehru Marg, Malviya Nagar, Jaipur - 302017, Rajasthan, India, 2024rma9099@mnit.ac.in

[2]Department of Mathematics, Malaviya National Institute of Technology, Jawahar Lal Nehru Marg, Malviya Nagar, Jaipur - 302017, Rajasthan, India, schaudhary.maths@mnit.ac.in

ABSTRACT

This study provides a comprehensive theoretical investigation of magnetohydrodynamic (MHD) bioconvective flow in a tangent hyperbolic nanofluid containing gyrotactic microorganisms over an inclined porous Riga plate. Thermal relaxation effects are incorporated through the Cattaneo–Christov heat flux model, allowing for finite-speed thermal propagation, while cross-diffusion phenomena (Soret and Dufour effects) and activation energy are included to account for realistic temperature-dependent solute transport. The governing nonlinear partial differential equations for momentum, thermal energy, concentration, and microorganism density are transformed into a set of coupled ordinary differential equations via similarity transformations. This formulation simultaneously captures shear-thinning rheology, electromagnetic forcing from the Riga plate, porous medium resistance, buoyancy-driven interactions, nanoparticle migration, and microorganism motility.

The resulting system is solved numerically using MATLAB's bvp4c, which efficiently handles coupled boundary-value problems and ensures accurate, stable solutions through adaptive mesh refinement. The developed model provides a robust framework for analysing entropy generation, thermo-solute coupling, and bioconvective transport in non-Newtonian bio-nanofluid systems. The insights from this study are broadly applicable to microfluidic heat and mass transfer, energy-efficient cooling technologies, biomedical engineering, and industrial processes involving complex rheological fluids with active microorganisms.

Key Words: Bioconvection, Tangent Hyperbolic Nanofluid, Entropy Generation, Riga Plate, MHD

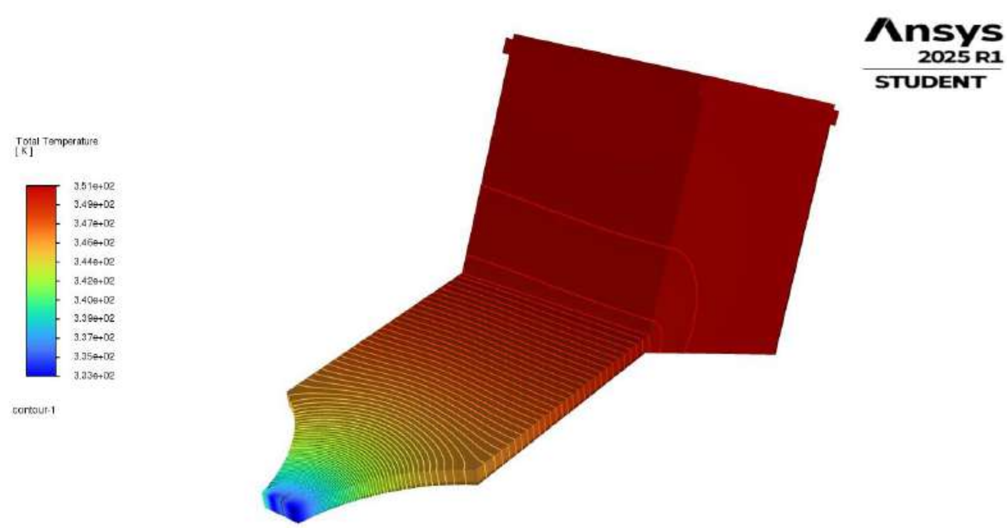
NUMERICAL STUDIES ON THERMAL PERFORMANCE OF SOLAR AIR DRYER USING NEW DRYING CHAMBER

Bathula Mahesh, Praveen D Sawarkar, Someshwar S Bhakre
Department of Mechanical Engineering, Visvesvaraya National Institute of Technology, Nagpur-440010, Maharashtra, India, [1]maheshmai2030@gmail.com

ABSTRACT

This Study presents design and analysis of a Solar Air Dryer to evaluate its thermal performance. A three-dimensional (3D) numerical investigation of a solar air dryer is done to evaluate the temperature and flow characteristics of air in a solar air dryer. The 3D model is created and simulated in ANSYS Fluent to observe the temperature and airflow within the drying chamber. The system consists of a diffuser to uniformly distribute the air over air heater, an air heater, and new drying chamber made of acrylic sheet while the structure of the chamber is made of steel for holding the walls of the chamber. The air that entered will increase its temperature passing through air heater and the radiation passing from sheets aid in rise of temperature inside the chamber. The simulation results showed the highest temperature of about 351 K, which shows efficient solar energy absorption and even heat distribution in the drying chamber. The airflow contours exhibited smooth flow and less stagnation areas, resulting in improved drying capacity.

Key Words: *Solar Air Dryer, ANSYS Fluent, Drying chamber, Thermal Performance, Diffuser*



EFFECT OF PIPE LENGTH ON HEAT TRANSFER ENHANCEMENT BY JET IMPINGEMENT

Kuldhir Singh Bhati[1]

[1]Mechanical Engineering Department, IIT Indore, Khandwa Road, M.P. - 453552, phd2301103011@iiti.ac.in

ABSTRACT

A turbulent jet flow is known for its enhancement and mixing applications in various engineering applications. In cooling systems utilizing compact geometries, such as turbine blade cooling, a shorter pipe is desirable while still achieving a higher average Nusselt number. The existing literature predominantly addresses the heat transfer characteristics of jets issuing from long nozzles, where the flow is assumed to be fully developed at the exit. This study numerically investigates the effects of the pipe length (L) at various stand-off distances (H/D) on the average Nusselt number (Nu_{avg}) of a circular impinging jet. The k-ω SST gamma-Re_θ transitional turbulence model is employed for a three-dimensional round jet simulation. Numerical simulations are conducted using the commercial software ANSYS Fluent at a Reynolds number (Re = 10000) for four different pipe lengths (L = 20D, 40D, 80D, 100D). Results show that at higher H/D ratios, the average Nusselt number is higher for smaller L/D ratios. Such as, at H/D = 6, the average Nusselt number is 8.85% higher for a pipe length of L/D = 80 compared to L/D = 100, as illustrated in Figure 1. This is because the flow is in a transitional state at L/D = 80, while the flow becomes fully developed at L/D = 100.

Key Words: *Heat Transfer, Jet Impingement, RANS, CFD, pipe length.*

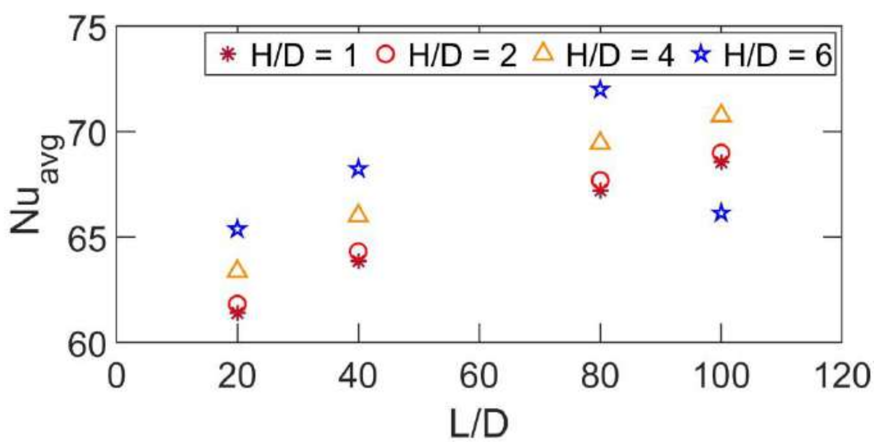


Figure 1: Variation of average Nusselt number with pipe length (L) at different stand-off distances (H/D)

STUDY OF A WASTE HEAT RECOVERY SYSTEM USING THERMOELECTRIC GENERATOR WITH POWER CONDITIONING

Lakshmisha Madhusudhana[1], Koshal Ram KS[1], Aaditya A[1], Dr.Venkatesh Lamani[1]
Corresponding author : venkateshtl.mech@gmail.com
[1] Department of Mechanical Engineering
B.M.S. College of Engineering Bengaluru 560019, India, venkateshtl.mech@gmail.com

ABSTRACT

Efficient utilization of low-grade waste thermal energy is critical for improving overall system efficiency. This study focuses on harnessing waste heat via a Thermoelectric Generator (TEG) system coupled to a DC/DC power conditioning circuit for Li-ion battery charging. The primary novelty of this work lies in the empirical, multi-variable optimization of the TEG system's thermal management chain. Research quantify the coupled effect of Heat Exchanger Material by using aluminium and copper plate for hot air with flow rate (5 m/s, 7 m/s and 10 m/s) on the instantaneous voltage output. Experimental results demonstrate that lower-conductivity aluminium yields a high peak voltage at a lower temperature (approx. 3.1V), but quickly saturates**.** At the same time, higher-conductivity copper provides a more stable, linear response and a higher sustained maximum output (approx. 3.2V). Furthermore, a parametric study was conducted to analyse the open-circuit output voltage in series and parallel configurations of the TEG array. This confirmed the series connection as the valid topology for our experiment with copper plate.

Key Words: *Seebeck effect, Waste heat recovery, TEG, Power generation, Energy Storage*

NUMERICAL STUDY OF MIXING IN MISCIBLE FLOWS THROUGH POROUS MEDIA

Manoj[1], Manoranjan Mishra[2] and Navaneeth K M[1]
[1]Department of Mechanical Engineering, Indian Institute of Technology Ropar, Rupnagar 140001, Punjab, India, manoj.21mez0023@iitrpr.ac.in, navaneeth@iitrpr.ac.in [2]
Department of Mathematics, Indian Institute of Technology Ropar, Rupnagar 140001, Rupnagar 140001, Punjab, India, manoranjan@iitrpr.ac.in

ABSTRACT

When fluid with distinct physical properties, such as density and viscosity, enters a saturated porous medium, hydrodynamic instabilities can develop, giving rise to finger-like patterns. These fingers enhance the interfacial surface area between the fluids, thereby promoting mixing. Such mixing is undesirable in applications such as underground hydrogen storage and enhanced oil recovery, but can be beneficial in processes such as CO_2 sequestration and in-situ remediation of contaminated aquifers. In this work, we investigate the problem of mixing in rectilinear, incompressible, miscible flow through a two-layer porous medium, considering the effects of gravity. The Darcy scale governing equations are formulated in terms of the vorticity–stream function representation, rather than the primitive variables (velocity and pressure). Both viscosity and density are modelled as functions of solute concentration. The governing equations are solved using the finite element method implemented in COMSOL Multiphysics. Different permeability configurations of two-layer systems are examined in the presence of gravity to quantify their influence on the mixing. A representative two-layer system is illustrated in Figure 1, where the permeability of the upper layer is twice that of the lower layer. The results of the present study illustrate that mixing can be effectively controlled by adjusting the density contrast, Peclet number, and permeability distribution within the system. Furthermore, the degree of mixing is quantified as a function of the density contrast, permeability contrast, Peclet number, and viscosity ratio in two-layer systems.

Key Words: *COMSOL Multiphysics, Gravity effects, Mixing, Porous Media, Viscous Fingering.*

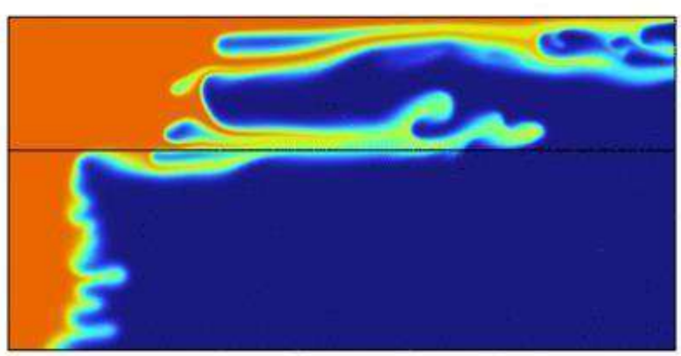


Figure 1: Concentration field showing solute distribution, where red denotes $c=1$, and blue denotes $c=0$. The injected fluid has $c=1$, while the resident fluid has $c=0$

ANALYSIS OF HEAT AND MASS TRANSFER IN MAXWELL NANOFLUID FLOW UNDER MAGNETIC DIPOLE

Nisha Chouhan[1], Reema Jain [1]

[1]Department of Mathematics and Statistics, Manipal University Jaipur - 303007, Rajasthan, India
E-mail address: reema.jain@jaipur.manipal.edu, nisha.202505041@muj.manipal.edu

ABSTRACT

This investigation rigorously examines the magnetohydrodynamic (MHD) heat and mass transport phenomena in a non-Newtonian Maxwell nanofluid under the influence of a magnetic dipole field over an exponentially stretching surface, incorporating motile gyrotactic microorganisms. The novelty of this study resides in its unprecedented exploration of the synergistic interplay among Maxwell fluid viscoelasticity, magnetic dipole induced Lorentz forces, and microorganism bioconvection, elucidating coupled magneto-viscoelastic and bio-convective transport mechanisms that remain largely unexplored in the extant literature. A robust mathematical framework is constructed via the governing equations of momentum, energy, nanoparticle concentration, and microorganism density. By implementing judicious similarity transformations, the nonlinear partial differential equations are transmuted into a coupled system of ordinary differential equations, subsequently resolved with MATLAB's bvp4c solver to ensure numerical precision and computational stability. Benchmark validation demonstrates deviations below 0.5%, corroborating the fidelity of the proposed model. Parametric analyses reveal that an augmented Maxwell relaxation parameter diminishes the primary velocity gradient by 12–18%, manifesting pronounced elastic resistance. Enhanced Brownian motion and thermophoretic effects elevate the temperature by 10–15%, thereby amplifying the thermal boundary layer. Nanoparticle concentration attenuates with increased Brownian diffusion, whereas thermophoresis intensifies solutal transport. Furthermore, the Peclet number and bioconvection parameters markedly modulate microorganism distribution, augmenting the bioconvection layer thickness by 8– 14%.The findings impart profound insights into the coupled dynamics of magneto-viscoelasticity, nanoparticle kinetics, and microbial motility, offering pivotal guidance for the optimization of advanced cooling systems, biomedical devices, and microfluidic applications employing magnetized non-Newtonian nanofluids.

Keywords: *Magnetohydrodynamics, Maxwell nanofluid, magnetic dipole, exponentially stretching sheet, motile gyrotactic microorganisms.*

DESIGN AND PERFORMANCE ASSESSMENT OF IN-SITU RESIN EXTRACTION SCHEME FOR ION EXCHANGERS OF SPENT FUEL STORAGE BAY OF PFBR USING FLUIDIZATION TECHNIQUE

Pritam Kumar Patel[1], V Krishnamurthy, R Nandakumar, B Anoop, Amzad Pasha, Partha Sarathy Uppalla
Indira Gandhi Centre for Atomic Research (IGCAR), Kalpakkam-603102, India. Email: pritam@igcar.gov.in

ABSTRACT

Prototype Fast Breeder Reactor (PFBR) is a 500 MWe liquid sodium cooled reactor located at Kalpakkam, Tamil Nadu. The Spent Subassembly Storage Bay (SSSB) of the PFBR is provided for the interim storage of sodium cleaned spent fuel subassemblies discharged from the reactor core to water pool. It is equipped with an independent cooling and purification systems to remove decay heat from the spent fuel subassemblies and for control of radioactivity and chemical parameters of bay water. The purification system includes mixed-bed ion exchangers (IOX), each containing 500 L of nuclear-grade anion and cation resins, which remove dissolved ionic contaminants from the bay water. Conventionally, replacement of the radioactive resin was planned by transporting the entire IOX column to the Waste Immobilization Plant (WIP). However, due to site constraints, this approach was found to be impractical. To overcome these limitations, an innovative in-situ ionexchanger resin fluidization and transfer scheme was developed and demonstrated. The proposed method enables batch wise extraction of radioactive resin (70 L per batch) from the IOX located 11 m below ground level to a shielded hopper drum positioned near entrance of the building at ground level. The scheme utilizes the existing purification circuit pump along with a water tank (~1500 L capacity), quick-connect piping, sight glass tubes, pressure and flow measuring instruments. During operation, water is circulated in reverse mode through the IOX to fluidize the resin bed and transport the resin-water mixture to the hopper drum, where the resin is retained using a strainer arrangement. The process is carried out in batches to transfer the entire 500 L resin inventory. Experimental demonstrations were performed with 140 L and 70 L hopper drums to evaluate the hydraulic performance, establish pressure-drop correlations, and confirm the feasibility of using the existing purification pump. The resin transfer scheme was successfully demonstrated at controlled flow rates. At higher flow rates (~15 m^3/h), rapid transfer led to pipeline choking, while stable operation was achieved at 2–5 m^3/h. The overall system pressure drop was within acceptable limits and it is majorly contributed by the hopper drum. A sudden rise in inlet pressure of the hopper drum was identified as a reliable indicator to terminate the resin transfer operation. It is additionally supported by pump discharge flow and pump input current trends. The study confirmed the technical feasibility and radioactive safety of the innovative in-situ resin fluidization approach utilizing the existing system components. The established pressure-drop correlations and operational parameters provided important design inputs for the permanent system. The demonstrated scheme offers a safe, cost-effective, reliable and remotely operable solution for replacement of radioactive resins from the IOX units of the water purification systems used in spent fuel storage bay for PFBR.

Key Words: *Spent fuel storage bay, radioactive water purification, ion exchanger, resin fluidization, in-situ resin transfer, hopper drum*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

LOCALIZED BED COOLING FOR SLEEP COMFORT: A SUSTAINABLE ALTERNATIVE TO ROOM AIR CONDITIONING

Atharva Sharma[1], Atharv Srivastava[1], Kartikey[1], Navaneeth K Marath[1,2]

[1]Department of Mechanical Engineering, Indian Institute of Technology Ropar, Punjab, India. Email: 2022MEB1302@iitrpr.ac.in , 2022MEB1301@iitrpr.ac.in , 2022MEB1320@iitrpr.ac.in , navaneeth@iitrpr.ac.in

[2]Centre of Research for Energy Efficiency and Decarbonization (CREED), Indian Institute of Technology Ropar, Punjab, India

ABSTRACT

Conventional room air-conditioning cools the entire room volume even though sleep comfort is governed mainly by the thermal microclimate near the body. Recent commercial products hint at the potential of localized bed cooling, but systematic engineering analysis for domestic sleep environments is still limited. This work examines a gel-topper-based water-circulation system that removes heat directly at the human–bed interface, aiming to maintain comfortable skin temperatures with far lower energy use than room-level cooling. A steady-state thermal-resistance model was developed to represent heat flow from the skin through the contact layer, gel topper, foam slab, and water channels, while also accounting for convective–radiative losses to ambient air. A typical adult produces about 72 W of metabolic heat, of which 57.6 W is sensible. Simulations show that a foam-only mattress ($R \approx 1.59$ K/W) traps heat and leads to skin temperatures of 29–30 °C even with cold coolant, confirming its poor thermal pathway. Adding a 10 mm conductive gel layer ($k = 1$–3 W/m·K) on top of the mattress lowers the bed resistance by nearly two orders of magnitude, enabling **60–100 W of heat removal** at water inlet temperatures of **20–22 °C**, and maintaining skin temperature within **22-24 °C** under typical room conditions (T_a = 26 °C). A constrained SLSQP optimisation was used to identify feasible operating regions under comfort (22–26 °C) and condensation limits ($T_w \geq T_{dew} + 1$ °C). The resulting operating window lies at **moderate coolant temperatures** and **low-to-moderate flow rates** (≈1–3 L/min), where pump and chiller demands remain small. These results indicate that localized interface cooling can handle the entire sensible heat load of a sleeping adult using modest coolant temperatures and significantly lower power (up to 70% lower) than room-level air conditioning.

Key Words: *Localized cooling, Heat transfer, Thermal comfort, Optimization, Energy efficiency*

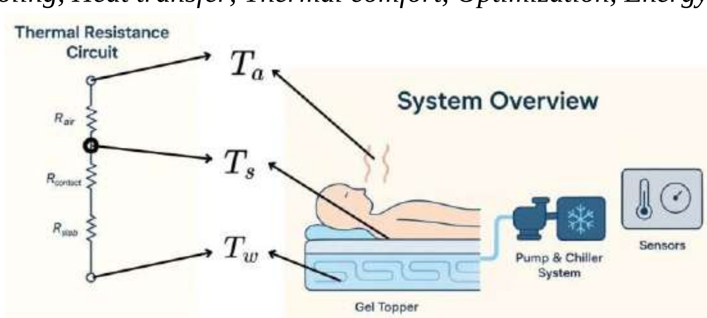


Figure 1: System overview and equivalent thermal resistance circuit showing key temperatures (T_a, T_s, T_w)

PARAMETRIC NUMERICAL ANALYSIS OF A REFRIGERANT EJECTOR: INFLUENCE OF MIXING CHAMBER GEOMETRY AND NOZZLE EXIT POSITION

Lokesh S Aurangabadkar and Sharad Chaudhary
Institute of Engineering and Technology, Devi Ahilya Vishwavidyalaya
Khandwa Road Indore-452017 (M.P.), India, lokeshsaurangabadkar@gmail.com, schaudhary@ietdavv.edu.in

ABSTRACT

Ejectors have emerged as promising components for environmentally sustainable refrigeration systems due to their simple construction, negligible maintenance requirements, and ability to recover expansion losses. However, their performance is highly sensitive to internal geometric configuration, especially in two-phase ejectors where complex shock structures and phase interactions dominate the flow. Although several studies have examined ejectors, the individual influence of key mixing-chamber geometric parameters on two-phase flow dynamics remains insufficiently quantified, limiting the development of optimized designs. This study provides a novel and systematic CFD-based parametric investigation of a two-phase constantarea ejector by isolating the effects of three critical geometric parameters: (i) mixing-section diameter, (ii) mixing-chamber length, and (iii) nozzle exit position (NXP). Each parameter was varied independently under fixed baseline operating conditions, and its influence on entrainment ratio, pressure recovery, shock location, and overall coefficient of performance (COP) was quantified to highlight geometric sensitivity with high resolution. The results show strong, quantifiable trends. Increasing the mixing-section diameter from 2.44 mm to 4.54 mm enhanced the entrainment ratio from 0.05 to 1.27 and increased COP from 0.046 to 1.19, demonstrating more than a 25-fold improvement in entrainment capability. Variations in mixingchamber length displayed non-monotonic behaviour; the optimal performance was obtained at 35.6 mm, where the entrainment ratio reached 0.839 and COP 0.787, compared with a minimum entrainment ratio of 0.165 at 39.6 mm. Sensitivity to NXP was especially notable: shifting the NXP from 33.64 mm to 38.64 mm increased the entrainment ratio from 0.796 to 0.874 and improved COP from 0.863 to 1.041, indicating significant enhancement in mixing and pressure recovery as the shock position stabilized. Overall, this study offers new quantitative insight into the geometric sensitivities of two-phase ejectors and clearly identifies optimal parameter ranges that maximize entrainment and COP. The findings provide valuable design guidelines for developing high-efficiency ejector-based refrigeration systems and contribute directly to improving the energy performance of next-generation low-GWP cooling technologies.

Key Words: *Two-phase ejector, CFD simulation, Mixing-chamber geometry, Nozzle exit position*

NUMERICAL INVESTIGATION OF THERMOHYDRAULIC PERFORMANCE OF MINICHANNEL HEAT SINK FOR AUTOMOTIVE EXHAUST WASTE HEAT RECOVERY COOLED BY HYBRID NANOFLUID

Saurav Chakrawerti[1], Rakesh Kumar[2]

[1]Department of Mechanical Engineering, Indian Institute of Technology (ISM) Dhanbad, Jharkhand – 826 004, India.

[2]Department of Mechanical Engineering, Indian Institute of Technology (ISM) Dhanbad, Jharkhand – 826 004, India.

ABSTRACT

An effective approach to boost the efficiency of thermoelectric generator (TEG) systems is to enhance heat transfer on the cold side. Lately, nanofluids/hybrid nanofluids have been getting significant interest and become a great deal of focal point in thermoelectric applications due to their excellent heat transfer capabilities from the cold side of the thermoelectric generator (TEG). It proposes a novel thermoelectric generation (TEG) system utilizing nanofluid/hybrid nanofluid as the coolant to enhance heat transfer efficiency. This design aimed at generating electricity from a limited hot surface area, such as that mounted on exhaust pipe. In traditional TEG system, thermoelectric modules (TEMs) are directly affixed on the exhaust pipe cooled without heat sink, with their electricity output being directly proportional to the heated surface area.The primary emphasis of this research work is to analyze the forced convective heat transfer within a square finned minichannels heat sink (SFMCHS) with a hybrid nanofluid composed of water, magnesium oxide and silver $[H_2O/MgO+Ag]^{hnf}$. Through the novel studied geometry, the results are described using power output, conversion efficiency, Nusselt numbers, average Nusselt number, convection heat transfer coefficient, including hot/cold side temperatures. Mono/hybrid nano fluids have been proven to play a crucial role in enhancing the effectiveness of TEG for waste heat recovery. Following the aforementioned literature route, we found that, to the best of our knowledge, spherical dimpled square finned heat sink was not mentioned in these kinds of publications. In the present work power output and voltage were increased by 5.28% and 6.23% utilizing 1% Ag-MgO/Water hybrid nanofluid as coolant compared to the base fluid at an inlet exhaust temperature of 500K.The total area of TEGs can be reduced by 11.90% using an Ag-MgO/water hybrid nanofluid as coolant as compared to the base fluid, it would decrease overall cost of the TEG system while offering ease of setup.

Key Words: *SFMCHS, minichannels, AETEG, nanofluids, Hybridnanofluids, convective heat transfer coefficient, poweroutput.*

CFD ANALYSIS OF FLOW CONTROL VALVE FOR SPACECRAFT PROPULSION SYSTEMS

P S B Pratyush, Savitry Kumari, Ram S N, Rajeev Senan C, Dr. CH Sreenivasa Rao and K K M Shanbhogue

Liquid Propulsion Systems Centre, 80 Feet Road, HAL 2nd Stage, Kodihalli, Bengaluru, Karnataka, 560008, India.

psbpratyush@lpscb.gov.in, savitry@lpscb.gov.in, ram_sn@lpscb.gov.in, c_rajeev@lpscb.gov.in, chsrao@lpscb.gov.in, kkms@lpscb.gov.in

ABSTRACT

The Earth observation & communication spacecrafts of ISRO uses Monopropellant and BiPropellant Propulsion systems for various purposes viz, Station keeping, Orbit raising manoeuvres,
Orbit maintenance, Re-boosting and De-boosting manoeuvres of the Spacecraft. Flow Control Valves are used for Supply of Propellant to Thrusters in Spacecraft Propulsion systems. Conventionally, Solenoid actuated Flow Control Valves are employed due to the quick response time, remote operation capability and simplicity in configuration. Direct acting Solenoid valves consists essentially of coil, Plunger and a valving element. Flow field analysis of a Flow Control Valve is essential in estimating the Pressure drop across the valve which directly influences the performance of the thrusters. The parametric analysis wrt the Stroke provides the margins available on the valve in terms of Flow rate for further optimisation. This stroke optimisation directly influences the Actuation voltage of the valve and the usage of this study is predominant in the design phases, where experimental results will not available. This CFD problem is solved using a 3D, Static, Viscous, Turbulent model in ANSYS Fluent®. A pressure based solver is used and SST kω turbulent viscous model is employed to characterise the flow. The present study is done without considering any heat transfer characteristics and hence, energy equation is not solved. Grid Independence analysis was carried out. The model is validated by carrying out experimental verification of flow rates at different ΔP for the FCV. The results predicted by the model were within 6.5% of the experimental results. The CFD model is useful for estimating the Pressure drop and flow characteristics of future new valve developments.

Key Words: *CFD, Spacecraft Propulsion, Solenoid valve, ANSYS, Experimental validation*

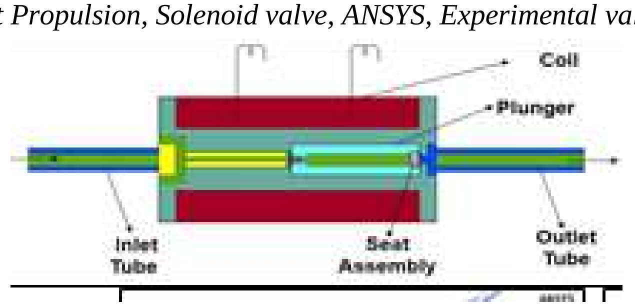


Figure 1 Geometry of the problem

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

CFD OPTIMIZATION OF A GRADED-POROSITY FLOW REDISTRIBUTION DEVICE FOR UNIFORM FLOW IN REFLUXTYPE ANNULAR LINEAR INDUCTION PUMPS

Ram Kumar Maity*, M. Rajendrakumar, R. Suresh Kumar and K. Natesan
Thermal Hydraulics Section, Indira Gandhi Centre for Atomic Research, Kalpakkam, India,
*rammaity@igcar.gov.in

ABSTRACT

Liquid sodium, employed as a coolant in Indian Fast Reactors, is circulated through auxiliary circuits using electromagnetic (EM) pumps. Such pumps have no moving parts and rely on Lorentz forces to impart motion to a conducting liquid metal. Lack of moving parts ensures a leak-free and highly reliable pumping system. In reflux-type Annular Linear Induction Pumps (ALIP), sodium enters an annular section through a single radial inlet nozzle. The annular portion of an ALIP is the zone where sodium is subjected to EM forces and pumping action takes place. A radial entry leads to circumferential flow maldistribution that can induce magneto-hydrodynamic instabilities and adversely affect pump performance. To minimise non-uniformity in flow velocity distribution, a compact flow (re)distribution arrangement using a graded-porosity baffle (Fig. 1) has been conceived. Three-dimensional CFD studies towards optimisation for minimal pressure drop and flow mal-distribution are presented in this paper. The proposed arrangement redistributes the flow at the inlet to the annular portion using a sector-wise variable porosity distribution within the inlet plenum. The porosity distribution, imposed as pressure-drop coefficients, is optimised to achieve a near-uniform entry velocity into the pump annulus. Increase in hydraulic losses is minimised and implementability is ensured. A 180° sector of the ALIP is studied using ANSYS Fluent 19.2, with liquid sodium at 250°C as the working fluid. Three alternate configurations with varying porosity gradients are evaluated and compared with the reference case. The graded-porosity arrangement reduces the velocity maldistribution from 2.4 m/s in the nominal case to a very low value of 0.4 m/s (Fig. 1). These results demonstrate that a sector-wise graded-porosity arrangement significantly improves flow uniformity in reflux-type ALIPs.

Key Words: *Flow redistribution, ALIP, Fast Reactor, Liquid Sodium*

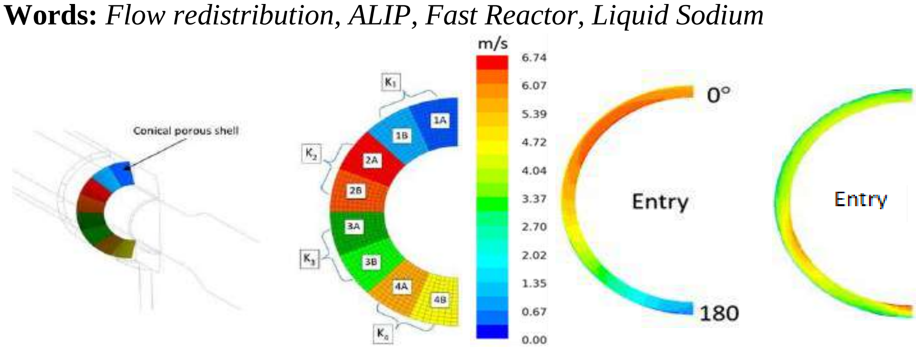


Fig.1. Concept of the flow distribution device (left) and improvement in velocity field at the inlet to annular portion (right)

FLOW REDISTRIBUTION IN REFLUX-TYPE ELECTROMAGNETIC PUMPS: CFD-BASED DESIGN REALIZATION

R. Nandakumar, Ram Kumar Maity*, M. Rajendrakumar, R. Suresh Kumar, Amzad Pasha, U. Parthasarathy and K. Natesan

Reactor Design Group, Indira Gandhi Centre for Atomic Research, Kalpakkam, India*, rammaity@igcar.gov.in

ABSTRACT

The study focuses on a reflux-type Annular Linear Induction Pump (ALIP) designed for auxiliary circuits of Indian sodium-cooled fast breeder reactors. The pump is designed to operate at a nominal flow rate of 120 m³/h. In this configuration, liquid sodium enters the annular section through a single radial inlet nozzle, which leads to significant circumferential maldistribution of flow at the entry region that can interact with magneto-hydrodynamic forces and potentially trigger instabilities within the pumping zone. A proposed graded-porosity conical shell-based flow distribution device (FDD) redistributes the flow circumferentially, minimizing velocity nonuniformity while maintaining low hydraulic losses and ensuring ease of fabrication. A graded porosity arrangement to effect flow redistribution has been conceived in earlier studies. The present study describes the translation of the derived optimised porosity distribution into an implementable configuration. The device requires specific pressure-drop characteristics to achieve the desired redistribution effect while ensuring manufacturability and structural integrity. The specific sectorwise distribution is achieved by arranging appropriately spaced circular holes on an annular plate, based on porous plate pressure-drop correlations. Subsequently, a 180° three-dimensional CFD model of the ALIP with an integrated model of the FDD is developed in ANSYS Fluent 19.2. Parametric studies to understand and verify the effectiveness of the FDD are carried out. The velocity maldistribution shows a reduction from 2.4 m/s to 0.35 m/s at the annulus inlet after installation of the FDD. This improvement is achieved with an 8 kPa increase in overall hydraulic pressure drop, which is negligible compared to the developed pump head of 400 kPa. Further, the effectiveness of the FDD has been verified experimentally through in-house water model studies.

Key Words: *FBR, ALIP, Liquid Sodium, Flow maldistribution*

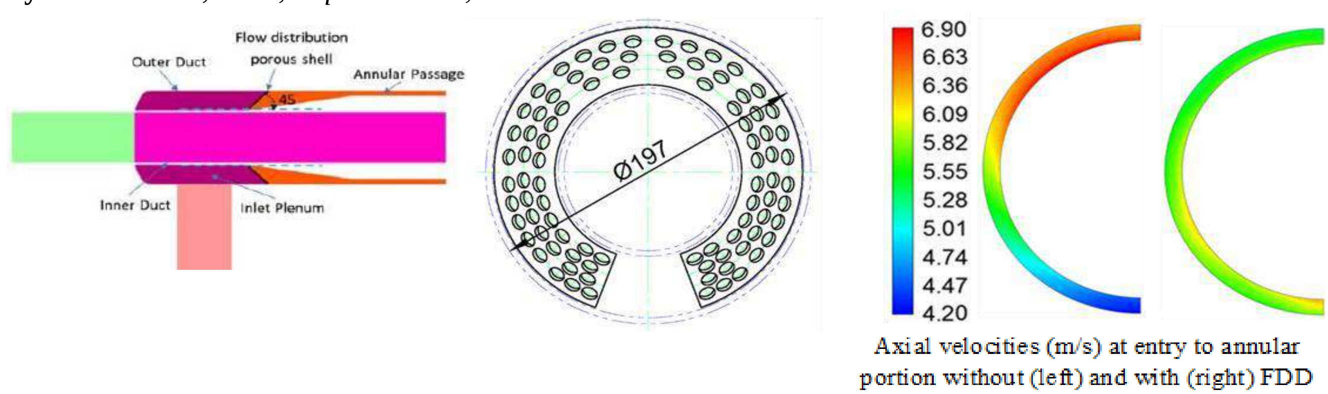


Fig.1. View of inlet plenum of ALIP showing FDD at suction to annular path (left), detailed view of FDD (centre) and effect of FDD on flow field (right)

ANALYSIS OF CONVECTIVE HEAT TRANSFER IN POWELL EYRING FLUID FLOW OVER A CURVED STRETCHING SURFACE WITH THE INFLUENCE OF MAGNETIC FIELD AND HEAT SOURCE

Nand Kishor Sharma[1], Reema Jain [1 *]

[1] Department of Mathematics and Statistics, Manipal University Jaipur Jaipur- 303007, Rajasthan, India

E-mail address: nandkishor.2410180013@muj.manipal.edu, reema.jain@jaipur.manipal.edu

ABSTRACT

The convective heat transfer properties in magnetohydrodynamic (MHD) flow of a nonNewtonian Powell–Eyring fluid over a curved exponentially stretching surface in the presence of a uniform transverse magnetic field and internal heat generation are thoroughly investigated numerically in this work. The current work provides a more realistic depiction of industrial stretching processes by explicitly incorporating surface curvature effects within a curvilinear coordinate framework, in contrast to previous research that mainly concentrate on flat or linearly stretching geometries. Boundary-layer approximations are used to build the governing nonlinear partial differential equations for momentum and energy, which are then similarly translated into a coupled system of ordinary differential equations. The MATLAB bvp4c solver is used to solve the resulting boundary-value problem numerically, guaranteeing great accuracy and stability. Numerous dimensionless parameters, such as the curvature parameter, magnetic parameter, Powell–Eyring fluid parameter, heat source parameter, and Prandtl number, are used in parametric calculations. The velocity profile is improved by roughly 18–25% when the curvature parameter is increased, according to quantitative study, because of the decreased effective contact area and reduced viscous resistance. On the other hand, Lorentz drag is greatly suppressed by the applied magnetic field, which causes the skin-friction coefficient to drop by as much as 22%. An internal heat source increases thermal energy transport, which causes the temperature field to rise by over 30% and the thermal boundary layer to thicken. Furthermore, a decrease in surface heat transfer efficiency is shown by the local Nusselt number decreasing monotically as the strength of the heat source increases. The current findings differ from traditional Newtonian and flat-sheet investigations found in the literature due to the combined influence of curvature and non-Newtonian rheology. These results have practical implications for polymer extrusion, coating technologies, metallurgical processes, and thermal control systems using electrically conducting non-Newtonian fluids and offer new physical insights into MHD heat transfer behavior in curved stretching systems.

Keywords. Powell Eyring fluid, MHD, Heat Source, Convective heat transfer, curved stretching sheet.

EXTRACTION OF CARPAINE FROM CARICA PAPAYA LEAVES AND ITS PHARMACOLOGICAL INSIGHTS: A COMPREHENSIVE REVIEW

Samarth Pasilkar[1], Pallavi Mahajan-Tatpate[2*]

[1]MTech. Chemical, Department of Chemical Engineering, Dr.Vishwanath Karad MIT World Peace University, Pune-411038 (samarthpasilkar0509@gmail.com)

[2]Department of Chemical Engineering, Dr.Vishwanath Karad MIT World Peace University, Pune-411038 (pallavi.tatpate@mitwpu.edu.in) *

ABSTRACT

Papaya (Carica papaya) plants, abundant in Central America and tropical regions of Africa and Asia, are valued both as food and for their medicinal properties. The leaves and fruit contain diverse bioactive compounds such as alkaloids, quinones, saponins, terpenoids, steroids, vitamins, and tannins, which contribute to various therapeutic applications. These compounds exhibit antibacterial, anti-inflammatory, antioxidant, antiulcer, and wound-healing activities. Traditionally, Carica papaya extracts have been used to manage ailments ranging from common conditions like headaches, fever, bronchitis, cough, and chickenpox, to serious illnesses including asthma, cancer, diabetes, liver disorders, and cardiovascular diseases. Additionally, the plant demonstrates notable Phyto-extraction and phytoremediation capabilities, underscoring its ecological significance. Among its bioactive constituents, carpaine, an alkaloid predominantly found in papaya leaves, holds significant pharmaceutical interest. Extracting carpaine effectively is crucial to harness its full potential. Conventional extraction techniques such as maceration and Soxhlet are widely used but suffer limitations like long extraction times, low yields, reliance on toxic organic solvents that pose environmental and health hazards. Advanced extraction techniques like Ultrasound-Assisted extraction (UAE) and Microwave-Assisted extraction (MAE) offer improvements but introduce challenges: UAE typically involves high energy consumption, while MAE faces difficulties in large-scale application. Integrating UAE and MAE technology presents a promising strategy to overcome these drawbacks. This approach can reduce extraction time, decrease CO_2 emissions, and enhance yield efficiency. Optimizing parameters such as solvent type and volume, irradiation duration, temperature, and microwave power is essential to maximize process efficiency, costeffectiveness, and environmental sustainability. However, standardizing extraction protocols remains challenging due to variability in carpaine yield and purity influenced by leaf maturity, environmental factors, and solvent systems. This review addresses the pharmacological and cosmetic applications of carpaine, evaluates existing extraction methods along with their benefits and limitations, this also describe in detail about different type of solvent used along with its quantitative analysis with its merits and demerits, and proposes standardized, eco-friendly, and sustainable approaches to optimize carpaine extraction from Carica papaya leaves.

Key words: *Carpaine, Carica Papaya Leaves, Extraction, Microwave and Ultrasonic assisted extraction, Sustainable technology*.

DEVELOPMENT OF SUSTAINABLE HIGH-PERFORMANCE 3D PRINTED POLYMER COMPOSITES AND FUNCTIONAL MEMBRANES FOR WASTEWATER TREATMENT APPLICATIONS

Sohail Ansari, Prarthana Deshpande, Pallavi Mahajan-Tatpate*

Department of Chemical Engineering, Dr. Vishwanath Karad MIT World Peace University, Pune, (Maharashtra-411038)

Sa9150905@gmail.com , deshpandeprarthana07@gmail.com , pallavi.tatpate@mitwpu.edu.in*

ABSTRACT

The rapid growth in industrialisation and urbanization has intensified contamination of water, growing demands for low-cost and flexible technologies for wastewater treatment. The conventional techniques, including chemical precipitation, adsorption, and advanced oxidation processes, are often stuck by various issues, such as low selectivity, poor regeneration, high operational costs, and secondary pollution. Membrane-based filtration techniques possess superior energy efficiency and outstanding rejection capabilities regarding contaminants; however, conventional polymeric membranes still face severe challenges in terms of fouling, short service life, poor reproducibility, and inability to control pore architecture preciselyRecent studies have demonstrated that 3D printing can significantly enhance membrane performance through precision control of pore geometry, thickness, and structural uniformity not possible in conventional phaseinversion membranes. In related membrane systems, 3D-printed internal structures showed up to 30% higher permeation rates than conventional modules. Similarly, 3D-printed turbulence promoters demonstrated ~15% reduction in the fouling rate during ultrafiltration. These imply that additive manufacturing allows for better hydrodynamics with reduced fouling, hence superior filtration performance. Besides, 3D printing is intrinsically a zero-waste fabrication technique that could reduce material waste by up to 90%, hence being highly suitable for sustainable industrialscale production.PLA-based biopolymer composites with nano/bio-filler reinforcement were formulated in this work to constitute a mechanically strong and sustainable membrane structure. The addition of natural reinforcements and nanomaterials has been proposed in the literature to effectively enhance the mechanical strength and chemical resistance of polymer composite materials (Zhang et al., 2019). These advancements encourage the development of multi-functional, reusable membranes. This study will therefore adopt additive manufacturing methods for material processing to introduce a state-of-the-art design–fabrication methodology that incorporates polymer science, composite engineering, and sustainable material design into next-generation, customizable membrane systems.This research underlines the fact that 3D printing is poised to revolutionise industrial wastewater treatment through the provision of scalable, low-cost, highperformance membranes that overcome the major drawbacks associated with conventional approaches to membrane fabrication.

Key words*: 3D Printing, Membrane technology, Waste water treatment, Fused Deposition Modelling Sustainable and Eco-friendly technolog*

NUMERICAL STUDY ON FLOW CHARACTERIZATION AND PERFORMANCE OPTIMIZATION OF SAVONIUS TURBINE IN A CONFINED CHANNEL

P. D. Vijaya Kumar[1], P.S. Mohanasaravanan[2], S. Jayavel[2], Shaligram Tiwari[1]
[1]Department of Mechanical Engineering, Indian Institute of Technology Madras Chennai-600036, India, vijayperumalla20@gmail.com
[2]Department of Mechanical Engineering, Indian Institute of Information Technology in Design and Manufacturing, Melakottaiyur, Chennai-600127, India, sjv@iiitdm.ac.in

ABSTRACT

The meteoric rise in global demand for energy calls for more investment and research in renewable sources of energy. Wind energy stands out among these renewable sources for it is available in most of the places during most of the days. While large-scale windmills use the efficient Horizontal Axis Wind Turbines (HAWT), they require high wind speeds and higher initial investment, making them a tough choice for use in low wind speed urban areas, as compared to Vertical Axis Wind Turbines (VAWT) which are also cheaper. Savonius turbine, a drag-based VAWT, consists of a simple design, better self-starting capability and is cheaper as compared to its lift-based counterpart, Darrieus turbine. The major drawback of Savonius turbine is its low efficiency. Confining the flow in a channel has proved to be useful in increasing the efficiency of Savonius turbine.

The present two-dimensional numerical study proposes a hybrid strategy for Savonius turbine by integrating a confined wall and deflector plates on both the walls. The design of the Savonius turbine and the confinement channel are adopted from Bai et al. [1], in which the turbine is placed in a confined channel of width two times the diameter of the turbine resulting in an increase of coefficient of performance by 130%, from 0.25 to 0.57. The confinement increases the flow momentum acting on the turbine by reducing flow leakage and maintaining a high-pressure region near the blades, resulting in an increase of the positive torque. However, this configuration results in increase of momentum on both the advancing and returning blades, thereby causing higher negative torque on the returning blade.

To overcome this limitation, the present study introduces deflector plates in addition to the confined wall. The deflector plates shield the returning blade from direct wind impact and redirect the flow towards the advancing blade, thereby increasing the net positive torque. This combination of confinement and dual deflector plates has not been reported in the literature. For a flow of air with Reynolds number (Re) 1.0×10^5 and turbulence intensity (I_{tu}) of 10%, deflector plates are placed on either side of the channel varying the angle of the obstacle plate and distance of the obstacle plates from the turbine blades. Computations in the present study are carried out using ANSYS Fluent 2023 R2, with the grid generated in ANSYS ICEM CFD 2023 R2 to accurately resolve the boundary layer and wake regions. The results will be quantified based on power coefficient (Cp) and momentum coefficient (Cm) over a range of tip speed ratios (TSR).

Key Words: *Savonius Wind Turbine, Confined Wall, Deflector Plates*

A COMPARATIVE RHEOLOGICAL ASSESSMENT OF VISCOELASTIC FLUIDS FOR MEMBRANE-DRIVEN MICRO-PUMPING

Vinita Goyal, Kushal Sharma
Department of Mathematics, MNIT Jaipur – 302017, India
vinitagoyal740@gmail.com

ABSTRACT

This research conducts a comparative investigation of various viscoelastic fluid models: Maxwell, Jeffery, and Oldroyd-B to examine the dynamics of membrane-driven micro pumping systems operating within microchannels. The study focuses on the influence of characteristic rheological time scales of the models, including elastic relaxation, retardation and solvent coupling under cyclic membrane actuation on flow behaviour. The simulation reveals that the Maxwell fluid model, which exhibits significant elastic recoil and stress relaxation due to cyclic membrane deformation, yields strong transient flow under rapid oscillations; the Jeffrey fluid model, which characterized by retardation-driven phase lag, producing steadier yet dampened pumping, crucial for polymeric and biological fluids; and the Oldroyd-B fluid model, which merges polymeric elasticity with solvent viscosity, leading to smoother pumping through balanced viscous-elastic interactions. Through computational simulations, these models are compared based on their impact on flow characteristics, pressure gradient, streamline distribution, and overall efficiency of micropumps. The finding highlights the choice of an appropriate viscoelastic model has a substantial impact on micro-pumping efficiency, offering essential insights for the optimal design of advanced microfluidic devices, particularly in fields such as biomedical engineering, where precise manipulation of non-Newtonian fluids is crucial.

Key Words: Membrane kinematics*, Viscoelastic fluid, Pressure gradient, Streamlines, Flow characteristics*

FORMATION OF ELECTROSTATIC DISCHARGE SHEET USING BIODEGRADABLE POLYVINYL ALCOHOL/CARBON BLACK COMPOSITE FOR SUSTAINABILITY

Prarthana Deshpande, Sohail Ansari Pallavi Mahajan-Tatpate*
M.Tech. Chemical, Department of Chemical Engineering, Dr. Vishwanath Karad MIT World Peace University, Pune, Maharashtra 411038
Department of Chemical Engineering, Dr. Vishwanath Karad MIT World Peace University, Pune, Maharashtra 411038 deshpandeprarthana07@gmail.com, Sa9150905@gmail.com, pallavi.tatpate@mitwpu.edu.in*

ABSTRACT

The research focuses on developing a biodegradable electrostatic discharge (ESD) sheet by using polyvinyl alcohol (PVA) as a sustainable polymer matrix by incorporating carbon black or graphene as conductive fillers. This study aims to form an eco-friendly alternative to conventional petroleum-based, non-biodegradable ESD materials typically used in industrial and electronic applications. PVA was selected for its excellent film-forming ability, transparency, water solubility, and biodegradability. To optimize electrical and mechanical performance, composite films with various concentrations of carbon black were formulated and tested. Findings showed that excessively high filler concentrations did not significantly improve conductivity or related ESD properties, emphasizing the need for an optimal ratio between polymer and filler content. The films were fabricated through the solution casting technique aided by ultrasonic dispersion and reactor-based processing to ensure homogeneous mixing, improved filler dispersion, and stable film formation. The resulting composites demonstrated balanced mechanical strength, flexibility, and electrostatic dissipation capability. Structural and thermal characteristics were evaluated through field emission scanning electron microscopy (FE-SEM), water contact angle analysis, and thermogravimetric analysis (TGA). The prepared PVA–carbon black films exhibited surface resistivity within the static dissipative range of 10^5–10^{11} Ω/sq, confirming efficient charge dispersion and electrostatic protection properties. Additionally, the sheets possessed strong flexibility and mechanical integrity, making them suitable for packaging sensitive electronic components, antistatic coatings, and industrial protective films. This study concludes that the combination of biodegradable PVA with conductive carbon black or graphene can effectively yield a sustainable, high-performance ESD material. The proposed solution casting and ultrasonicassisted processing approach provides a reliable pathway to fabricate eco-friendly, staticdissipative films that reduce environmental impact while maintaining industry-standard performance.

Key words: *Polyvinyl alcohol (PVA); Carbon black; Electrostatic discharge (ESD);Conductive polymer composites Sustainable, Biodegradable and Environment-friendly material*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

WASTE TYRE PYROLYSIS CHAR: ACTIVATION METHODS, STRUCTURAL FEATURES, AND APPLICATIONS- A REVIEW

Harshwardhan Patil, Samarth Pasilkar, Pallavi Mahajan-Tatpate*

Department of Chemical Engineering, Dr. Vishwanath Karad MIT World Peace University,
Pune, Maharashtra 411038
patilharshwardhan117@gmail.com, samarthpasilkar0509@gmail.com, pallavi.tatpate@mitwpu.edu.in*

ABSTRACT

The continuous generation of tyre waste worldwide is becoming a major environmental threat due to the non-biodegradable and complex nature of tyres. End-of-Life Tyres (ELTs) are one of the fastest-growing global waste streams. Annually, 1.5 billion tyres are discarded globally, out of which hundreds of millions of tyres are produced in India. There are various methods to recycle tyres, including mechanical grinding, devulcanization, and pyrolysis. Each method has its own advantages and limitations. The mechanical grinding is simple but has low value input, and is energy-intensive, whereas devulcanization produces inconsistent quality and has potential chemical hazards. Pyrolysis has emerged as a favourable method of converting tyres to various useful products such as pyrolysis oil, gas and solid char. Tyre char is a waste by-product, and its quality can be improved by accurate activation and optimisation control measures to a value added multi-functional carbon material. Also, the gas emissions can be controlled to make the process eco-friendly. This study focuses on the activation and characterization of tyre char and its reinforcing behavior in rubber compounding. Its performance is evaluated by assessing the improvements in adsorption and surface properties. The mechanical and morphological properties of activated tyre char are comparable to commercial carbon black and are a viable substitute. The activated tyre char has several benefits of reducing waste, lower production expenses, recovering energy and circular use of carbon materials. Also, it helps prevent landfilling and reliance on petroleum-based fillers. The activated tyre pyrolysis char achieves surface areas up to 800–2000 m^2/g and heavy metal adsorption capacities exceeding 100 mg/g, which is high efficiency compared with commercial activated carbon. Such improvements in performance validate its potential for environmental remediation and sustainable resource recovery. This review discusses details of recycling methods for waste tyres, their advantages/ disadvantages, various methods of tyre char production, the effect of adding functional groups to tyre char for its activation, and its applications. It also suggests the best processing method and prospects for developing a sustainable, eco-friendly solution for tyre reprocessing.

Key words: *Waste Tyre Pyrolysis, Activated Tyre Char, Recycle, Rubber Compounding, Carbon Black Substitute.*

PERFORMANCE AND EMISSION OPTIMIZATION OF A DIESEL ENGINE FUELED WITH WASTE PLASTIC OIL–NANOPARTICLE BLENDS USING RESPONSE SURFACE METHODOLOGY

Tushar Anand[1]*, Neha Kumari[2], Jyoti Khandelwal[3], Sneh Krishna[4]

[1]Department of Mechanical Engineering, Amity University Rajasthan, Jaipur, India. tusharanand.sinha@gmail.com
[2]Department of Electrical and Electronics Engineering, Amity University Rajasthan, Jaipur, India. nkumari@jpr.amity.edu
[3]Department of Computer Science & Engineering, Amity University Rajasthan, Jaipur, India. jkhandelwal@jpr.amity.edu
[4]Department of Computer Science & Engineering, Government Polytechnic College, Khagaria, India. snehkrishna497@gmail.com

ABSTRACT

The rising demand for sustainable and cleaner-burning fuels has intensified interest in converting plastic waste into alternative energy sources within a circular economy framework. Waste plastic oil (WPO), produced through thermochemical pyrolysis, is a promising diesel substitute due to its adequate calorific value and its potential to reduce plastic pollution. However, its direct use in diesel engines results in incomplete combustion, higher BSFC, and elevated HC and CO emissions because of its low cetane number and high aromatic content. This study investigates nanoparticleenriched WPO–diesel blends in a single-cylinder diesel engine to overcome these limitations and enhance combustion, performance, and emission characteristics. The novelty of this work lies in integrating nanoparticle-assisted combustion improvement with a Response Surface Methodology (RSM)-based multi-response optimization framework tailored specifically for WPO fuels. Unlike prior studies that mainly report qualitative trends, this research develops statistically validated quadratic RSM models to predict and optimize BTE, BSFC, and key emissions (NOx, CO, HC, and smoke opacity), thereby strengthening scientific understanding and predictive capability. Experimental results show that nanoparticle addition markedly improves combustion behaviour. BTE increased by 6–9% and BSFC decreased by 4–7% compared to untreated WPO–diesel blends. HC and CO emissions decreased by 12–18% and 15–20%, respectively, owing to enhanced oxidation and catalytic effects, while NOx showed only a marginal rise of 3–5%, remaining within acceptable limits. The developed RSM models demonstrated high accuracy with R^2 values ranging from 0.93 to 0.98, and ANOVA confirmed statistical significance ($p < 0.05$) for major responses. Optimization results identified ideal combinations of WPO blend ratio and nanoparticle concentration that maximize efficiency and minimize emissions. Overall, the study establishes nanoparticle-assisted WPO blends as a viable, cleaner alternative to diesel and contributes a robust optimization framework for advancing waste-to-energy fuel technologies.

Key Words: *Waste Plastic Oil, Nanoparticles, Diesel Engine, RSM Optimization, Emissions.*

NANOFLUIDS HEAT PIPES FOR COMPACT THERMAL MANAGEMENT: STATE OF THE ART AND OPEN GAPS

Bisma Ali[1*], Afzal Husain[1*], Adnan Qayoum[2], Nasser Al-Azri and Nabeel Al-Rawahi

[1]Mechanical and Industrial Engineering Department, Sultan Qaboos University, Muscat, Oman, Email address: bismaqubravi@gmail.com, afzal19@squ.edu.om

[2] Mechanical Engineering Department, National Institute of Technology Srinagar, J & K, India

ABSTRACT

In recent years, significant research has focused on enhancing the thermal performance of heat pipes through the use of nanofluids, which are suspensions of nanoparticles in conventional heat transfer fluids (HTFs). Traditional heat transfer fluids exhibit inherent limitations in thermal conductivity and convective heat transfer capability, which significantly constrain thermal efficiency. This review consolidates experimental and analytical studies to synthesize recent advancements in the application of nanofluids for heat transfer enhancement within heat pipes, with a particular focus on their impact on the thermal performance and operational reliability. The findings consistently demonstrate that nanofluids improve heat transport characteristics by increasing effective thermal conductivity, reducing thermal resistance, and enhancing phase-change efficiency. These enhancements are primarily attributed to mechanisms such as intensified Brownian motion, improved surface wettability, and nanoparticle deposition that promotes nucleate boiling. However, the extent of performance improvement depends strongly on nanoparticle type, concentration, and size, as well as on heat pipe geometry, configurations, and operating conditions. Incorporating nanofluid into micro-grooved heat pipes results in a consistent decrease in thermal resistance by approximately 30-66%, along with 10-65% improvements in both thermal conductivity and heat transfer coefficients. For pulsating heat pipes, reduction in thermal resistance of about 32% and increase of roughly 33% in the maximum heat transport capacity were observed, whereas sintered wick configurations exhibited enhancements in effective thermal conductivity of up to 63%. Overall, the most favorable nanoparticle concentration range was typically identified between 0.1 and 1 wt.%. However, excessive nanoparticle loading may cause agglomeration and an increase in viscosity, which may diminish heat pipe performance. The review discusses an optimal concentration range for maximizing efficiency, which is critical for long-term stability, sedimentation, and reliability. Overall, nanofluid-based heat pipes offer a promising pathway to compact, high-performance thermal management systems for electronics, renewable energy, and aerospace technologies.

Key Words: *Heat Pipe, Nanofluid, Thermal Conductivity, Thermal Resistance, Heat Transfer*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

DEVELOPMENT OF LATTICE BASED COMPACT HEAT EXCHANGER FOR EFFICIENT HEAT RECOVERY SYSTEM

Ashok Kumar, N. Gnanasekaran

Indian Institute of Technology Tirupati, Department of mechanical engineering, Tirupati, Andhra Pradesh, India,
me24m217@iittp.ac.in and gnanasekaran@iittp.ac.in

ABSTRACT

A significant amount of thermal energy from engine exhaust is often lost to the surroundings in industrial applications. Recovering this waste heat plays a vital role in improving energy efficiency and enhancing the overall performance of engine systems. In the present study, a compact heat exchanger is designed to effectively utilize this exhaust heat, and its performance is evaluated for different lattice structures, including Octet truss, Uniform Body Centered Cubic (UBCC), and FaceBody Centered Cubic (FBCC) configurations. The novelty of the present work is not only identifying the appropriate structural properties but also to find out flow based interfacial area which can increase the heat transfer at the same time reduces the pressure drop. The design employs a unit cell length (UCL) of 4 mm with a porosity of 0.937 to achieve an optimal balance between fluid flow and heat transfer characteristics. The inlet velocity of the exhaust gas varies from 6 to 30 m/s at a temperature of 523 K, while a pressure outlet boundary condition is imposed at the outlet. The bottom wall of the channel is maintained at an isothermal temperature of 300 K to absorb the incident heat flux and enhance the overall heat transfer rate. Key performance parameters such as pressure drop, heat transfer coefficient, and Nusselt number are analysed and compared for all lattice configurations to identify the most efficient structure for waste heat recovery applications. The analysis plays a major role in developing new compact heat recovery system using 3D printing and satisfying one of the Sustainable Development Goals.

Key Words: *CFD, Heat exchanger, Lattice structure, Unit cell, Waste heat recovery*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

TURBULENT TRANSPORT AROUND A SLENDER BLUFF BODY UNDER FLOWING WIND

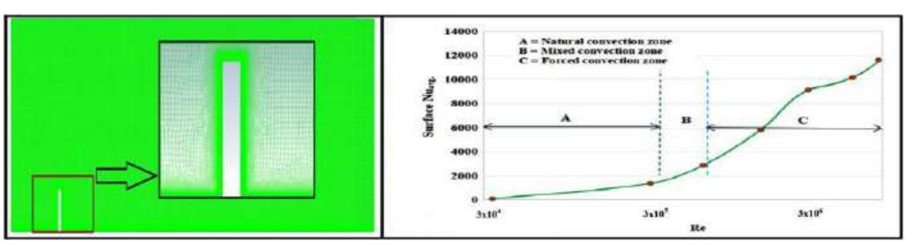
Surajit Bit[1,2], Satyajit Das Karmakar[3], Sudhir Ch Murmu[2], Himadri Chattopadhyay[2]

[1]Department of Defence Production, North 24 PGS 743144, India, surajitbit77@gmail.com
[2]Department of Mechanical Engineering, Jadavpur University, Kolkata 700032, India
[3]Department of Mechanical Engineering, Jalpaiguri Government Engineering College, Jalpaiguri 735102, India

ABSTRACT

Under the background of more number of high-rises buildings with increasing extreme weather events, studies on wind flow pattern is becoming extremely important. This work focus to understand the flow dynamics for tall structures in the range of wind speed up to 100 Km/h. Investigations on flow behaviour around a slender bluff body has been performed in turbulent regime in the Reynolds number range, Re ϵ [0.0342×10^6, 9.508×10^6]), where Re is based on average flow velocity and height of the bluff body. The bluff body is of 5 metre height with the domain aspect ratio of 1:20 subject to wind speed up to 100 km/h. A domain with sufficient length has been considered where a slender body is present, flushed with the bottom wall. RNG k-ϵ two-equation model with standard wall functions have been used for turbulent closure. Simulation results show that the value of coefficient of pressure (C_p) on the top surface of the slender bluff body changes from negative to positive with the increase of Re. Details of turbulent flow field with turbulent statistics are presented which includes wake deficit, turbulent kinetic energy and pressure coefficients. From this investigation it is observed that the normalized turbulent kinetic energy (TKE) ranges from 0.08 at highest Re to 0.17 at lowest Re. Apart from the flow field, the slender body is considered at elevated temperature for which heat heat transfer is analyzed. The overall average Nu of bluff body varies from 98.48 to 12511.28 corresponding to the Re=0.0342×10^6 to Re=9.508×10^6 respectively. Heat transfer from the slender bluff body is analyzed based on the distribution of Nusselt number (Nu) over its vertical surfaces and the top surface. The values of Richardson number (Ri) which determines the relative strength of natural convection to forced convection varies within 0.006 at high wind velocity to 466 at low wind velocity. In figure 1(a), the meshing of the computational domain is shown while 1(b) shows averaged Nu distribution at different Re and Ri. The thermal transport shifts from natural convection dominated zone at low wind velocity to forced convection dominated zone at high wind velocity as observed through changing slopes of the Nu-Re plot.

Key words: *Slender bluff body; Turbulent flow; Wake velocity deficit; Pressure coefficient*



(a) (b)

Figure 1: (a) Mesh generation around the slender body and (b) distribution of averaged Nusselt number

INVESTIGATING THE AERODYNAMIC EFFECTS OF LEADING-EDGE TUBERCLES IN SUPERSONIC FLOW: A CFD-BASED PARAMETRIC STUDY

Rishab Maurya[1], Pranay Katke[2]

[1] MCT Rajiv Gandhi Institute of Technology, Mumbai, India, rm11042003@gmail.com
[2] Indian Institute of Technology, Kharagpur, West Bengal, India, pranayvijaykatke25@kgpian.iitkgp.ac.in

ABSTRACT

Strong shockwave–boundary layer interactions (SBLI) cause aircraft to sharp drag increase at supersonic speeds, causing flow instability and early separation **[5,6].** The tubercle effect, inspired by the humpback whale flipper, offers a potential passive control strategy **[1]**. In subsonic and transonic regimes, these sinusoidal leading-edge bumps create streamwise vortices that excite the boundary layer and postpone separation **[5,6]**. However, their impact on supersonic flow, where compressibility and shock interactions remains unexplored **[5,6]**. Hence, this study aims to examine the ability of tubercles to influence shock behaviour, reduce SBLI, and improve flow stability across a range of Reynold's numbers and angles of attack on a NACA 64(1)-212 wing with leading-edge tubercles at Mach 1.5 and 2.5. Because of its thin leading edge, the NACA 64(1)-212 airfoil was selected as the baseline **[5]**. The tubercles model featured sinusoidal protrusions of 2% chord amplitude (20 mm) and 6% chord wavelength (60 mm) **[3,4]**. Both geometries were created in ANSYS Design-Modeler with similar domains and trailing edge was trimmed for mesh consistency. Twenty-four cases were analysed across Mach 1.5–2.5, Re = 1×10^5–2.74×10^6, and AoA = 0°, 8°, and 12°. The aerodynamic comparison between the baseline and tubercles NACA 64(1)-212 wings demonstrated a clear flowregime dependence. At Mach 1.5, both configurations performed similarly. For example, at 0° AoA, the tubercled wing produced 14–24% more lift but also experienced higher drag, resulting in no net gain in L/D. However, at Mach 2.5, the influence of tubercles became more pronounced, achieving drag reductions of 1–5% and lift increases up to 6%, improving L/D ratio by approximately 5–6%. At 8° AoA, the drag coefficient decreased from roughly 0.138 (baseline) to 0.132 (tubercles), and at 12° AoA, the improvement approached 6%. Furthermore, shock analysis showed a downstream shift of primary shocks from x/c ≈ 0.003–0.005 to x/c ≈ 0.011–0.013, indicating delayed formation and redistribution of compression strength. At higher Mach numbers and AoA's, baseline cases diverged due to strong shock oscillations, while tubercled cases converged smoothly. Also, residuals in baseline cases oscillated in the 10^{-2}–10^{-3} range, whereas the tubercled cases consistently converged below 10^{-4}, confirming improved numerical stability. This study uses RANS-based CFD to assess the aerodynamic influence of leading-edge tubercles on a NACA 64(1)212 wing in supersonic flow. Limited benefits appeared at lower Mach numbers, but higher Mach cases observed modified shock structure and pressure distribution, delaying shock formation and promoting smoother flow recovery. Their effectiveness remains geometry- and regime-dependent, requiring further tubercle optimization.

THERMAL ANALYSIS OF ELECTRIC VEHICLE BATTERY COOLING SYSTEM WITH ELLIPTICAL ALUMINIUM FOAM OF VARYING POROSITIES AND LOCATIONS IN A FLOW CHANNEL

Samson Olusegun Fatukasi[1] and Tunde Bello-Ochende[1,2]

[1]Department of Mechanical Engineering, University of Cape Town, Private Bag X3, Rondebosch, 7701, South Africa

[1,2]Applied Thermal Process Modelling Research Unit, Department of Mechanical Engineering, University of Cape Town, South Africa.

tunde.bello-ochende@uct.ac.za

ABSTRACT

In this paper, thermal analysis of a cylindrical lithium-ion battery pack coupled with a cooling system of a rectangular frame mounted with several flow channels with or without inserting an elliptical configuration of aluminum foam (ECAF) of different porosities located at various points in the channel(s) for dissipating heat generated by battery cells is conducted and reported. Operating under a steady state, a volumetric heat transfer of 3340135 WW/mm^3 equivalent to the total heat generated by the battery pack is transferred to the rectangular frame of the battery cooling system through the conducting element. The effect of the insert location (ECAF) in the fluid domain on the performance enhancement of the battery cooling system was analyzed. At the highest (0.9) porosity, an efficiency index (EI) of 93.4% and 63.7% for the battery cooling system (BCS) were obtained, respectively, compared to low (0.1) and medium (0.5) porosity of ECAF, specifically at a Reynolds number (Re) of 1155.4. Precisely at Re 2054.08, 21.17%, 14.58%, 11.46% and 24.14% of improved heat transfer rate density were recorded when three channels of parallel flow arrangement with ECAF of 0.9 porosity insert were mounted on the rectangular frame of the BCS compared to 1, 2, 4, and 5 channels. Regardless of the design, BCS with a counterflow arrangement achieved a slight thermal performance advantage over those with a parallel flow arrangement. Meanwhile, at Re 2054.08, 92.62% of the performance improvement was achieved with the counterflow BCS of three channels containing an ECAF with 0.9 porosity, compared to their counterpart without an insert. The results obtained show that the spacing arrangement of 2.0 mm is more beneficial to the BCS performance enhancement than 1.5 mm and 2.5 mm. However, a single ECAF of 0.9 porosity, located at one-eighth of the channel (L/8) towards the inlet, was found to yield the best thermal performance of the BCS compared to three ECAFs of 0.9 porosity, irrespective of the spacing arrangements. Hence, a counterflow cooling system with three channels, inserted with an ECAF of 0.9 porosity at L/8 to the inlet, ($NNNNGG_{\varepsilon\varepsilon=0.9CCCC}$) and documented as having the highest efficiency index, is found and recommended as the most suitable cooling system for the thermal control of the cylindrical lithium-ion battery pack.

Key Words: *Aluminum foam, battery pack, porosity, Electric vehicles*.

DESIGN AND DEVELOPMENT OF SOLAR STILL WITH VARIOUS MODIFICATIONS, APPLICATION TO SOUTH AFRICAN WEATHER SYSTEMS

Luke Solomons and Tunde Bello-Ochende

Department of Mechanical Engineering, University of Cape Town, Private Bag X3, Rondebosch, 7701, South Africa. **tunde.bello-ochende@uct.ac.za**

ABSTRACT

Access to clean water is crucial for human health and survival, yet much of the world continues to face severe water scarcity and contamination. In South Africa, population growth, industrial expansion, and failing wastewater infrastructure have intensified the pressure on freshwater resources. Despite the country's extensive coastline, seawater is unusable without treatment, and many inland areas still lack access to reliable, safe water supplies. This creates a clear need for simple, small-scale, low-cost purification technologies. Solar distillation is a promising off-grid water purification approach that requires only sunlight to operate. A solar still uses solar energy to evaporate contaminated water and then condense the vapor into clean water, essentially mimicking the natural rain cycle. However, conventional solar stills typically have low efficiency and yields, motivating research into modifications to increase performance. This paper examines the performance of a single-slope solar still under various modifications aimed at enhancing its yield for rural South African applications. Both passive and active changes were explored. Passive enhancements (no external power) included adding a mirror booster to increase incident solar radiation on the still, using a pyramid wick to increase the evaporative surface area, integrating internal aluminium fins to improve heat transfer, and introducing CuO nanofluids into the basin. The singular active modification involved incorporating a sun-tracking mechanism to maintain alignment with the sun. According to the literature review, such changes are known to enhance evaporation or condensation rates in solar stills. In the Python-based simulations, the unmodified reference still produced 6.95 L/m²/day of distilled water. The mirror booster proved to be the most effective improvement, increasing daily yield by 33.57%, while the pyramid wick raised yield by 21.00%. The sun-tracking system provided a moderate yield gain of 8.2%, while adding nanofluids resulted in a minor 3.9% improvement. By contrast, the inclusion of internal aluminium fins had essentially no effect on daily yield, with a percentage change of −0.14%. Overall, the results show that increasing solar energy capture and evaporative surface area using mirrors and a pyramid wick substantially improved still productivity. More advanced measures, such as nanofluids and active sun tracking, yielded only modest returns. The addition of fins provided a negligible benefit, slightly reducing yield due to the much larger thermal mass. The mirror booster emerged as the single most impactful modification, whereas fins were the least effective. These findings suggest that focusing on simple, low-cost enhancements can significantly boost solar distillation performance. These simple additions can make solar distillation more practical for deployment in water-scarce rural communities in South Africa.

Key words: *Solar still, Python, Solar distillation, Heat Transfer, active and passive system*

PERFORMANCE ENHANCEMENT OF R290 BASED COLD STORAGE SYSTEMS USING ADVANCED SUBCOOLING TECHNIQUES

Prosenjit Singha[a*], Kavan Utpal Lad[a], Mani Sankar Dasgupta[a]

f20230394@pilani.bits-pilani.ac.in
dasgupta@pilani.bits-pilani.ac.in
[a]Department of Mechanical Engineering, BITS Pilani, 333031, India, *Corresponding author:
prosenjit.singha@pilani.bits-pilani.ac.in

ABSTRACT

India's cold storage sector plays a critical role in reducing post-harvest losses, particularly for temperature-sensitive agricultural products such as potatoes. However, many existing facilities rely on high-GWP synthetic refrigerants like R404A or ammonia (NH_3)-based systems. While ammonia offers high efficiency, its use is restricted in several facilities due to safety and regulatory concerns. This work proposes an R290 (propane)-based cold storage system as a sustainable alternative to R404A in NH_3-restricted applications. The baseline R290 system is further enhanced through the integration of two advanced subcooling techniques—Integrated Mechanical Subcooling (IMS) and Economiser-based subcooling—which require minimal modifications and few additional components. These configurations are evaluated for their potential to improve cooling COP, reduce compressor power consumption, and enhance overall energy efficiency under Indian climatic conditions. In addition, economic feasibility is assessed through a payback period analysis based on component costs and energy savings, considering bin-hour temperature distributions across ambient conditions. The novelty of the work lies in providing a detailed comparative evaluation of multiple subcooling configurations applied to R290 systems, addressing the current lack of systematic performance comparison with widely used refrigerants. This study quantifies efficiency improvements and operational suitability relative to R404A and R717 for cold storage applications under high-ambient conditions. The results highlight that combining natural refrigerants, energy-efficient subcooling strategies, and renewable energy integration offers a scalable pathway for modernizing India's cold storage infrastructure. The approach reduces operational costs, lowers greenhouse gas emissions, and contributes to multiple UN Sustainable Development Goals, addressing both food security and climate change mitigation. The findings provide actionable insights for policymakers, equipment manufacturers, and cold storage operators seeking sustainable, cost-effective refrigeration solutions.

Key Words: *Techno-economic; Cold storage; COP; Subcooling techniques; Natural refrigerants.*

EXPERIMENTAL ANALYSIS AND DYNAMIC ENERGY MODEL OF A GAS LIQUID ENERGY STORAGE (GLES) FOR ELECTRIC AND THERMAL STORAGE

C. V. Fiorini[1], A. R. Massulli[1], G. Lo Basso[1] and A. Vallati[1]

[1] DIEE Department of Electrical and Energy Engineering, “Sapienza” University of Rome,
Via Eudossiana 18, 00184 Rome, Italy
costanzavittoria.fiorini@uniroma1.it

ABSTRACT

Energy storage systems have been identified as a pivotal strategy for integrating renewable sources, thereby ensuring energy stability and flexibility of the electricity grid. The present work analyses a Gas Liquid Energy Storage (GLES) prototype, an experimental mechanical storage system, by performing field tests and developing a dynamic simulation model. In detail, excess renewable electricity drives a volumetric gear pump for oleodynamic compression of gaseous N_2 within the cylinder. Similarly, during the expansion phase, the N_2 pressure gradient pushes the oil through the pump impellers, generating electricity again. Contrary to conventional Compressed Air Energy Storage (CAES) systems, this technology is based on a closed thermodynamic system. In this context, the small-scale combined production of electrical and thermal energy enabled by the GLES represents a key contribution of this work. The mathematical model implemented in MATLAB/Simulink has been validated by an experimental campaign. The latter consisted of temperature, pressure and volume measurements for three different oil flow rates (namely 1.8, 3.6 and 5 $L \cdot min^{-1}$) and two pre-charge pressures (i.e. 2.65 bar and 25.2 bar). As a result, absolute errors of ±1.2 °C and ±1.4 bar for temperature and pressure were observed, respectively. Moreover, an energy density greater than 0.32 $kWh \cdot m^{-3}$ was achieved and can be further increased by optimizing the compression phase. Round-trip efficiency, mechanical work and recovered heat were used as performance indicators, yielding a round-trip efficiency of 0.76 and a recovered heat during compression of 0.0051 kWh. Finally, the higher the oil flow rate, the lower the heat release. This result suggests the potential exploitation of the GLES for both electrical and thermal storage purposes. This work also establishes a foundation for further experimental analysis involving the replacement of N_2 with CO_2 and Kr, in order to increase the stored electricity due to their higher molecular weight.

Key words: *CAES, GLES, power to power, electrical storage, renewable energy storage, thermodynamic model.*

SYNERGISTIC EFFECT OF SURFACE MICROMACHINING AND Fe_3O_4 NANOPARTICLE CONCENTRATION ON POOL BOILING HEAT TRANSFER CHARACTERISTICS

Manisha[1], Surjo Das[1], Suvanjan Bhattacharyya[1], Abhishek Tandey[1]

[1]Department of Mechanical Engineering, Birla Institute of Technology and Science Pilani, Pilani Campus, Vidya Vihar, Pilani, Rajasthan - 333031, India

*Corresponding Author: suvanjan.bhattacharyya@pilani.bits-pilani.ac.in

ABSTRACT

In boiling process, the inclusion of ferrofluids as well as surface modification leads to a remarkable enhancement in both the Critical Heat Flux (CHF) and Heat Transfer Coefficients (HTC). Present study experimentally examines the pool boiling performance of Fe_3O_4/Water ferrofluid on modified copper surface at three different volume concentrations (0.01 %, 0.05 %, and 0.1 %). The ferrofluid was prepared by a two-step method using Fe_3O_4 nanoparticle, SDS and deionized (DI) water as a base fluid. The hole pattern was created on copper surface by micromachining. This surface modification improves surface roughness and wettability, which enhances nucleation site density and results as significant enhancement in CHF and HTC.

The pool boiling tests using three concentrations were conducted twice at saturated conditions and at atmospheric pressure to ensure results. This investigation divulged a notable improvement in both CHF and HTC for the ferrofluid using hole pattern surface as compared to base fluid and plane surface. The 0.01 vol % concentration showed maximum enhancement for CHF and maximum HTC was shown by 0.1 vol % concentration. On the basis of performed experiments it was concluded that using hole pattern surface and ferrofluid the CHF increases by 48% for 0.01 vol concentration as compared to flat surface. The improved boiling performance of hole pattern surface as compared to plane surface was due to increased number of active nucleation sites and ameliorated ferrofluid rewetting capability. The observed significant enhancement in pool boiling performance is mainly attributed to increased surface area which is provided by hole pattern and also due to faster bubble formation.

Overall, the amalgamated use of ferrofluid and micromachined hole pattern on copper surface indicated a synergistic effect by improving boiling heat transfer characteristics. These revelations highlight the potential of Fe_3O_4/ Water ferrofluids and machined surfaces for advanced thermal management applications such as cooling of electronic devices with high heat flux, heat removal in Nuclear Reactors and managing heat in fuel cells and batteries.

Key Words: *Critical Heat Flux, Heat Transfer Coefficients, Nanoparticles, Nanofluid, Fe_3O_4, Concentration, Volume.*

LAMINAR FLOW HEAT TRANSFER THROUGH A CIRCULAR TUBE USING AXIAL CORRUGATION AND TWISTED TAPES WITH AND WITHOUT OBLIQUE TEETH

Suvanjan Bhattacharyya

Department of Mechanical Engineering, Birla Institute of Technology and Science Pilani, Pilani Campus, Vidya Vihar, Pilani, Rajasthan - 333031, India.

*Corresponding Author: suvanjan.bhattacharyya@pilani.bits-pilani.ac.in

ABSTRACT

Laminar flow of viscous oil (183<Pr<502) of heat transfer through a circular tube using axial corrugation and twisted tapes with and without oblique teeth have been studied experimentally. The heat transfer and the pressure drop measurements have been taken in separate test sections. Heat transfer tests were carried out in electrically heated stainless steel channel incorporating uniform wall heat flux boundary conditions. Pressure drop tests were carried out in acrylic ducts. Correlations for predicting friction factor and Nusselt number have been developed and presented. The thermohydraulic performance was evaluated. This combined geometry of fins performs better than individual enhancement technique under consideration. Where pressure drop in a heat exchanger is small fraction of the total system pressure drop; this experimental research finding is useful in designing tubes carrying solar thermal mass of viscous oil in a parabolic trough solar collector used in environmentally sound and increasingly cost-effective solar thermal electric power plants. The result is also useful in designing heat exchangers used in process industries.

Key Words: *Convection, Laminar Flow, Twisted-Tape, Heat Transfer Augmentation, Swirl Flow, Uniform Wall Heat Flux*

THERMO-HYDRAULIC PERFORMANCE AND STATISTICAL ASSESSMENT OF TRANSITIONAL FLOW REGIMES IN INCLINED SOLAR AIR HEATERS FOR ENHANCED ENERGY CONVERSION

Devendra Kumar Vishwakarma[1], Andrea Vallati[2], Abdel El Abed[3], Rachid Bennacer[4], Suvanjan Bhattacharyya[5,*]

[1]Department of Mechanical Engineering, Manipal University Jaipur, Jaipur, Rajasthan, 303007, India.

[2]Department of Astronautical, Electrical and Energy Engineering, Sapienza University of Rome, Rome, Italy.

[3]Laboratoire Lumière, Matière et Interfaces (LuMIn), UMR 9024, Institut d'Alembert (IDA), ENS Paris-Saclay, 4, Avenue des Sciences, 91190, Gif-Sur-Yvette, France

[4]LMPS - Laboratoire de Mécanique Paris-Saclay, Université Paris-Saclay, Centrale Supélec, ENS Paris-Saclay, CNRS, 91190, Gif-Sur-Yvette, France

[5]Department of Mechanical Engineering, Birla Institute of Technology and Science Pilani, Pilani Campus, Vidya Vihar, Pilani, Rajasthan - 333031, India.

[*]Corresponding Author: suvanjan.bhattacharyya@pilani.bits-pilani.ac.in

ABSTRACT

This study presents an experimental investigation into the thermo-hydrodynamics performance of inclined solar air heaters operating in the transition flow regime (TranFR). Experiments were carried out using plain channels and channels fitted with ribbed wavy tape inserts at inclination angles of 15° and 30°. The effects of Reynolds number (Re), wave ratio (w), and applied heat flux (q) were systematically analyzed to determine their influence on heat transfer and pressure drop. Flow regime identification was achieved using line-fitting methods and temperature standard deviation analysis, establishing the onset and end of TranFR for different conditions. Results showed that the use of ribbed wavy tape significantly enhanced heat transfer by intensifying secondary flows and boundary layer disruption, with up to 200% increase in the Nusselt number observed in the TranFR compared to laminar (LaFR) conditions. Inclination angle was found to strongly influence thermal performance, with lower angles (15°) consistently yielding higher Nusselt numbers due to stronger buoyancy-driven effects. While heat transfer improved markedly, friction factor variations remained modest, indicating minimal penalties in pumping power. Novel correlations for both Nusselt number (Nu) and friction factor (f) were developed for the studied range, with high predictive accuracy ($R^2 > 0.95$). Statistical analysis (ANOVA) confirmed Reynolds number as the most dominant parameter, with significant contributions from heat flux, wave ratio, and inclination angle, alongside key non-linear and interaction effects. The findings demonstrate the viability of deliberately operating solar air heaters in the TranFR to balance enhanced heat transfer with acceptable pressure drops, offering new insights for the design of efficient thermal systems.

Key Words: *Mixed Convection; Transition Flow Regime (TranFR); Heat Transfer; Turbulator; Inclination.*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

MHD NATURAL CONVECTION INSIDE A SQUARE ENCLOSURE WITH POROUS PARTITION FILLED BY NANO-FLUID

Saddam Hossain Mullick[1], Pranab Kumar Kundu[2], Shantanu Datta[3] and Sourav Majumdar [3]

[1]Calcutta Institute of Technology, Uluberia, India, mullick014@gmail.com
[2]NIT Jamshedpur, Adityapur, Jharkhand, India, pranab.me@nitjsr.ac.in
[3]Guru Nanak Institute of Technology, Kolkata, West Bengal, India. sourav030192@gmail.com

ABSTRACT

In the current work, MHD natural convection inside an enclosure filled with nano-fluid and containing a porous partition has been critically investigated with the help of stream function, isotherms and Nusselt number which is essential for many engineering applications such as cooling of high energy system, crude oil production, cooling system of electronic devices. The principal objective of the current work is to analyse the heat flow mechanism inside enclosure with the application of porous partition. The parameters varied for the presents work is the width of the partition (0.1 to 0.3) and volume fraction of nanoparticles (0.1 to 0.3), while keeping Rayleigh (Ra), Darcy (Da) and Hartmann (Ha) numbers constant at 10^5, 10^{-2}.and 10^2, respectively. Governing equations are solved using Ansys fluent, a commercial software, while SIMPLE algorithm is employed to couple the velocity and pressure terms in the y-momentum equation. It is seen from the current work that the heat transfer mechanism is drastically altered by the porous partition width and volume fraction of the nanoparticles. Heat flow rate reduces with the width of porous partition whereas it enhances with volume fraction. Figure 1 represents the problem domain with boundary conditions. The bottom wall is heated uniformly whereas both the vertical walls are relatively cooler, while the top wall is kept insulated. The results also indicate that pattern of stream function and thermal penetration significantly alters with d/H. With the decrease in d/H, strength of the circulation enhances. This happens due to the reduction of resistance to fluid flow as the width of the porous partition decreases. Further, it has been observed that thermal penetration enhances with the reduction of the width of the partition.

Key Words: *Free Convection, Magnetohydrodynamic Fluid, Enhanced Heat Transfer, Numerical Analysis, Finite Volume Method.*

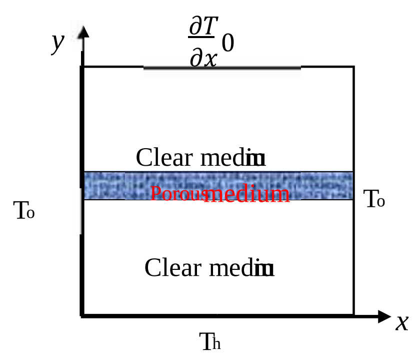


Figure 1. The problem domain with boundary condition

NATURAL CONVECTION INSIDE A SQUARE POROUS CAVITY WITH THERMALLY CONDUCTING EXTENDED SURFACES

Saddam Hossain Mullick[1], Yogeshwar Singh[2], Shantanu Datta[3] and Pranab Kumar Kundu[4]

[1] Calcutta Institute of Technology, Uluberia, India, mullick014@gmail.com
[2]MNNIT Allahabad, Prayagraj, Uttar Pradesh, India, yogeshwarsingh17@gmail.com
[3]Guru Nanak Institute of Technology, Kolkata, West Bengal, India. Shantanu.datta@gmail.com
[4]NIT Jamshedpur, Adityapur, Jharkhand, India, pranab.me@nitjsr.ac.in

ABSTRACT

The primary goal of this work is to analyse free convection numerically inside a square porous medium with fins added to the bottom heated wall, which could be one method to improve the rate of heat transfer in a closed environment. The work offers guidance for optimizing heat transfer inside porous enclosures, providing an insight of convection phenomena that helps in designing of real thermal management systems. Analyses of stream function graphs, isotherms, local and average Nusselt number are performed with the aid of ANSYS-FLUENT and MATLAB. The problem domain with boundary conditions is depicted in Fig. 1. The key purpose of the current reconsideration is to analyse the consequences of extended surface applied to the lower wall on the thermo-fluid characteristics inside the enclosure. These features are significant for the designing of sensitive engineering devices. The parameters varied for this work are Ra (10^4 to 10^6), Da (10^{-4} to 10^{-2}) and number of extended surfaces (0 to 7). The results indicate that the flow patterns drastically alter with Ra, Da and number of fins attached at the bottom surface of the cavity. It has been observed that the strength of circulation enhances with Ra and Da whereas it decreases with the increase in the number of fins. Furthermore, the average Nusselt number along the bottom wall (Table 1) seems to augment with Ra and Da due to increase of convection strength inside the cavity whereas it reduces with the increase in number of fins.

Key Words: *Free Convection, Enhanced Heat Transfer, Numerical Analysis, Finite Volume..*

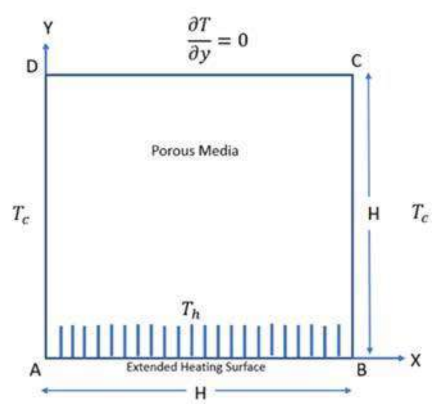


Figure 1. The problem domain with boundary condition

DYNAMIC ANALYSIS OF THE BLEED AIR ENVIRONMENT CONTROL SYSTEM FOR THE CIVIL AND FIGHTER AIRCRAFT

Chandra Shekhar Sharma[1*], Chennu Ranganayakulu[1]

[1]Mechnaical Engineering Department, Birla Institute of Technology and Science-Pilani, Pilani Campus, Rajasthan, India-333031

[*]p20220046@pilani.bits-pilani.ac.in

ABSTRACT

The Environment Control System (ECS) is a crucial component of an aircraft, as it regulates the pressure, temperature, and humidity within the cabin under varying altitude and Mach number conditions. The bleed air ECS comprises several components, including compact heat exchangers, a compressor, a turbine, a mechanical shaft, and a temperature control valve (TCV). According to the literature, there is a lack of studies focusing on the dynamic analysis of ECS, with most existing work limited to steady-state analysis and energy optimization. This study presents a novel system-level dynamic modeling framework for aircraft bleed air ECS by integrating MATLAB-developed component characteristic maps with Dymola-based transient simulations to capture coupled thermal–mechanical behavior and provide the first comparative dynamic response analysis between civil and fighter aircraft for early-stage design without experimental data. The dynamic analysis of the bleed air ECS is carried out using the ECS library in Dymola software for both civil and fighter aircraft configurations. The characteristic maps of key components are developed through steady-state analysis using MATLAB code under various flight scenarios and subsequently imported into the Dymola model to estimate the system-level dynamic response time in terms of thermal and mechanical performance. The validity of the model is supported through component-level validation reported in the literature. The results indicate that the system stabilizes more quickly in the civil aircraft case compared to the fighter aircraft case. The developed dynamic model can be effectively used for aerospace applications at the initial design stage, particularly in situations where experimental data are not available, thereby reducing development time and experimental costs. Finally, this article provides detailed component characteristic maps, a comparative dynamic investigation of civil and fighter aircraft cases, and system-level dynamic response times for the bleed air ECS.

Key Words: *Bleed air, Characteristics maps, Civil Aircraft, Dynamic analysis, Environment control system, Fighter Aircraft, Mechanical performance, Steady state analysis, Thermal performance.*

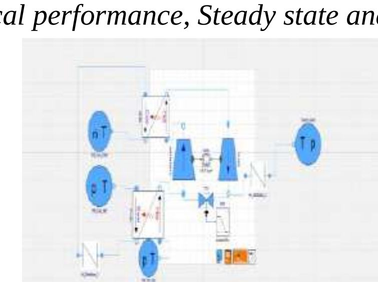


Figure 1: Dynamic Model of Simple Type Bleed ECS

DYNAMIC ANALYSIS OF FOUR-WHEEL BLEED-LESS AIR CYCLE AIRCRAFT ENVIRONMENT CONTROL SYSTEM

Chandra Shekhar Sharma[1*], Chennu Ranganayakulu[1]

[1]Mechnaical Engineering Department, Birla Institute of Technology and Science-Pilani, Pilani Campus, Rajasthan, India-333031

[*]p20220046@pilani.bits-pilani.ac.in

ABSTRACT

The most common technique for monitoring an aircraft's cabin pressure and temperature is the bleed environment control system (ECS). There is thrust loss and drag rise when extracting highpressure and high-temperature air from the engine outlet as bleed air. To overcome this limitation, an electric compressor–based system, known as a bleed-less ECS, is employed in modern aircraft ECS. This research presents the dynamic modeling of a four-wheel air cycle system, considering civil aircraft requirements for a bleed-less ECS. A dynamic model of a moderate-type, four-wheel bleed-less ECS with high-pressure water separation is developed using the commercial software Dymola. The novelty of this work lies in the development and dynamic validation of a four-wheel bleed-less air cycle ECS with high-pressure water separation, integrating compressor–turbine performance characteristics under realistic civil aircraft flight conditions. This study also discusses the performance characteristics of the compressor and turbine. The validity of the model is supported through component-level validation reported in the literature. To validate the dynamic model, steady-state analysis of the ECS is carried out using MATLAB code. During the dynamic analysis, the system's dynamic response is obtained for two operating conditions: 0 km altitude at a Mach number of 0.50 and 11 km altitude at a Mach number of 0.85.

Key Words: *Dynamic Simulation, Four-Wheel, Dymola, MATLAB, Air Cycle, Bleed-less, Civil Aircraft, High-Pressure Water Separation.*

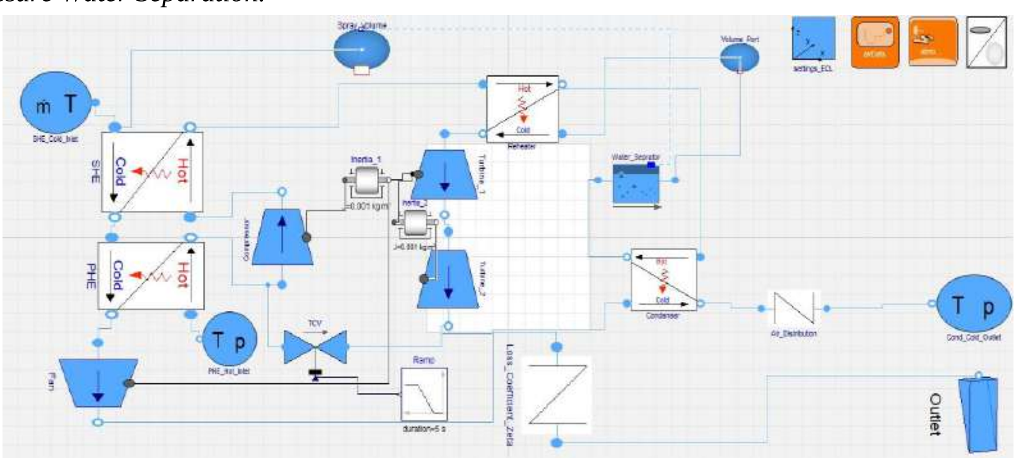


Figure 1: Dynamic Model of Four-wheel Bleed-less ECS

COMPUTATIONAL INVESTIGATION OF POST-STENT REMODELLING AND HYBRID BIODEGRADABLE STENT INNOVATION

Priyanshu Soni[1*], Abhra Bhattacharyya[1], Vedant Wanjari[1], Sanjay Kumar Rai[1], and B. V Rathish Kumar[2]

[1]School of Biomedical Engineering, Indian Institute of Technology (BHU) Varanasi, 221005,
[2]Department of Mathematics and Statistics, Indian Institute of Technology Kanpur, Kanpur
*Corresponding author: priyanshusoni.rs.bme22@itbhu.ac.in

ABSTRACT

The long-term success of cardiovascular stenting depends on maintaining favourable hemodynamics and minimizing post-implant complications such as restenosis, thrombosis, and late stent malposition. Conventional metallic stents, particularly those fabricated from *Ti6Al4V* alloys, remain permanently implanted, leading to chronic inflammatory responses, including damage to the endothelial layer of the artery and a mismatch in vascular compliance. This study introduces two key innovations. First, it presents a high-fidelity poststent hemodynamic analysis using Fluid–Structure Interaction (FSI), enabling accurate assessment of Wall Shear Stress (WSS), Oscillatory Shear Index (OSI), arterial deformation, and stress transfer at the stent artery interface, critical factors often overlooked in conventional CFD-only studies. Second, the work proposes a partially biodegradable hybrid cardiovascular stent, engineered with Zn–1Mg (≈ 70–75%) for controlled, polymer-regulated degradation and Ti6Al4V (approximately 25–30%) circumferential rings for sustained radial strength. This novel material configuration achieves superior compliance matching, reduces long-term stress concentration, and minimizes the risk of restenosis compared to fully metallic permanent stents. This study presents a comprehensive in-silico investigation of blood flow behaviour in stented arteries using FSI analysis, followed by the design and evaluation of a partially biodegradable hybrid stent. Three idealized arterial configurations were considered: (M_a) an artery with a standard stent, (M_b) a stented artery with 25% postimplant luminal blockage, and (M_c) a restenosis-free (0% blockage) stented artery. FSI simulations were conducted to assess WSS, OSI, deformation response, and stress concentration at the stent–artery interface. After developing the FSI model of poststent physics, a novel hybrid stent innovation was proposed, in which the inner and longitudinal segments are composed of *Zn–1Mg*, while the circumferential expandable rings are fabricated from *Ti6Al4V*. The *Zn–1Mg* segment offers controlled biodegradation, facilitated by a polymer (*PLLA*) coating applied to the stent's outer surface, which regulates the degradation rate. The results indicate that localized flow disturbances and elevated *WSS* gradients near the stent struts are primary contributors to neointimal hyperplasia and eventual stent failure. Meanwhile, the *Ti6Al4V* ring elements provide the necessary radial strength and structural stability throughout the arterial healing period. A comparative insilico analysis under physiological hemodynamic loading between the full metallic *Ti6Al4V* stent (S_1) and the hybrid stent incorporating a *Zn–1Mg* inner segment with *Ti6Al4V* anterior and posterior support rings (S_2) demonstrated a significant reduction in stress concentration, improved arterial compliance matching, and a lower likelihood of restenosis relative to the conventional *Ti6Al4V* design. This hybrid configuration provides adequate radial support during the critical healing period while gradually degrading thereafter, thereby reducing long-term vascular complications and improving overall patient outcomes.

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

THERMO-ELASTOHYDRODYNAMIC OPTIMIZATION OF SEAL GEOMETRY FOR ACHIEVING ZERO LEAKAGE IN HYDRAULIC CYLINDERS

Naveen Kumar Sain[1], Mani Sankar Dasgupta[1], Suvanjan Bhattacharyya[1], and Kundrapu Ayyappa Swamy[2]

[1]Department of Mechanical Engineering, Birla Institute of Technology and Science, Pilani, Pilani, India-333031, P20242202@pilani.bits-pilani.ac.in, dasgupta@pilani.bits-pilani.ac.in, suvanjan.bhattacharyya@pilani.bits-pilani.ac.in

[2]WIPRO Research, Bangalore, India-560100, ayyappa.kundrapu@wipro.com

ABSTRACT

Hydraulic cylinder reliability and environmental compliance are strongly constrained by dynamic seal leakage that arises from coupled pressure–temperature–deformation effects, rather than geometry alone. The novelty of this study is a geometry-to-performance optimization framework built on a coupled thermo-elastohydrodynamic (TEHD) model that simultaneously resolves thermal generation and transport (viscous dissipation–driven heating), links temperature rise to viscosity loss and film collapse, includes radial compression–induced contact mechanics, and tracks thermally accelerated material degradation to predict how “zero-leak” performance degrades over cycling. Unlike conventional isothermal EHD seal analyses, the proposed TEHD formulation quantifies how seal geometry influences leakage robustness under thermal transients, enabling direct multi-objective optimization. A parametric study and optimization are performed for the Oring-seal geometry, using contact width (b) and groove depth (d) as primary design variables. Seal performance is evaluated using quantitative indicators: minimum film thickness (h_{min}), net leakage per stroke (Q_{net}) predicted from Reynolds-based flow, friction force (F_f), and wear index (W*) derived from contact/shear loading. Simulations are conducted over representative pressure and temperature cycles relevant to reciprocating extraction and retraction strokes. The results show that geometry can be tuned to maintain a stable lubricating film and suppress leakage under thermal cycling. Optimized single-lip configurations preserve h_{min} above a critical threshold, reduce Q_{net} toward near-zero, and concurrently lower F_f and W* compared to baseline designs. The findings offer a theoretical framework for designing high-performance sealing systems in hydraulic cylinders, contributing to zero leak, enhanced operational efficiency, reduced maintenance, and environmental sustainability.

Key Words: *Hydraulic cylinder sealing; Thermal degradation; Seal contact width; Leakage; Film thickness*

FLOW OVER TWO TANDEM SQUARE CYLINDERS: CONTROL USING A DOWNSTREAM SPLITTER PLATE

Sarath R S[1], R Ajith Kumar[1] and K Suresh Kumar[2]

[1] Department of Mechanical Engineering, Amrita Vishwa Vidyapeetham, Amritapuri, India,
sarathrs@am.amrita.edu, r_ajithkumar@am.amrita.edu [2] RWDI, Dubai, UAE, suresh.kumar@rwdi.com

ABSTRACT

This study investigates laminar flow past two tandem square cylinders fitted with a downstream splitter plate to examine the effectiveness of the plate to control flow interference between the cylinders. The Reynolds number (defined based on the cylinder side dimension, D) was fixed at 150, and the centre-to-centre cylinder spacing was maintained at L/D = 5. The splitter gap ratio, defined as the distance between the downstream cylinder and the leading edge of the splitter plate (G/D), was varied from 1 to 6. Numerical simulations were performed using a finite-volume-based, incompressible flow solver (ANSYS Fluent). The key parameters analysed included the mean drag coefficient (C_d), rootmean-square lift coefficient ($C_{l,rms}$), and Strouhal number (St). The results showed that the splitter plate significantly altered the wake structure and force behaviour of the tandem configuration. At G/D = 1, the splitter completely suppresses vortex shedding, leading to a steady recirculating region with negligible lift oscillations ($C_{l,rms}$ = 0.055) and negative drag (thrust) on the downstream cylinder DC (C_d= -0.13). As the splitter was moved farther downstream, periodic shedding gradually reappeared. A critical spacing occurs at G/D ≈ 2, where the wake transitions from steady to unsteady, accompanied by a sharp increase in drag and lift fluctuations, and $St \approx 0.14$ for the DC. Beyond the critical spacing, the DC drag coefficient exceeds that of the upstream cylinder (UC) and a single cylinder (SC) (exceeds by 12.35% compared to SC), signifying the complete recovery of independent vortex shedding. This study identified the critical splitter gap that governs wake transition and quantified the drag recovery of the downstream cylinder beyond the upstream and single-cylinder levels, confirming the splitter plate's role in regulating vortex formation and suppressing wake-induced unsteadiness. These insights support the design of effective passive flow-control strategies for bluff-body aerodynamics, heat transfer systems, and vibration mitigation.

Key Words: *Tandem square cylinders, splitter plate, vortex shedding, wake structure, passive flow control*

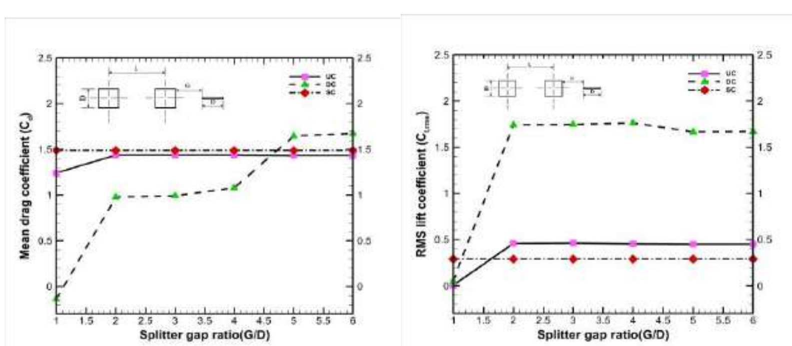


Figure 1: Variation of aerodynamic forces (C_d and $C_{l,rms}$) with Splitter gap ratio(G/D) compared with Single cylinder (SC)

NUMERICAL ASSESSMENT OF HEAT TRANSFER AND FLOW DYNAMICS IN NANOFLUID ENHANCED WAVY DUCTED PIN-FINNED SOLAR AIR HEATER

Ashish Babarao Khelkar[1*], Rajat Subhra Das[1], Ganesh Sahadeo Meshram[2], Gloria Biswal[2]

[1]National Institute of Technology Meghalaya, Saitsohpen Sohra, Meghalaya, India- 793108

[2]Indian Institute of Technology Kharagpur, West Bengal, India-721302, E-mail address:ashishkhelkar05@gmail.com

ABSTRACT

In this work, we conduct an in-depth CFD analysis to observe the flow and thermal characteristics of a solar air heater (SAH) equipped with artificial roughness, using air and Al_2O_3 nanofluids at 2% and 4% volume concentrations as working fluid. The SAH comprising of wavy duct equipped with pin fin type turbulators has been evaluated over a Reynolds number (*Re*) range of 3,500–16,000. The influence of base fluid properties, nanoparticles concentration, and duct configuration on flow behaviour, heat transfer augmentation, and frictional penalties has been assessed using Reynolds-Averaged Navier–Stokes (RANS) modeling with the RNG k–ε turbulence model. The heat flux of 1 kW/m^2 is applied on the upper surface of the absorber plate SAH. The outcomes show that the wavy profiled absorber plate within the flow domain demonstrated superior heat transfer characteristics compared to the flat plate duct due to extended heat transfer area and enhanced wall shear near the apex. The mixture of working fluid with the nanofluid's volume fraction continuously performed better than the air, enhancing the heat transfer coefficients and Nusselt numbers. In a pin-fin wavy duct, air + 4% Al_2O_3 yielded the maximum Nusselt number of 155 at *Re* of 16,000. Nevertheless, greater friction factors accompanied this performance improvement, particularly at lower Re and higher concentrations of nanoparticles. 3–4% Al_2O_3 provided the best compromise between flow resistance and heat transfer increase, according to a thorough thermo-hydraulic parameter analysis. In a wavy duct with air + 4% Al_2O_3 gives the maximum thermo-hydraulic performance parameter (THPP ≈ 1.7) at a Re of 16,000. The highest outlet temperature observed for the air with nanoparticles working fluid SAH at a Re of 3,500. The velocity, temperature, and pressure contours represent the significant change in flow structure due to nanoparticles and duct shape. The current study comes to the conclusion that significant performance enhancement in SAHs is made possible by the combination of air and AlO_3 nanofluids, as well as a pin-fin wavy duct shape. This opens the door to compact and thermally sustainable energy systems for solar thermal purposes.

Key Words: *Pin fins, Wavy duct, Al_2O_3 Nanofluids, Nusselt number, Friction factor*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

EFFICIENCY ENHANCEMENT OF PHOTO-VOLTAIC CELL USING NOVEL ETHANOL BASED PHASE CHANGE COOLING

Kavan Utpal Lad[a], Parth Uppal[a], P. Srinivasan[a]

f20230450@pilani.bits-pilani.ac.in psrinivasan@pilani.bits-pilani.ac.in
[a]Department of Mechanical Engineering, BITS Pilani, 333031, India,
*Corresponding author: f20230394@pilani.bits-pilani.ac.in

ABSTRACT

India's rapidly expanding solar energy sector faces a critical challenge in maintaining photovoltaic (PV) cell efficiency under high ambient temperatures, particularly in urban regions such as Mumbai where elevated thermal loads significantly reduce power output. Conventional cooling strategies often rely on forced air or water circulation, which add complexity and operational costs. This work proposes a novel ethanol-based phase change cooling system as a sustainable alternative for temperature regulation in PV modules. Ethanol, with its favorable thermophysical properties and low environmental impact, is employed as a phase change fluid (PCF) to absorb excess heat during peak solar irradiation, thereby stabilizing PV cell temperatures and improving energy yield. This study introduces a novel ethanol-based phase-change fluid (PCF) cooling system integrated directly into PV module encapsulation work to propose ethanol as the embedded PCF for rooftop PV and validate performance using multiphase conjugate heat-transfer and vapor–liquid simulations driven by Mumbai bin-hour climate data. In addition, economic feasibility is assessed through a payback period analysis based on ethanol containment costs, system integration requirements, and energy gains across seasonal bin-hour temperature distributions. The results highlight that combining phase change cooling with ethanol, advanced thermal management strategies, and renewable energy deployment offers a scalable pathway for modernizing India's solar infrastructure. The approach reduces operational losses, enhances PV reliability, and contributes directly to multiple UN Sustainable Development Goals, addressing both clean energy expansion and climate resilience. The findings provide actionable insights for policymakers, PV manufacturers, and solar farm operators seeking sustainable, cost-effective cooling solutions for high-temperature environments.

Key Words: *PV Cell; Multiphase; Cooling techniques; Phase change cooling.*

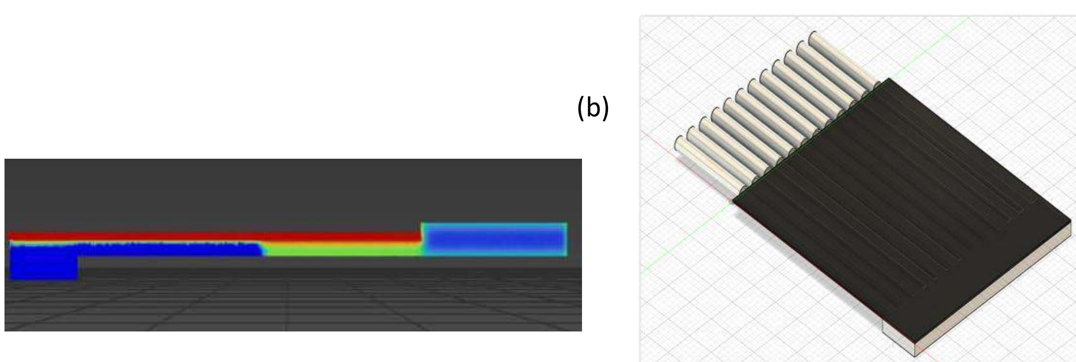


Figure 1: (a) Temperature contour of pipe cross-section and (b) Overall design

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

CHARACTERISTICS ANALYSIS OF SPHERICAL SENSIBLE THERMAL ENERGY STORAGE SYSTEMS FOR SOLAR ENERGY APPLICATIONS

Ram Kunwer, Dinesh Kumar, Gaurav Kumar Singh, Swapnil Sureshchandra Bhurat

DY Patil International University, Akurdi, Pune, India

ABSTRACT

Solar energy is a promising renewable energy resource for India's future, offering a sustainable solution to global energy demands. However, its intermittent nature makes energy storage a critical challenge for continuous and reliable utilization. Among various storage technologies, sensible heat storage (SHS) using solid or liquid media is a simple, cost-effective, and thermally stable option. Extensive research has been conducted on cubic and cylindrical sensible storage tanks due to their ease of fabrication and the development of a modeling approach. However, spherical sensible heat storage systems have received limited attention despite their superior surface-to-volume ratio and potential for minimized heat loss. The spherical TES system enhances thermal stratification and improves charging and discharging efficiency compared to a conventional cylindrical tank due to its reduced surface-to-volume ratio. This study focuses on analyzing the thermal characteristics of spherical SHS at different void fractions and mass flow rates using MATLAB. A two-dimensional energy equation was developed to simulate the charging and discharging characteristics in spherical SHS systems. The governing equation, based on the Schumann model, is solved using the finite volume method under appropriate initial and boundary conditions. The result shows the same thermocline behavior for both spherical and cylindrical tanks. The parametric study shows longer charging and discharging times to reach the same state for the smallest pebble diameter. Future work will focus on experimental validation, material optimization, and techno-economic assessment of a spherical SHS system.

Key Words: *Sensible heat storage, spherical tank, Schumann Model, Finite volume method*

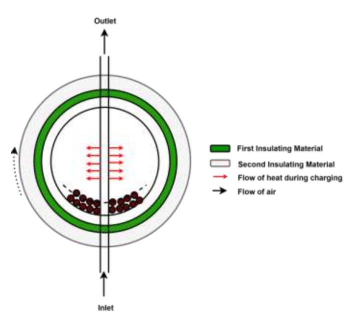


Figure 1: Physical model

CFD ANALYSIS OF A CYLINDRICAL LITHIUM-ION BATTERY WITH SPIRAL FINS

Alfred Francis J, Surya M and Deepakkumar Rajagopal*

Department of Thermal and Energy Engineering, School of Mechanical Engineering, Vellore Institute of Technology, Vellore, 632014, India.
*Corresponding Author E-mail: deepakkumar.r@vit.ac.in

ABSTRACT

This study presents a computational fluid dynamics (CFD) analysis of a cylindrical 21700 NMC811 lithium-ion battery equipped with spiral fins to enhance air-cooling performance. The novelty of the work is focused on identifying optimized fin location for the better cooling of the battery cell. Simulations were carried out in ANSYS Fluent at a Reynolds number of 1878 and a 2.5C discharge rate to evaluate the impact of fin configuration on heat dissipation. Four cases of fin coverage 25%, 50%, 75%, and 100% were analyzed, yielding cell temperatures of 61.7°C, 59.6°C, 59.1°C, and 59.6°C, respectively for 100% discharge condition. While the bare cell without fins reached a cell temperature of 65°C under identical operating conditions for 100% cell discharge. With a base temperature of 30°C, the 75% fin coverage showed the best performance, reducing the temperature by 5.9°C (16.7%) compared to the bare cell. At 50% cell discharge condition, the fin coverages of 25%, 50%, 75%, and 100% resulted in cell temperatures of 49.63°C, 48.62°C, 48.33°C, and 48.56°C, respectively, while the bare cell reached 51.1°C under the same conditions. This improvement results from the balanced interaction of fast and slow-moving air streams that enhance convective cooling while maintaining smooth airflow. The findings confirm that spiral fins significantly improve thermal regulation without significant pressure losses. Future work will be explored on fin geometry optimization and hybrid air–liquid cooling for advanced battery systems.

Keywords: *Cylindrical Battery; Spiral Fins; Air Cooling; Thermal Management; CFD Simulation*

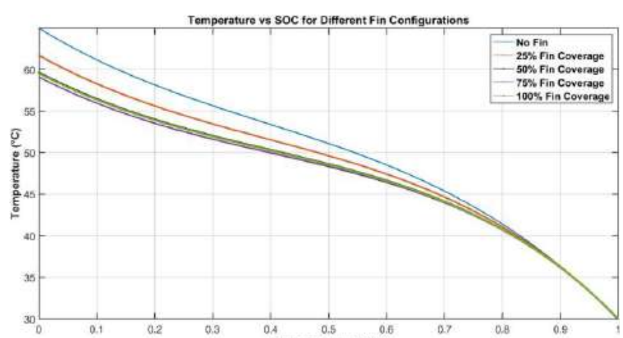


Fig.1.Effect of fin coverage on average temperature of battery cell with respect to state of discharge

NUMERICAL INVESTIGATION OF BATTERY THERMAL MANAGEMENT USING PCM WITH NANOADDITIVES

Surya M, Alfred Francis J and Deepakkumar Rajagopal*

Department of Thermal and Energy Engineering, School of Mechanical Engineering, Vellore Institute of Technology, Vellore, 632014, India.
*Corresponding Author E-mail: deepakkumar.r@vit.ac.in

ABSTRACT

A numerical investigation is carried out for the thermal management of an 18650 lithium-ion battery using ANSYS Fluent under transient condition. The model includes the battery cell, a PCM layer, and an aluminium enclosure. PCM thicknesses from 2 to 10 mm are analysed for the first 20 minutes to study their effect on temperature rise and thermal uniformity. Without PCM, the battery reaches 385.57K, while PCM reduces the temperature to 327.1 K for 2 mm, 314.1 K for 4 mm, 313.6 K for 6 mm, 310.27 K for 8 mm, and 311.68 K for 10 mm. The 8 mm layer gives the best performance with a 19.9% drop in maximum average temperature compared to without PCM and 5.12% compared to the 2 mm case. The novelty of this work lies in identifying the optimum PCM thickness through detailed transient analysis and extending the study to select the most suitable PCM for 18650-cell cooling. The extended analysis compares Eicosane, Paraffin, Capric Acid, and EG composite PCM based on achieving uniform cell temperature. Analysis of PCMs with Al_2O_3, Cu nanoparticles are studied to further improve heat absorption and uniformity. Overall, the results show that proper PCM material choice combined with optimum thickness greatly improves passive EV battery cooling.

Keywords: Battery Thermal Management; CFD Simulation; Phase Change Material (PCM); Nano-Enhanced PCM (NePCM); Computational Fluid Dynamics.

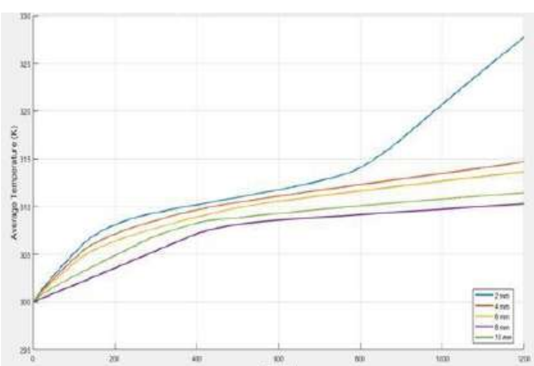
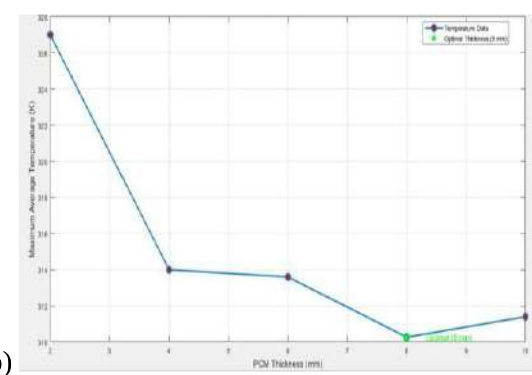
(a) 
(b) 

Fig. 1. Variation of (a) average temperature w.r.to time (b) maximum average temperature w.r.to PCM thickness

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

A NOVEL HYBRID JET-SWIRL COOLING SYSTEM FOR HEAT TRANSFER AUGMENTATION IN THE LEADING EDGE OF GAS TURBINE BLADE

Ayesha Wasim[1], Bhargava Teja[1] and Prasanth A K Lam[1]

[1]Computational Fluid Dynamics Laboratory, Department of Mechanical Engineering, NIT Warangal, Warangal, Telangana, India 506004

aw23meb0a29@student.nitw.ac.in, gb24mem1r25@student.nitw.ac.in, prasanth_anand@nitw.ac.in

ABSTRACT

Modern gas turbines operate at inlet temperatures far above the melting point of blade materials, making highly efficient internal cooling essential to ensure blade strength and durability. Among several techniques, such as jet impingement, swirl cooling, rib turbulators, pin fin, and dimpled surfaces, impingement and swirl cooling have proven to be one of the most effective methods for enhancing heat transfer, especially at the leading edge. The tangential injections of cooling with jet impingement generate strong vortices, improving near-wall mixing and producing a uniform circumferential cooling. Conventional jet impingement system provide strong local heat transfer but suffer from cross-flow interference and non-uniform temperature distribution[1], while swirl promotes mixing and coverage but suffers high pressure loss[2]. To overcome this issue, the present study proposes a combined hybrid jet-swirl cooling where jet impingement is introduced into a swirl chamber to utilize both advantages. The computational domain consisting of a rectangular jet plenum, connecting channels, and a semi-circular chamber. The domain is meshed with fine grids near the wall region, maintained $y^+<1$, ensuring accurate near-wall resolution. Numerical simulations were performed using the steady state Reynolds-Averaged Navier-Stokes (RANS) equation coupled with k–ω turbulent model by varying nozzle Aspect Ratio (AR). Air was used as the working fluid by varying the Reynolds numbers, and the target wall was isothermally maintained. Boundary conditions included a velocity inlet and a pressure outlet. Flow behaviour, pressure distribution, and heat transfer characteristics were analysed to evaluate the combined effect of jet impingement and swirl. Results indicated that the hybrid configuration with AR=5 produced a strong impinging jet along with controlled downstream swirl, leading to enhanced flow mixing and heat transfer uniformity. At a Reynolds number of 18500, the average Nusselt number increased by approximately 17% (Nu=197.3) compared to pure jet impingement (Nu=169.2), while the pressure drop decreased by about 14% relative to the pure swirl cooling. However, the Nusselt number of the hybrid cooling system remained lower than that of the swirl cooling (Nu = 206.2).

Key words: *Jet impingement, hybrid cooling, swirl cooling, heat transfer enhancement, vortex chamber, nozzle aspect ratio*

CFD ANALYSIS OF THERMAL PERFORMANCE OF A 9S2P LITHIUM-ION BATTERY PACK USING PHASE CHANGE MATERIAL (PCM) COOLING

Surekha Jayasurya [1], Vinay Pramod Hoshing[1] and Prasanth Anand Kumar Lam[1]

[1]Computatinal Fluid Dynamics Laboratory, Department of Mechanical Engineering, National Institute of Technology Warangal, Telangana - 506004, India,
js23meb0b42@student.nitw.ac.in, hv24mem1r24@student.nitw.ac.in, prasanth_anand@nitw.ac.in

ABSTRACT

A Battery Thermal Management System (BTMS) is an essential subsystem designed to monitor and regulate the temperature of battery packs to ensure safe, efficient, and reliable operation in electric vehicles (EVs). Lithium-ion batteries generate significant heat during charging, discharging, and high-power operation; without proper temperature control, this can lead to performance degradation, accelerated aging, or even thermal runaway. In this study, the thermal performance of a 9S2P lithium-ion battery pack integrated with Phase Change Material (PCM) as a passive cooling medium was analyzed. The battery pack was modeled and simulated in ANSYS Fluent, where the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm was used as the solver. Transient simulations were carried out to examine heat transfer, PCM melting behavior, and temperature distribution under high C-rate discharge conditions at different Depths of Discharge (DOD). The results showed that the maximum temperature fell within the 308–311 K range, indicating a slight but consistent decrease in peak temperature with increasing DOD due to reduced heat generation. Overall, the findings demonstrate that PCM integration effectively lowers peak cell temperatures and improves temperature uniformity, highlighting its potential to enhance passive BTMS performance in next-generation EV battery systems.

Key Words: *Battery Thermal Management System (BTMS), Passive Cooling, Phase Change Material (PCM), Electric vehicles (EV)*

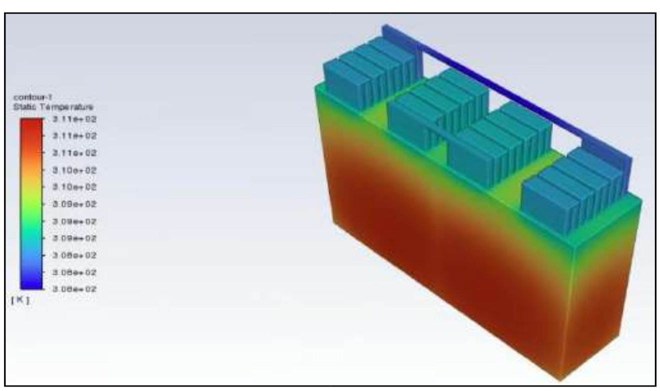


Figure 1: Temperature Contour of 9S2P Battery Pack

ROLE OF NANOPARTICLE IN IMPROVING HEAT TRANSFER PERFORMANCE OF AIR-BASED SOLAR THERMAL COLLECTOR EQUIPPED WITH VORTEX GENERATORS

Vishal Kumar Bhaskar[1*], Ashish B. Khelkar[2], Sachitananda Mishra[2], Rajat Subhra Das[2]

[1]Department of Mechanical Engineering, IIT (BHU), Varanasi, 221005, Uttar Pradesh, India

[2]Department of Mechanical Engineering, NIT Meghalaya, 793108, India

[*]Corresponding Author-bhashkarvishal386@gmail.com

ABSTRACT

A solar air heater (SAH) is an economical and environment friendly system that uses solar energy to heat ambient air for uses like space heating desalination, drying, and ventilation. Design changes and material improvements can greatly improve SAH's thermo-hydraulic performance. The present work conducts a comprehensive CFD analysis to examine the heat transfer increment and air flow parameters inside the SAH duct integrated with vortex generators. To optimize convective heat transmission and increase turbulence, parallelly aligned vortex generators (VGs) are added to the absorber plate. To further improve the absorber plate durability, thermal conductivity, and absorption capacity, the absorber plates may be coated with nanoparticles. Because of their improved heat transfer capabilities, common nanomaterials such as metal oxides, carbon-based, and metallic nanoparticles are taken into consideration. The vortex generator length, width, and height are regarded as 20, 8, and 10 mm, respectively. ANSYS 23 R1 software with RNG k- ε model was used for the entire computational investigation. The air duct test section measurements are maintained at 600 mm in length, 300 mm in width, and 25 mm in height, respectively. By changing the Reynolds number (*Re*) within the pertinent range of 6,000–18,000, the air duct performance is evaluated. When the axial pitch ratio and lateral spacing ratio are fixed at 1.76 and 0.05, respectively, the highest rate of convective heat transfer is recorded at a *Re* of 6000. The combination of vortex-generator and surface characteristics augmented by nanomaterials is expected to greatly increase the SAH overall thermal efficiency (55% - 65%) and Nusselt number (140-188) for a lower mass flow rate.

Key Words: *Heat Transfer, Vortex Generators (VGs), Natural Convection, Nanoparticles.*

SPATIAL DYNAMIC MODE DECOMPOSITION ANALYSIS IN THE WAKE OF AN ELLIPTICAL PLUNGING FOIL

Mohamed Aniffa S[1] and Sunetra Sarkar[1]

[1]Department of Aerospace Engineering, Indian Institute of Technology Madras, Chennai, India,
mohamedaniffas@gmail.com, sunetra.sarkar@gmail.com

ABSTRACT

A numerical investigation of the wake behind an elliptical plunging foil was conducted using an immersed boundary method solver. Although previous studies [1,2] suggest that variations in the spacing between consecutive vortices can cause wake deflection, the underlying mechanism remains unclear. This study investigates whether spatially growing disturbances are responsible for this deflection—an aspect not previously explored in the literature. Four non-dimensional plungevelocity cases (***kh*** = 0.5, 1, 1.5, 1.7) were analysed. Nearly symmetric wakes were observed for ***kh***= 0.5 and 1, whereas ***kh*** = 1.5 and 1.7 produced deflected wakes. The spatial frequencies associated with the vortex shedding for these cases were approximately 0.65, 0.53, 0.48, and 0.47, respectively. Figure 1(a) compares the maximum root-mean-squared resultant velocity ($\boldsymbol{u}_{\mathbf{R,rms,max}}$), showing downstream decay for undeflected wakes and downstream growth for deflected ones, indicating the presence of spatially growing disturbances. A Spatial-DMD analysis was performed to characterize spatially growing and decaying modes [3]. The spatial growth-rate spectrum in Figure 1(b) shows that the spatial frequency associated with the vortex shedding has a negative growth rate ($\boldsymbol{\alpha_r} < 0$) in all cases, whereas the deflected-wake cases (***kh*** = 1.5 and 1.7) exhibit an additional positive growth rate ($\boldsymbol{\alpha_r}$ = 0.2 and 0.16). These findings indicate that wake deflection may originate from spatially growing disturbances in the flow. Further quantitative analysis, including amplitude spectra and temporal frequencies of these modes, will be discussed in the conference/full paper.

Key Words: *flapping body wake dynamics, deflected wake, Spatial DMD.*

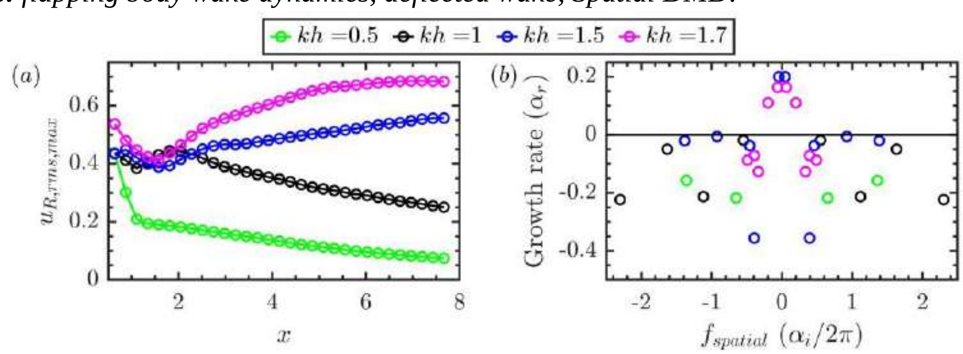


Figure 1: (a): Streamwise variation of the root mean squared value of resultant velocity for different *kh* values. (b): Spatial growth rate estimated from the spatial-DMD for different *kh* values.

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

PERFORMANCE ASSESSMENT OF DIFFERENT TYPES OF RIB-GROOVES ON THE RECEIVER PLATE OF A DOUBLE-PASS SOLAR AIR HEATER

Krittika Patwari[1], Ashish B. Khelkar[1] and Rajat Subhra Das[1]

[1]National Institute of Technology Meghalaya, Sohra-793108, p22me017@nitm.ac.in

ABSTRACT

The current investigation is carried out to investigate the thermohydraulic performance of three different geometrical shapes of the rib-groove arrangements on the receiver plate of a double-pass solar air heater. The rib-groove arrangement on the receiver plate of a doublepass solar air heater is a unique combination, and it gives an unusual ramification. The three designs are semi-circular, rectangular and triangular-shaped rib-groove arrangements. The designed configurations have identical geometrical dimensions, like the aspect ratio (W/H=10), pitch ratio (p/e=4.02), and blockage ratio (e/H=0.4), and they regulate under similar operating parameters to obtain a sense of clarity in obtaining a valid comparison of their thermal and hydraulic performance. The heat flux applied is kept constant at 880 W/m ^{2} with the variation in Reynolds number between the range of 3000-11000 to obtain and observe the contours of velocity, pressure drop, temperature, and turbulent kinetic energy over the range of different Reynolds numbers. The simulations are completed using ANSYS Fluent software, and to ensure the accuracy of the approach, it is validated with one of the numerical works of a previous researcher. The main focus of this study is the obtain the flow structure, heat transfer, pressure drop, and the overall thermohydraulic performance parameter (THPP) and the thermal efficiency for all three rib-groove geometries. The obtained results clearly revealed that the triangular-shaped rib-groove design showcased higher heat transfer enhancement, with a maximum Nusselt number enhancement of 2.5 at Re=3000. Balancing both the heat transfer and the pressure drop, the triangular-shaped rib groove achieved the maximum value of THPP, which is 2.9 at Reynolds number 3000.

Key Words: *DPSAH, Nu, f, THPP*

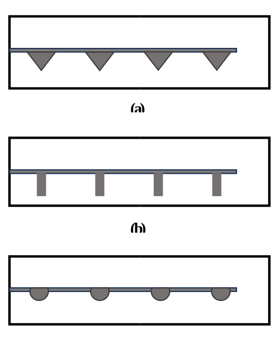


Figure 1: Geometrical design of different rib-groove (a) triangular (b) rectangular (c) semi-circular

REPORTING THERMAL PERFORMANCE OF STANDALONE SOLAR COOKING SYSTEMS USING COMMON ESTABLISHED SIMPLIFIED EVALUATION METHODS AND AN ENHANCED SIMPLIFIED EVALUATION METHOD

Celestino Ruivo[1,2] and Manoj Soni[3]

[1]Departamento de Engenharia Mecânica, Instituto Superior de Engenharia, Universidade do Algarve, Campus da Penha, 8005-139 Faro, Portugal, cruivo@ualg.pt

[2]ADAI, Departamento de Engenharia Mecânica, Rua Luís Reis Santos, Pólo II, 3030-788 Coimbra, Portugal

[3]BITS, Department of Mechanical Engineering, Birla Institute of Technology and Science, Pilani Campus, Vidya Vihar, 333031 Pilani, Rajasthan, India, mssoni@pilani.bits-pilani.ac.in

ABSTRACT

Most standalone solar thermal cooking devices function by focusing and capturing sunlight to produce heat inside a cooking container, allowing for tasks such as cooking and baking. These technologies hold great promise in sun-rich areas, as they can substantially decrease fuel use and environmental pollution linked to traditional cooking practices. Accurately reporting the thermal performance of a solar cooking device based on experimental results remains a significant challenge. The evaluation methods, commonly employed by various research groups and independent testers, are typically based on a load sensible heating test. The procedure defined in the ASAE S580.1 standard employs a relatively high load ratio, which is not representative of most of actual cooking practices. Moreover, the accuracy of the derived linear performance curve, expressed in terms of cooking power, may be questionable in some solar cooker designs and also because the procedure neglects experimental data from the initial period of the test. This procedure should also be revised to eliminate the scientific inconsistency when deriving the standardized cooking power. An alternative method focused on determining the optical thermal ratio of a solar cooker has been adopted also by some research teams. This method does not disregard the initial heating period of the load, whether water or an intermediate heat-transfer fluid. The method applies a fitting function to the load's temperature evolution to obtain a linear performance curve, usually expressed as thermal efficiency. In this study, thermal performance curves are obtained for representative solar cooker designs using both established simplified evaluation procedures and an enhanced simplified method that employs a more precise fitting function for the load temperature. The enhanced method yields a considerably more accurate performance curve, a benefit that becomes particularly significant for systems characterized by non-linear efficiency curves.

Key Words: *Solar cookers, Simplified methods, Enhanced method, Non-linear efficiency curve*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

EFFECTS OF METAL-OXIDE NANOPARTICLE-DOPED BIOCHAR ON MORPHOLOGY, CHEMICAL, AND THERMAL PROPERTIES OF PHASE CHANGE MATERIALS

Jyoti Moni Devi[1], Biplab Kumar Debnath[1] and Rajat Subhra Das[1]

[1] Department of Mechanical Engineering, National Institute of Technology Meghalaya, Sohra, India - 793108, p23me002@nitm.ac.in, biplab.debnath@nitm.ac.in, rajatsubhra@nitm.ac.in

The development of advanced thermal energy storage systems requires materials with enhanced heat transfer, stability, and energy conversion efficiency. Biochar, a renewable and carbon-rich material, has emerged as a promising additive to improve the thermal and structural performance of phase change materials (PCMs). In this study, biochar was synthesized through a green, costeffective route and doped with copper oxide (CuO) nanoparticles to produce engineered biochar (EB) with improved physicochemical properties. The engineered biochar was subsequently incorporated into lauric acid (LA)-based PCMs at varying nanoparticle concentrations (0 wt%, 1 wt%, 3 wt%, and 5 wt%) and termed as NB-PCM-0, NB-PCM-1, NB-PCM-3, NB-PCM-5 respectively. Characterization through FESEM and XRD confirmed uniform nanoparticle dispersion and surface modification, while BET and BJH analyses showed a 24.8% increase in pore volume and a 12.8% increase in surface area compared to raw biochar. The chemical properties of the samples were determined by FTIR spectroscopy. Differential Scanning Calorimetry revealed that the EB-based PCM exhibited an enhanced latent heat capacity of 119.3 J/g, representing a 13.1% improvement over pure PCM. The leakage test confirmed high thermal reliability and form stability during phase transitions, and the composite retained excellent thermal cycling performance after 100 heating–cooling cycles. These results demonstrate that CuO nanoparticle-doped biochar significantly enhances the thermal conductivity, energy storage efficiency, and stability of PCMs. The study highlights the potential of EB as a sustainable, cost-effective material for next-generation thermal energy storage applications.

Key Words: *Biochar, Nanoparticle, Phase change material, Thermal stability, Thermal storage capacity.*

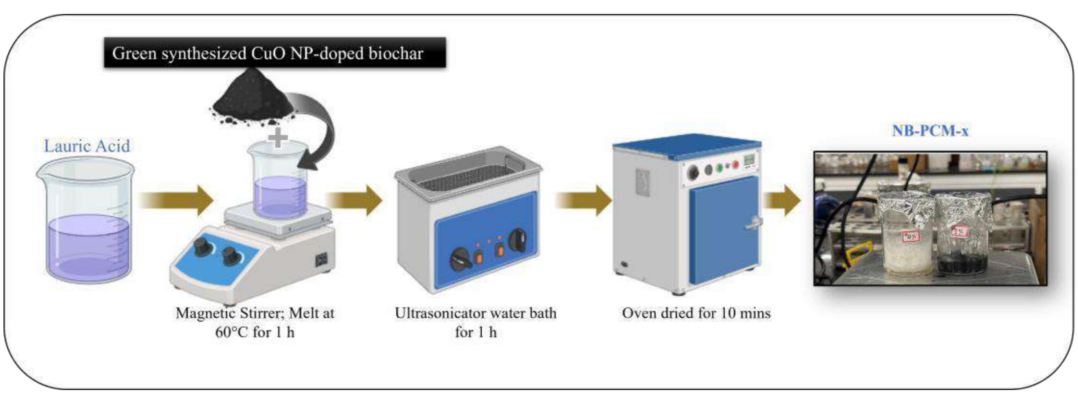


Figure 1: Schematic diagram of the synthesis of NP doped-biochar on LA based PCM composite

FLOW OVER TWO TANDEM CIRCULAR CYLINDERS: INVESTIGATION OF WAKE PARAMETERS

K Arunkumar[1], R Ajith Kumar[1] and K Suresh Kumar[2]

[1] Department of Mechanical Engineering, Amrita Vishwa Vidyapeetham, Amritapuri, India,
arunkumark@am.amrita.edu, r_ajithkumar@am.amrita.edu

[2]Vice President, RWDI, Dubai, UAE, suresh.kumar@rwdi.com

ABSTRACT

A numerical investigation of laminar flow over tandem circular cylinders is presented; the focus of this study was on the effects of spacing ratio (L/D = 2-6), at a Reynolds Number of 160, on both the wake characteristics and separation behaviour of the tandem cylinders. The novelty of this study lies in its detailed, spacingdependent characterization of wake and separation behaviour for laminar flow over tandem circular cylinders using a consistent finite-volume-based incompressible solver framework. Unlike many prior works that emphasize global force coefficients or higher-Reynolds-number regimes, this work systematically links spacing ratio to four key near-wake descriptors non-dimensional vortex formation length ($L_f{}^*$), recirculation length ($L^*{}_r$), wake width (W^*), and separation angle (θ) and interprets their joint evolution as evidence of a spacingcontrolled regime shift in wake interference and shedding regularity. As the spacing ratio (L/D) of the tandem cylinders increased so did the vortex formation length ($L^*{}_f$) and recirculation length ($L^*{}_r$), however, the wake width (W*) became wider and stabilized, which indicates that there is less wake interference as well as a transition from irregular to more regular shedding (Figure A). Similar to the trends for the vortex formation length and recirculation length, the separation angle (θ) of both cylinders also increased with increasing spacing and will stabilize past a L/D ratio= 4(Figure B), which indicates that the tandem cylinders are experiencing more symmetric vortex shedding behaviour with the upstream cylinder having a separation characteristic similar to an isolated cylinder. Therefore, these results indicate a significant regime shift due to spacing ratio, where the close spacing has a great deal of interaction resulting in an elongation of the wake and compression of the wake width; conversely, as spacing is increased the wake dynamics and separation behaviour of the tandem cylinders will exhibit more symmetry and thus offer valuable insight into design of structural components as well as control of fluid flow.

Key Words: *Tandem circular cylinders, vortex shedding, wake parameters, separation angle*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

THERMAL HYDRAULICS INVESTIGATION OF PASSIVE HEAT TRANSPORT SYSTEM

A.K. Chauhan, Dhrumil G., M. Rajendrakumar, R. Sureshkumar, K. Natesan

Thermal Hydraulics Section,
Thermal Hydraulics & High Temperature Analysis Division,
Nuclear Physics & Computational Engineering Group, Reactor Design Group,
Indira Gandhi Centre for Atomic Research, Chengalpet, Tamil Nadu, India – 603 102
amitchauhanmecher@gmail.com; *amitchauhan@igcar.gov.in

ABSTRACT

NACIE-UP (NAtural CIrculation Experiment–UPgraded) facility, located at the ENEA Brasimone Research Centre (Italy), is an experimental setup designed to investigate key phenomena in Heavy Liquid Metal (HLM) hydraulics. The primary loop consists of two vertical (riser and downcomer) and two horizontal pipes, as shown in Figure 1, and is coupled with a pressurized secondary water circuit for heat removal from the primary loop. The study focuses on the transition from forced to natural circulation in the NACIE-UP loop. As part of an international collaborative program, all participants are required to validate their numerical models and simulation codes through open-phase and blind-phase analyses. The present article discusses the ADP07 test conducted at the NACIE-UP facility. A 3D CFD simulation has been performed for the 19-pin bundle (heat source), while a system thermal–hydraulic (STH) analysis of the entire primary circuit has been carried out using FLOWNEX. The FLOWNEX model shows excellent agreement with experimental data (with less than 5 – 10% deviation), accurately capturing the forced-to-natural circulation transition, including inertia-driven flow decay, buoyancy recovery, and corresponding temperature evolution. The gas bubbler is modelled as pseudo pump, to drive the forced flow. The numerical modelling strategy is the novelty in the present work. Overall, the results demonstrate that FLOWNEX provides a reliable and computationally efficient alternative to conventional STH codes for HLM loops. Its flexibility and modular structure make it a valuable tool for safety analysis and design support of advanced liquid metal–cooled fast reactors.

Key Words: *System thermal hydraulics, Liquid metal, Natural convection, Passive heat transfer, Fast reactor,LBE.*

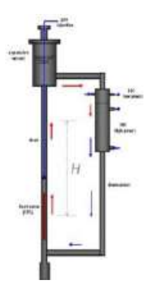


Figure 1: Thermocouples and transducers locations on the primary circuit

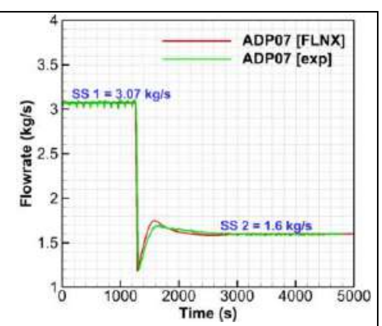


Figure 2: Evolution of LBE flow rate in the primary circuit during transition of ADP 07 test

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

CFD ANALYSIS OF A PCM-LIQUID HYBRID BATTERY THERMAL MANAGEMENT SYSTEM FOR ENHANCED THERMAL REGULATION

Rahul Ranjan, Rambabu Gupta, Devesh Kumar, Tanmay Dutta*

Department of Mechanical Engineering, Indian Institute of Technology (ISM) Dhanbad, Jharkhand – 826004, India.

ABSTRACT

This study presents a detailed Computational Fluid Dynamics (CFD) investigation of a hybrid Battery Thermal Management System (BTMS) that integrates phase change material (PCM) with liquid cooling to regulate the thermal behavior of a lithium-ion battery module. A three-dimensional numerical model comprising 24 cylindrical 18650 cells embedded in RT35HC PCM is developed using ANSYS Fluent. Straight microchannel liquid-cooling plates are positioned at a defined distance from the module, with water supplied through the inlets at 1 m/s. Simulations conducted at a 0.75C discharge rate reveal that PCM-only cooling leads to significant heat accumulation in the central region of the pack, resulting in a temperature rise of 327.66 K and a peak liquid fraction of 0.52. In contrast, the hybrid PCM–liquid cooling configuration markedly improves thermal dissipation, reducing the maximum pack temperature to 315.25 K and lowering the liquid fraction to 0.23. The liquefaction distribution indicates that melting occurs primarily in the outer regions of the pack. Overall, the hybrid system achieves a 44.86% reduction in temperature rise and a 51.06% decrease in liquid fraction compared with PCM-only cooling. These findings demonstrate the superior effectiveness and reliability of PCM-based liquid-cooled BTMS designs for advanced lithium-ion battery applications.

Key Words: *CFD, Phase change material (PCM), Battery Thermal Management System, liquid cooling.*

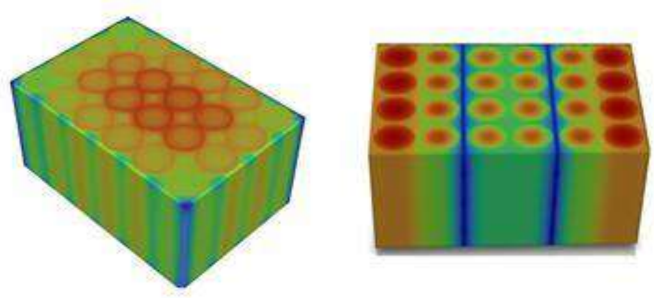


Figure 1 : Battery pack with PCM and Hybrid cooling.

IMPACT OF WAVY WALL CORRUGATIONS ON PERFORMANCE OF SOLAR AIR HEATER

Srijan Pramanik[1], Nirmalendu Biswas[1*], Aparesh Dutta[2], and Suvanjan Bhattacharyya[3]

[1]Department of Power Engineering, Jadavpur University, Kolkata 700106, India [b]Department of Mechanical Engineering, National Institute of Technology Durgapur, West Bengal, 713209, India

[c]Department of Mechanical Engineering, Birla Institute of Technology and Science Pilani, Pilani Campus, Vidya Vihar, Pilani, Rajasthan, India.

[*]corresponding author email id: *biswas.nirmalendu@gmail.com*

ABSTRACT

Passive heat transfer techniques through surfaces modification have shown significant potential for heat transfer enhancement in solar air heaters. Recent advances in the application of modified surfaces in the form of fins and corrugations have been comprehensively investigated as an alternative to the use of ribs, baffles, vortex generators, grooves and blocks for heat transfer enhancement. The current investigation aims to evaluate the thermo-hydraulic performance of a solar air heater (SAH) with a sinusoidal wavy-walled absorber plate. The wavy corrugations of the top-wall absorber plate acts as a modified surface for enhanced heat transfer and the back plate is non-corrugated. The governing equations are solved numerically using a finite volume solver for the turbulent flow regime (Re = 6000 – 18000) [1-2]. The SST k-ω model is used to solve the transport equations in the solver and the numerical results have been found to be in good agreement with former experimental studies. The impact of the sinusoidal wave amplitude and wave pitch on the average heat transfer characteristics, flow friction characteristics and thermohydraulic performance of the SAH is monitored and systematically presented in the form of contours of temperature, pressure, and velocity. The range of the roughness parameters that have been used in the simulation are as follows: amplitude of A = 1 mm, 2 mm, 4 mm and pitch of P = 10 mm, 20 mm, 30 mm, 40 mm. Subsequently, an optimal configuration of the roughness parameters for the proposed study of solar air heater is devised such that the surface coefficient of convective heat transfer is maximum with minimum pressure penalty [3-4]. Fig.1 illustrates the velocity, temperature, pressure contours for Re = 6000, A = 1 mm, P = 10 mm, Q = 1000 W/m^2.

Key words: *Solar air heater (SAH), Wavy wall corrugation, Thermo-hydraulic performance, Enhancement.*

THERMODYNAMIC ANALYSIS OF A COMPRESSION-ABSORPTION CASCADE REFRIGERATION CYCLE WITH PROPANE, N-BUTANE, AND CO2

Parv Goyal, Manoj Kumar Soni

Department of Mechanical Engineering, Birla Institute of Technology and Sciences, Pilani Pilani Campus, Rajasthan, 333031, India f20230157@pilani.bits-pilani.ac.in

ABSTRACT

This study presents a thermodynamic analysis of a compression–absorption cascade refrigeration system that have an ammonia–water (NH_3–H_2O) absorption cycle with a low-temperature vaporcompression cycle. A computational model was developed to quantify how two key operating parameters—the evaporator temperature (–30°C to –10°C) and the condenser temperature lift (ΔT=5°C to 15°C)—influence system performance. A main aspect of this work is the comparative analysis of three refrigerants Propane (R-290), n-Butane (R-600), and CO_2 (R-744) within the vapor-compression stage of a cascade system, an area not extensively explored in previous literature.The results reveal clear performance differences among the refrigerants. At an evaporator temperature of 265 K, the vapor-compression COP decreases with increasing condenser temperature lift for all three refrigerants. n-Butane delivers the highest COP values (≈7.6 to 5.2), closely followed by Propane (≈7.4 to 5.0), while CO_2 shows significantly lower performance (≈6.0 to 3.3). When these refrigerants are integrated into the full cascade system, using Propane or nButane can improve the overall COP by about **30–35%** compared to CO_2. These results emphasize how important the choice of refrigerant is and offer practical guidance for designing hybrid refrigeration systems that deliver higher efficiency.

Key Words: *Refrigeration; Cascade Cycle; Absorption; Propane; Butane; CO2*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

VOLTAGE-DRIVEN CAPILLARY FLOW IN A PARALLEL-PLATE CHANNEL VIA ELECTROWETTING OF COLLOIDAL LIQUIDS

Sumit Kumar

Department of Mechanical and Industrial Engineering,
Manipal Institute of Technology Bengaluru,
Manipal Academy of Higher Education (MAHE), Bengaluru, India
Email: sumitkumar.b@manipal.edu sumitkumar.besu2010@gmail.com

ABSTRACT

Flow through a narrow parallel plate channel is a ubiquitous and omnipresent phenomenon with potential applications in various fields, including liquid lenses, microswitches, and reflective displays. The motion of the liquid within such a narrow channel is primarily controlled by capillary forces and the wettability of the channel walls. The application of electrowetting in this channel provides an additional means of controlling the wettability of the channel wall by applying an electrical voltage. The combined role of electrowetting, along with capillarity in flow through the microchannel, makes it more suitable for various applications, ranging from lab-on-chip diagnostics to bioengineering. The effect of colloidal suspension on capillary rise remains unclear, despite several studies attempting to understand capillary flow behavior under electrowetting conditions. Therefore, this study examines the dynamics of capillary flow in a parallel plate channel containing a colloidal suspension liquid. Figure 1 shows the schematic of the system used for this study. A mathematical model has been developed to predict the equilibrium height of the meniscus height which can be written as $H\,(V_{eq}, \lambda)=\frac{2\lambda}{\rho g}\left(\cos\theta_0 + \frac{\varepsilon_0\varepsilon_r V^2}{2\delta\gamma} + \psi\lambda^{1/3}V\right)$
A theoretical model is further developed to understand the dynamics of capillary flow under a parallel plate channel via electrowetting of colloidal liquids. It has been noted that the capillary's meniscus height rises as both the concentration of colloidal particles and the applied voltage increase. Additionally, with a given voltage and colloidal particle concentration, the meniscus velocity first rises and then falls with time.

Key Words: *Electrowetting, Capillary rise, Colloidal suspension, Dielectric film*

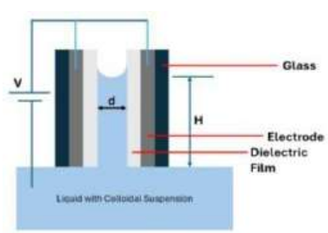


Fig. 1 Schematic of capillary flow between a parallel plate channel through electrowetting

COMPUTATIONAL INVESTIGATION ON THE THERMAL PERFORMANCE OF HEAT PIPE-FLAT PLATE COLLECTOR

Vinay Prakash Verma [1], Prasun Dutta [2], Vikas[3]

[1, 2]Department of Mechanical Engineering, SR University, Warangal, Telengana- 50637, India.

vinaypraverma@gmail.com[1], pd.iiest@gmail.com[2],vikas@sru.edu.in[3]

ABSTRACT

The Numerical simulation is being performed for this work to study behavior of flat plate solar collector attached with heat pipes with help of a closed two phase thermosyphon. ANSYS Fluent is being used to analyze the modelled collector using CFD simulation to assess the temperature field distribution on the absorber plate and evaluate the airflow characteristics within the air gap between the glass cover and absorber. The assessment shows that the temperature of working fluid raised by 7.7 K between inlet and outlet, showing good energy absorption and transfer. The model consists of copper absorber plate, glass cover, aluminum heat pipe and a rockwool insulation layer. Water is being used a working fluid with Solar radiation intensity of 1000 W/m^2 as well as an absorber radiation of 50 W on the plate is being used. A Fine polyhedral mesh, around 4.1 million cells are being applied to observe effect of conduction, convection and radiation. The properties of copper, insulation, glass given as per standard material properties thermal data, with inlet temperature of 300k and with mass flow rate of 0.0642 kg/s. The results of simulation shows that uniform velocity distribution of water, inside heat pipe and gradual rise of temperature along its length. The insulation layer having highest temperature near upper region, while bottom part comparatively cooler. The temperature contour along glass cover, absorber and insulation shows non linear variation due to radiative and localized heating. The overall results shows that simulated collector attained efficiency of 62.45% , is higher than 57%, observed on previous similar models. This result shows the better combination of better geometric configuration and material combination. The study shows capability of CFD model for predicting properties for showing behavior of coupled solar collector and provides strong input for advancement in this area.

Key words: *Flat plate solar collector, CFD simulation, Heat pipe, Absorber Plate, Thermal efficiency*

AIRFOIL SHAPE OPTIMIZATION USING GENETIC ALGORITHM

Navneet Manoj[1], Anwesha Mishra[2] and Pardha S. Gurugubelli[1]

[1]Department of Mechanical Engineering, Birla Institute of Technology & Science- Pilani, Hyderabad Campus, Hyderabad, Telangana, 500078, India

[2]Department of Chemical Engineering, Birla Institute of Technology & Science- Pilani, Hyderabad Campus, Hyderabad, Telangana, 500078, India

ABSTRACT

The aerodynamic efficiency of an airfoil is the ratio of its lift generated to the drag experienced. The factors influencing this includes Angle of Attack (AOA), airfoil's shape i.e. thickness, taper and camber, Angle of Twist, Aspect Ratio etc. We are developing a genetic algorithm for airfoil shape optimization that includes evolving airfoil geometries to achieve efficient aerodynamic performance, generally minimizing drag and maximizing lift to drag ratio. Assuming an Elliptical Lift Distribution, the aerodynamic forces are determined using two approaches, the Prandtl Lifting Line Theory and the vortex panel method. The optimization is carried out at a fixed target lift coefficient, wherein the total drag is the sum of viscous and induced drag components. Under the same aerodynamic load, each individual wing is evaluated to get the best optimized profile. The genetic algorithm updates a population of airfoil shapes over successive generations by applying selection, crossover, and mutation to the design parameters stated earlier. Within the code, the baseline airfoil provided by the user is resampled and modified through thickness and camber scaling and then launched into a 3D airfoil defined by taper, aspect ratio (AR) and angle of twist. The framework remains lightweight, general, and compatible with any airfoil input, making it suitable for rapid design and parametric studies.

Key Words: *Genetic Algorithm, Shape Optimization, Drag Minimization, Prandtl Lifting Line Theory, Vortex Panel Method*

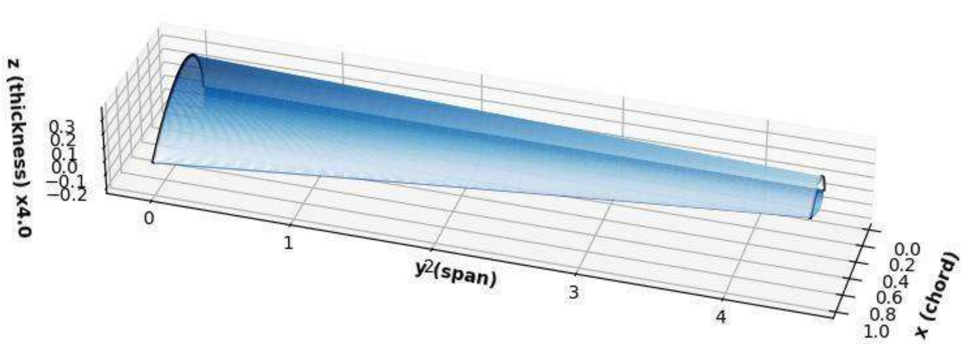


Figure 1: An optimized NACA-2415 airfoil

STUDY OF THERMOMECHANICAL BEHAVIOUR OF ROTARY KILNS

Vikas Grover[1], Sharad Srivastava[2] and Arun Jalan[1]

[1]Mechanical Department, BITS Pilani, Pilani Campus vikasagrover@gmail.com,

sharad_shrivastava@pilani.bits-pilani.ac.in,

arunjalan@pilani.bits-pilani.ac.in

ABSTRACT

This paper focuses on the thermomechanical analysis of a Rotary kiln used in the cement industry. A rotary kiln is subjected to both thermal and mechanical loads. Rotary kiln diameter varies from 2 m to 6.0 m and length from 30 m to 95 m. In this analysis, a rotary kiln with a diameter of 5.2 m and a length of 82 m inclined at 2.29 degrees is modelled with all major components, including the Kiln shell, tyre, rollers, gear, kiln refractory bricks, inlet, and outlet. To produce clinker, the kiln is heated to approximately 1400 °C and inside the kiln shell, temperatures vary from 600 °C to 1400 °C from inlet to outlet. Refractory lining or bricks are installed inside the kiln, serving as both a protection for the kiln shell and an insulating layer. Due to this, the kiln shell temperature gets reduced to approximately 250 to 350 °C. In addition to thermal loads resulting from high temperatures, mechanical loads such as the dead weight of rotating components, raw mix or feed material, axial thrust due to kiln inclination, and torque act on the entire kiln. Thermal load and mechanical loads are combined to get the actual equivalent stresses in the kiln shell.

In this paper, a thermomechanical stress analysis is carried out by first applying thermal loads to the kiln shell, which is mounted with refractory, and then the temperature profile is transferred to a static structural analysis. Mechanical loads are applied in Static structural analysis to get the results of Equivalent stresses and deformation in the kiln shell. Numerical calculations were also performed for comparison with the simulation results. Further, these results are also compared with actual data from cement plants.

Key Words: *Thermomechanical study, Numerical calculations, Thermal stresses, Equivalent stresses*

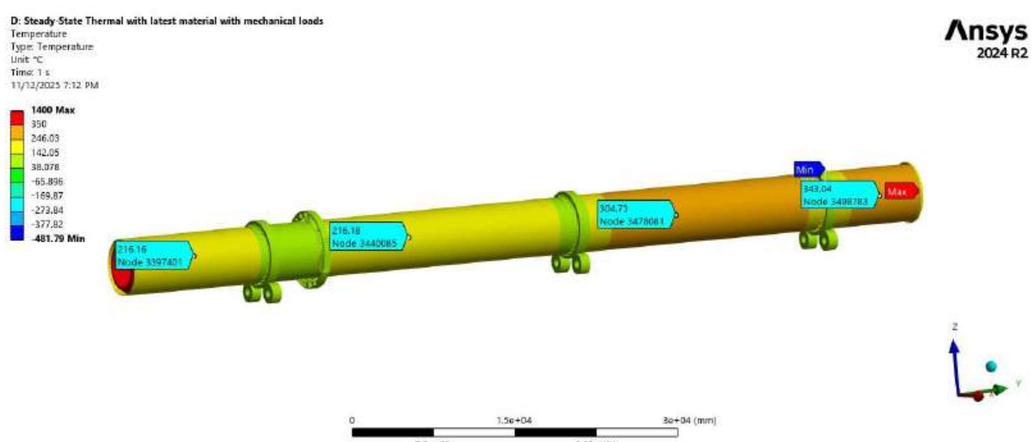


Figure 1: Temperature Profile of Rotary Kiln

NUMERICAL ANALYSIS OF A CO_2-BASED PRESSURE EXCHANGER DEVICE

Ayan Sengupta[1], Mani Sankar Dasgupta[2]

[1]Smart Building Lab, BITS Pilani, Pilani, Rajasthan-333031, meayansengupta4029@gmail.com [2]Smart Building Lab, BITS Pilani, Pilani, Rajasthan-333031, dasgupta@pilani.bits-pilani.ac.in

ABSTRACT

The study presents a numerical analysis to demonstrate the flow physics of a novel energy recovery device, the Pressure exchanger (PX), augmented in a vapour compression refrigeration system with natural refrigerant CO_2. The PX utilizes the energy of the supercritical CO_2 to compress the lowpressure flash gas, enabling more effective energy transfer between the fluid streams compared to conventional reciprocating compressors. The PX enables direct exposure of the high- and lowpressure CO_2, resulting in the propagation of compression and expansion waves, which simultaneously compress the high-pressure CO_2 and expand the low-pressure gas. The compression process is associated with negligible external electricity requirements, which tends to improve the energy efficiency of refrigeration systems. This also eliminates the need for a separate expansion device in refrigeration units. Unlike ejectors, the PX device is capable of generating higher pressure lifts and can even operate under sub-critical conditions, however, the flow phenomena in the PX are not very well established in reported literature. The novelty of the study lies in bridging this knowledge gap. Three-dimensional computational fluid dynamics (CFD) simulations are carried out to capture the propagation of pressure waves within the rotor ducts of the PX device. The shock propagation and its impact on the rotor speed of the PX are also explored to get a better insight into the novel device. The Equilibrium phase change model, in conjunction with the Span Wagner equation of state, is utilized to model flow dynamics within the PX and the simulations are carried out in ANSYS CFX. The simulations were conducted over a wide range of operating conditions commonly experienced in CO_2-based refrigeration systems. The high- and low-side pressures of CO_2 were varied in the range of 75 – 100 bar and 18 – 35 bar, respectively. The PX device was found capable of generating pressure lifts ranging from 26 to 47 bar, depending on the operating conditions. Although the efficiency of the PX (12 – 26%) was found to be comparable to that of CO_2 ejectors, the refrigerant handling capacity was relatively lower (9.6 – 10.7%). The Mach number of the normal shock was found in the range of 1.45 to 1.7. The findings from the numerical study are further validated with experimental data to ascertain the fidelity of the model.

Key Words: *Pressure Exchanger, Wave Propagation, Energy Recovery.*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

D3Q19-MRT LATTICE BOLTZMANN METHOD FOR UNBOUNDED FLOWS WITH OPENMP PARALLELIZATION

Vemu Sahiti[1], PS Gurugubelli[1] and VK Surasani[1]

[1]Birla Institute of Technology and Science, Hyderabad Campus, Shamirpet, p20200041@hyderabad.bits-pilani.ac.in

[1]Birla Institute of Technology and Science, Hyderabad Campus, Shamirpet, pardhasg@hyderabad.bits-pilani.ac.in

[1]Birla Institute of Technology and Science, Hyderabad Campus, Shamirpet, surasani@hyderabad.bits-pilani.ac.in

ABSTRACT

In the past few decades, lattice Boltzmann method (LBM) has emerged as one of the viable and promising computational tools for fluid flows by offering advantages like ease in implementation and parallelization. However, simulating unbounded flows has always been a challenge due to the difficulty in balancing the computational efficiency and physical accuracy, especially in the implementation of the far-field boundary conditions along with solver stability. For unbounded flows, where a large domain becomes essential to maintain the stability and accuracy, capabilities of parallelization become very valuable for practical simulations. While LBM has been extensively used for bounded flows, its application to unbounded external flows remains less explored. Present work demonstrates the OpenMP-parallel implementation of multiple relaxation time (MRT) collision model with D3Q19 lattice configuration for unbounded flow past a circular cylinder. The parallel implementation is achieved with OpenMP directives for shared-memory parallelization on multi-core architecture. The single-relaxation-time (SRT), or Bhatnagar-Gross-Krook (BGK), collision model employs a single relaxation rate for all discrete velocity directions. In contrast, the MRT collision model provides an enhanced numerical stability for the solver through independent relaxation rates for the same. At low Mach numbers, the BGK collision model suffers from stability constraints as the relaxation parameter approaches its lower limit. The present work addresses this gap by implementing D3Q19-MRT for unbounded flow simulations with stable far-field boundary conditions. Additionally, the OpenMP parallelization strategy is specifically optimized for the MRT collision step, which is computationally more intensive than BGK due to the transformation between velocity and moment spaces. D3Q19 provides a good balance between computational cost and accuracy when compared to higher-order velocity sets for laminar three-dimensional flows, making it particularly suitable for parallelized unbounded flow simulations. The methodology is validated for unbounded flow past a circular cylinder for Re 100, a wellestablished benchmark for three-dimensional external flows. The implementation achieves excellent agreement with literature: the predicted Strouhal number St 0.16 matches the benchmark value of 0.164-0.167, mean drag coefficient 1.505 falls within the expected range of 1.4-1.6 (threedimensional flows), and lift coefficient oscillations accurately capture vortex shedding dynamics. The OpenMP implementation on a 24-core workstation demonstrates the feasibility of this approach for engineering applications requiring moderate-scale 3D external flow simulations.

Key Words: *Lattice Boltzmann method, Parallel computing, D3Q19 lattice configuration, MRT collision model, Unbounded flows.*

RECONSTRUCTION AND VALIDATION OF A DATA-DRIVEN OPERATIONAL FRAMEWORK FOR HYDROGEN-INTEGRATED DISTRIBUTION NETWORKS

Shivesh Dwivedi[1], Suvanjan Bhattacharyya[1], Arun Kumar Choudhary[2]

[1]Department of Mechanical Engineering, Birla Institute of Technology and Science Pilani, Pilani Campus, 333031, shiveshdwivedi.23@gmail.com, suvanjan.bhattacharyya@pilani.bits-pilani.ac.in
[2]Ministry of New and Renewable Energy (MNRE), Government of India, New Delhi, 110003, akchoudhary.research@gmail.com

ABSTRACT

Green hydrogen systems require coordinated operation of electrolyzers, fuel cells, and storage units to ensure economic and grid supportive performance. Recent literature has proposed advanced learning-based frameworks, such as ACIVP (Active Constraint and Integer Variable Prediction), SSFR (Strategy Selection with Feasibility Ranking), and FSER (Feature Space Extension and Refinement), to improve decision making. However, these multistage architectures are technically intensive and difficult to reproduce without access to their proprietary datasets and intermediate workflow details.We reproduce a previously non-reproducible MEWH scheduling framework using a transparent, fully documented ML based surrogate modeling pipeline, replacing the original ACIVP–SSFR–FSER workflow with a reproducible data-driven approach. Thus, we present a framework for the optimal scheduling of Multi-Energy Water–Hydrogen (MEWH) systems (a nexus between Power, Water and Hydrogen Demands), addressing the transparency and replicability challenges found in prior literature. To overcome this barrier, we reconstruct the MEWH scheduling problem using only open-source tools, public datasets, and a documented ML workflow that enables full reproducibility of results. Using 90 days of publicly available wind generation and electrical load data, we solved a benchmark Mixed Integer Linear Programming (MILP) scheduling problem at 5-minute resolution, yielding 25,920 high fidelity operational samples representing electrolyzer operation, fuel cell usage, grid import/export, curtailment, and water flow. Based on these optimization derived trajectories, we developed a transparent surrogate modeling pipeline incorporating engineered features such as net excess power, positive ramp intensity, diurnal sinusoidal encodings, and water flow dynamics, which significantly enhance the predictive structure available to the ML models. Among the regressors, XGBoost models successfully replicated key power flow trajectories (e.g., electrolyzer power with $R^2 = 0.995$ and grid import with $R^2 \approx 1.00$), demonstrating near perfect fidelity to the optimization reference. For binary operational schedules, a surrogate classifier achieved an F1 score of 0.697 for the electrolyzer on/off trajectory, closely approximating the original MILP behavior across all 25,920 intervals. To ensure operational realism, we additionally incorporated a minimum on time filter and adaptive power thresholding, improving compatibility with physical start–stop constraints. The overall results confirm that the proposed framework can accurately emulate MEWH scheduling behavior without relying on opaque or unavailable components. This gives a robust foundation for further research on such hybrid systems, enabling transparent benchmarking, reproducible experimentation, and future comparisons under different conditions. The developed pipeline serves both as a scientific contribution in itself and as a reproducible baseline upon which additional innovations, such as new datasets, or modified MILP formulations, may be reliably evaluated.

Key Words: *Fuel Cell, Machine Learning, Energy Hub Operation, Switching Prediction, XGBoost.*

EXTENDING LAMB'S ACOUSTIC CUTOFF TO UNDERWATER ACOUSTICS: IMPLICATIONS FOR SOFAR CHANNEL PROPAGATION

Swati Routh[1], Anushka Sharma[1] and Saniya Lahoti[1]

[1] BITS Pilani, Dubai Campus, International Academic City, Dubai, United Arab Emirates

Email: swatirouth@dubai.bits-pilani.ac.in

ABSTRACT

Lamb first introduced the acoustic cutoff frequency for isothermal, stratified atmospheres— a key idea that sets limits on wave propagation though stratified environments. In this paper we extend Lamb's concept to the stratified non-isothermal oceans. The resulting cutoff isn't just a single number—it is inherently local, varying with temperature, density, and pressure profiles, and setting the minimum frequency required for acoustic waves to propagate at each depth. We investigate in this work how this local cutoff frequency can be derived from the ocean's layered sound-speed structure, focusing on the SOFAR channel as a case study. Our analysis shows that changes in the cutoff frequency help to define the acoustic boundaries of the channel itself. These boundaries decide which sound frequencies get trapped or reflected back and which escape, shaping how far and how efficiently sound can travel underwater. The cutoff–depth profile shows a rapid decrease in acoustic cutoff frequency from the highly stratified surface layer to deeper waters, eventually reaching a broad minimum at ~700–900 m—the depth of the deep-water SOFAR channel north of Hawai'i. This low-cutoff region corresponds to the sound-speed minimum and identifies the depth range where even extremely low-frequency sound can propagate.This is why the SOFAR channel acts as a natural acoustic waveguide, enabling long-range transmission.Our results therefore demonstrate that the SOFAR channel is not merely a sound-speed feature, but a dynamically significant zone defined by a locally minimal acoustic cutoff, governing long-range underwater communication and propagation in the Pacific. In short, this approach extends Lamb's theory to underwater acoustics and offers new insights into propagation in the deep ocean which is governed by local cutoff frequency.

Key Words: *Acoustic Cutoff Frequency, Underwater Acoustics, Sound Propagation*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

A COMPUTATIONAL STUDY OF CONVECTIVE HEAT TRANSFER OF SUPERCRITICAL CO2 IN PARALLEL AND COUNTER FLOW HEAT EXCHANGERS

Avipso Sinha1, Kaulik Das1, Aranyak Chakravarty2 and Koushik Ghosh1
1Department of Mechanical Engineering, Jadavpur University, Kolkata 700032 avipsosinha@gmail.com, kaulikdas2020@gmail.com, koushik.ghosh@jadavpuruniversity.in
2School of Nuclear Studies & Application, Jadavpur University, Kolkata 700106
aranyak.chakravarty@jadavpuruniversity.in

ABSTRACT

The study numerically investigates a vertical upward flow tube-in-tube counter-flow heat exchanger using supercritical carbon dioxide (sCO2) as the environment-friendly working fluid in the inner tube and cooling water in the outer annular jacket under isobaric conditions. The outer wall is considered adiabatic. The simulation was performed using ANSYS Fluent, employing the RNG k-ε model with Enhanced Wall Treatment for turbulence and the SIMPLEC algorithm for pressure-based solver. A key aspect of this study was accurately accounting for the severe thermo-physical property variations of sCO2 near the pseudo-critical point. Isobaric temperature-dependent data for 8.8 MPa were sourced from the NIST Standard Reference Database 23 (REFPROP) Version 7. These data were fitted with piece-wise high-order polynomial functions constructed via Python libraries and subsequently integrated into ANSYS Fluent via User-Defined Functions (UDFs) in C. The results demonstrate that the counter-flow arrangement achieves a consistently high rate of heat transfer, indicated by a significantly steeper temperature drop in the wall temperature profile when compared to the milder gradient of a parallel-flow arrangement. The heat transfer coefficient peaks at around 5400 W/m2 K near the pseudo-critical point because of Heat Transfer Enhancement (HTE). This research contributes a practical, robust simulation framework utilizing NIST Real Gas models and custom UDFs. This methodology offers an accurate digital modeling alternative to expensive 'trial-and-error' physical testing. Engineers can use this framework to safely predict sCO2 behavior, optimize heat exchanger size, and ultimately reduce the capital cost associated with next-generation power plants.

Key words: *Supercritical CO2, k- ε turbulence model, forced convective heat transfer, pseudo-phase transition, Non-linear Regression*

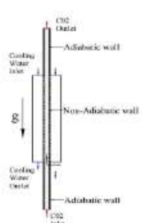


Fig.1: Setup for cross-flow heat exchanger

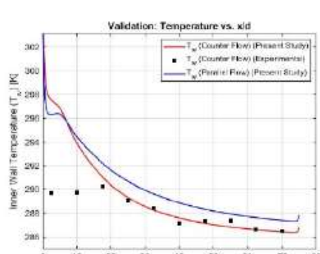


Fig.2: Validation for Wall temperature variation for Counter-flow and Parallel flow heat exchangers

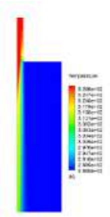


Fig.3: Temperature contour in Ansys Fluent for Parallel flow

STABILIZATION OF FREELY-FALLINGDISKS USING BRISTLE BRUSHES

Dhruwan Shah[1], Shashank Khurana[2] and Majid Hassan Khan[3]

[1,2,3] Department of Mechanical Engineering, Birla Institute of Technology and Science, Pilani, Dubai Campus,
Dubai International Academic City, Dubai, United Arab Emirates

Email: **f20220295@dubai.bits-pilani.ac.in**

ABSTRACT

With the increased use of underwater equipment and sensitive IoT marine monitoring systems, understanding their fluid-structure interactions can help predict its final descent in water. These fluid forces are generally dictated by factors like geometry, buoyancy, drag, and vortex formation. Interestingly, nature provides an inspiration for falling objects in fluids wherein organisms ranging from porous leaves to insects have evolved their body to manipulate flow fields around them. As a result, it stabilizes orientation during their gravity-driven descent. Earlier works established that flat-disks descend in different modes: steady fall, periodic zigzag, flutter, or chaotic tumble. Recent research focused on bristled designs, seen in insects like thrips' wings, and found the existence of a "virtual fluid barrier" that suppresses lateral oscillations. These structures use thin filaments to create symmetric vortex structures which stabilize the motion and reduce wake asymmetry. However, fall attitude is highly sensitive to shape, mass distribution, and release velocity. The objective of the current study is to investigate the effect of different flexible bristle distribution on the stabilization of circular disks during descent, relative to solid disks, in water.

A customized 3D-printed gripper mechanism was created and controlled using an Arduino-powered motor for consistent release from rest. This rig was mounted on a transparent acrylic tank filled with water at 23° to 25° C and its center was aligned to ensure the falling disks remained centrally. The test disks were 24 mm nickel-plated steel coins (mass = 6.1 ± 0.05 g and thickness = 1.8 mm) with flexible bristles of 0.5 mm polymer filament, each 10 mm long, attached at two, three, and six different configurations on the circumference of the coins. A freely-falling solid disk served as the baseline experiment. The framewise analysis of recorded footage yields lateral displacements, its angular fluctuations, and time of descent. Results showed a clear stabilization trend with the addition of bristles. The average descent velocity decreased progressively from 33.47 cm/s for the baseline to 13.28 cm/s for the six-bristle configurations. With higher circumferential bristle distribution, the increased drag from the bristles lowered terminal velocity and provided strong dampening against oscillations. Lateral drift showed clear dependence on the bristle distribution with solid disk demonstrating the largest lateral movement (21 cm) from its initial drop position, and reduced with the addition of bristles. Addition of bristles also decreased the angular instability. The solid disk tumbled and followed a zigzag path, showing an unsteady wake pattern. Two-bristled brushes attached diagonally to the disk decreased the amplitude of lateral zigzag trajectory. With three bristles, the descent became more stable as disks maintained vertical orientation with reduced zigzag motion. The six-bristled configuration showed most stable descent where the disk remained vertically aligned with negligible roll and nearly zero lateral drift. Overall, these findings corroborated the idea that a bio-inspired geometric alteration could stabilize the descent of freely falling objects. Exploring varying viscosities or disk sizes will further map stability regimes whereas hybrid/non-axisymmetric designs could extend these findings across marine systems.

Key words: *Bristled Disk, Wake Symmetry, Zigzag Motion, Tumble, Swirl*

CFD ANALYSIS OF FLOW AND HEAT TRANSFER CHARACTERISTICS IN A RECTANGULAR CHANNEL WITH DETACHED PIN FINS

Madhaneeswaran K.[1], Pradeep S. Jakkareddy[2]*, Shashi Kumar M.E.[1]

[1,2] Department of Mechanical Engineering, Amrita School of Engineering, Bengaluru, Amrita Vishwa Vidyapeetham, India–560035

*Corresponding author: js_pradeep@blr.amrita.edu

ABSTRACT

The present study involves computational investigation of turbulent flow and heat transfer in a rectangular channel equipped with three pin-fin configurations: (i) attached circular fins, (ii) detached circular fins with a uniform 6 mm clearance from the top wall, and (iii) detached elliptical fins of equivalent projected area to case (ii). The novelty of this research lies in the use of a detached elliptical fin geometry under the same flow and thermal conditions. This enables us to understand the geometric influence on the aerothermal performance of a heat transfer enhancement system. Simulations were performed in ANSYS Fluent software for inlet velocities of 6, 16, 23, & 30 m/s and with a constant heat flux of 1000 W/m² applied to the channel base and pin surfaces.

In all 3 cases, the friction factor consistently decreased with increasing Reynolds number, and the Nusselt number increased due to better inertial transport. At Re = 11203, the attached circular case gave the highest friction factor (0.695), whereas the detached circular and detached elliptical configurations reduced it to 0.394 (a 43% reduction) and 0.208 (a 70% reduction), respectively. This trend continued at the highest Reynolds number of 56018, with friction factors of 0.413 (attached circular), 0.282 (detached circular, a 32% reduction), and 0.125 (detached elliptical, a 70% reduction). These results show that detaching the pin from the top wall greatly reduces form drag by weakening horseshoe vortex & end wall interactions. The elliptical cross-section helps streamline the flow, producing the lowest pressure drop among all designs, indicating a smoother flow.

Heat transfer performance showed the expected relationship with fin geometry. For the attached circular configuration, the Nusselt number increased from 71.80 to 247.38 across the range of Reynolds number. Detached circular fins recorded values of 61.94 to 217.35, corresponding to reductions of about 14% at low Re and 12% at high Re when compared to attached fins. The detached elliptical pins yielded Nusselt numbers of 59.05 to 200.36, which represent decreases of 18% and 19% relative to attached circular fins at the lowest and highest Reynolds numbers, respectively. Even though the detached elliptical geometry has lower heat transfer, this reduction is significantly smaller than the corresponding drop in friction factor.As pumping power depends on pressure drop, the lower friction factor of detached elliptical fins significantly reduces energy consumption. Despite a small 15–20% drop in Nusselt number, they offer the best overall thermo-hydraulic balance. This makes detached elliptical fins more suitable for compact, low-power heat transfer enhancement systems.

Key words: *CFD, Rectangular channel. Pin fins, Detached fins, Heat transfer*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

COMPUTATIONAL ANALYSIS OF PULSATILE CARREAU BLOOD FLOW THROUGH AN AXISYMMETRIC TRIPLE-STENOSED POROUS ARTERY UNDER MAGNETOHYDRODYNAMIC EFFECTS

Priyanga R[1] and Girija Bai H[2]

1Research Scholar, Department of Mathematics, Sathyabama Institute of Science and Technology, Chennai, India, priyangaravi6@gmail.com

2Assistant Professor, Department of Mathematics, Sathyabama Institute of Science and Technology, Chennai, India, girijanameprakash@gmail.com

ABSTRACT

This study presents a detailed computational fluid dynamics (CFD) investigation of pulsatile, non- Newtonian blood flow through an axisymmetric triple-stenosed porous artery subjected to magnetohydrodynamic (MHD) effects. Blood rheology is modelled using the Carreau model to represent its shear-thinning behaviour, while Lorentz forces induced by a transverse magnetic field are incorporated via a User-Defined Function (UDF) in ANSYS Fluent 2025 R1. The combined influence of stenosis severity (· = 10%, 30%, 40%) and magnetic interaction parameter (M = 5, 20, 40) on the hemodynamic parameters such as axial velocity, wall shear stress (WSS) and pressure coefficient Cp are systematically examined. A mesh-independence study (GCI = 0.79%) and comparison with published CFD–PIV and analytical data (error < 4%) ensure numerical reliability.

Quantitative results show that increasing stenosis severity from 10% to 40% elevates peak axial velocity by ~48%, WSS by ~62%, and Cp by ~41% near the stenotic throats. Increasing magnetic strength from M = 5 to 40 produces a 15–28% decrease in axial velocity due to Lorentz damping, accompanied by 10–22% increases in WSS and Cp. These trends are reflected consistently in velocity contours, WSS maps and Cp distributions.

The novelty of this work lies in the combined modelling of triple stenosis, Carreau rheology, porous arterial walls and MHD modulation, an interaction not previously explored in a unified CFD framework. The findings provide clinically relevant insights into flow acceleration, shear stress amplification and pressure buildup in multi-stenosed arteries, with potential implications for magnetic-field-assisted therapy, thrombosis control and stent optimization.

Key Words: *CFD, Carreau model, MHD, Pulsatile Flow, Porous artery, Stenosis.*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

SHAPE MATTERS FOR ENHANCING PCM-BASED THERMAL ENERGY STORAGE

Anjan Nandi[1], and Nirmalendu Biswas[1]

[1]Department of Power Engineering, Jadavpur University, Kolkata 700106, India
Email: anjannandi6@gmail.com

ABSTRACT

Thermal energy storage systems based on phase change materials (PCMs) are very sensitive to the geometry of their enclosures with respect to melting dynamics, heat transmission behavior, and overall performance. The impact of geometric alterations on the thermal response of paraffin-based PCM (P-66) is investigated in this paper via a comprehensive numerical examination. We create and test a two-dimensional transient enthalpy-porosity model to represent the melting process in different geometric and operational settings. Three different enclosure structures are tested in the simulations with a heat transfer fluid (HTF) temperature of 85 °C and a Reynolds number (Re) of 1000. The results show that the melting rate and energy storage efficiency are both greatly improved by geometric optimization. Case 2, out of the three geometries evaluated, showed a significant increase in energy storage rate and melted almost 51.34% quicker than the usual straight-walled container (Case 1). Accelerated melting was also achieved at higher Reynolds numbers and higher HTF temperatures; however, thermal homogeneity was somewhat compromised. The results show that PCM-based latent heat energy storage may be significantly improved by combining optimal geometry improvement. For renewable and industrial thermal management systems, this integrated method provides a realistic solution to build small, high-performance, energy-efficient storage modules.

Key Words: *Heat Transfer, Phase Change Material (PCM), Thermal Energy Storage (TES), Geometry Optimization and Melting Enhancement*

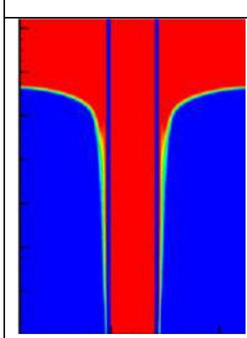
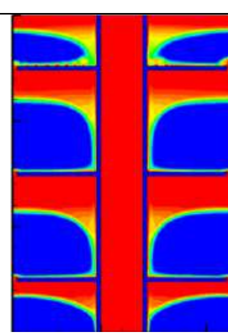
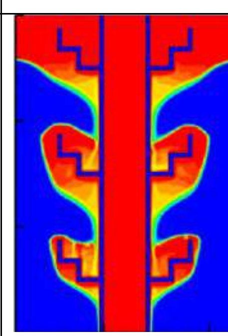
Case 1	***Case 2***	***Case 3***
		

Figure 1: Image of three different cases with PCM P-66 at Re = 1000 and T =85°C

ENHANCING MELTING DYNAMICS OF PHASE CHANGE MATERIAL BASED LATENT HEAT ENERGY STORAGE SYSTEMS USING WAVY CHANNEL

Dipankar Paul[1], Anjan Nandi[1], and Nirmalendu Biswas[1,*]
[1]Department of Power Engineering, Jadavpur University, Kolkata 700106, India
**email: biswas.nirmalendu@gmail.com*

ABSTRACT

This work is an analysis of the melting aspect of a vertical multi-channel Latent Heat Energy Storage System (LHTES), containing pure paraffin (RT-65) phase change material, which has been numerically studied. The fluid is water at 70°C as a heat transfer fluid (HTF) running continuously in uniform upward flow from the bottom inlet to the top outlet continuously with laminar behaviour. Finite Volume simulations are used to compare three geometries with the same total HTF-PCM cross-sectional area: Case 1 -with a straight HTF channel alongside PCM; wavy or sinusoidal HTF channel along with PCM; in Case 2- with and wavy central HTF path surrounded by left and right side PCM wavy channels, as illustrated in Case 3. Fig. 1 illustrates the Temperature and liquid fraction contours for Case 1, Case 2 and Case 3 with time *t* (min). Compared with Case 1, the wavy channel of Case 2 can shorten the complete melting time by 31.6% (from 12804 s to 8762 s) due to a larger HTF–PCM interfacial area. Case 3 -double partition layout is the best performing configuration in thermal terms, with a time to melt of $\sim$ 4004 s, which is ≈ 68.7% faster than Case 1 and ≈ 54.3% faster than Case 2. Enhancements result from the reduction of conductive lengths inside the PCM and more uniform temperature distributions that inhibit slow-melting areas. For a given HTF-PCM area budget, these results show that surrounding the HTF by divided PCM channels is the most efficient means of thermal management and charging for small footprint paraffin (RT 65) based LHTES modules.

Key Words: *Phase change material (PCM);Latent heat energy storage;Wavy channel;Heat transfer;Energy efficiency.*

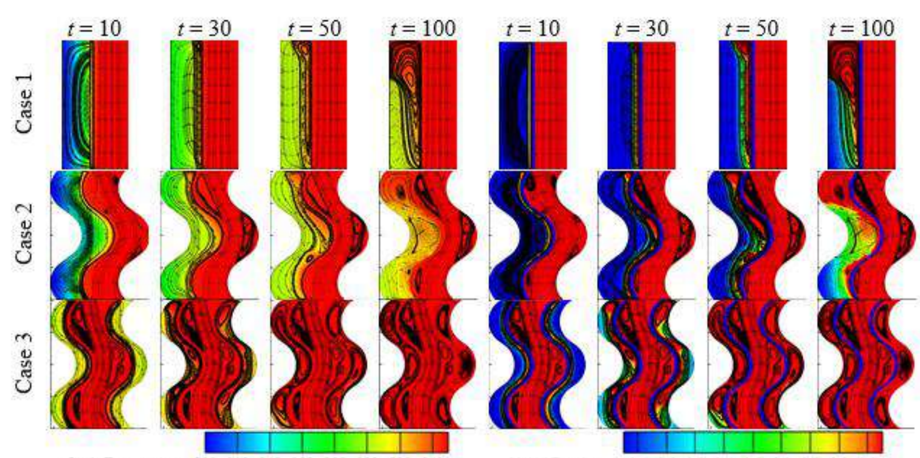


Figure 1: Temperature and liquid fraction contours for Case 1, Case 2 and Case 3 with time t (min).

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

HYDRODYNAMICS OF DROPLET INSIDE A TRIFURCATED MICROCHANNEL UNDER PRESSURE DRIVEN FLOW

Savnav Huda[1*], Deepanjan Das[1,2], Ranjit Barua[2], and Nirmalendu Biswas[1]

[1]Department of Power Engineering, Jadavpur University, Kolkata 700106, India
[2]Department of Mechanical Eng., OmDayal Group of Institutions, Howrah 711316, India
[*]corresponding author email id: *biswas.nirmalendu@gmail.com*

ABSTRACT

Droplet transport through branched microchannels plays a central role in microfluidic operations such as sorting, splitting, encapsulation, and biochemical analysis. In this study, we investigate the hydrodynamics of a single droplet propagating through a microchannel network consisting of three downstream branches under pressure-driven flow. Using a combination of numerical simulations and scaling analysis, we examine how geometric asymmetry, branch angle, and flow-rate distribution influence droplet trajectory, deformation, and breakup behavior. The interplay between viscous stresses, capillary forces, and confinement leads to distinct regimes of droplet dynamics, ranging from symmetric splitting to biased migration and complete diversion into a preferred branch. Results show that the droplet's entry position and size relative to channel width critically determine the pressure imbalance across the interfaces, which in turn governs branch selection. Higher capillary numbers promote elongated droplet shapes and facilitate droplet partitioning among the branches, whereas lower capillary numbers favor intact transit with minimal deformation. We further characterize the local flow perturbations induced by the droplet, highlighting their role in altering downstream resistance and feedback-driven path switching. The findings provide physical insights and design guidelines for multi-branch microfluidic networks aimed at controlled droplet routing, high-throughput assays, and precise flow manipulation. Our results further characterize transient flow feedback induced by the droplet, which changes downstream hydraulic resistance and can trigger dynamic path switching. These insights provide design guidelines for multi-branch microfluidic networks aimed at controlled droplet routing, high-throughput assays, and parallel biochemical processing.

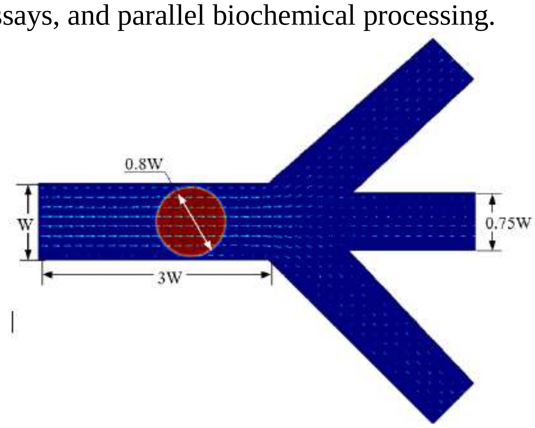


Fig. 1. Schematic computational domain

Key words: *Droplet, Capillary number, hydrodynamics, Poiseuille flow, trifucated microchannel*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

A PINN-GWO FRAMEWORK FOR ENHANCING MELTING DYNAMICS OF PCM IN FIN-ASSISTED TRAPEZOIDAL THERMAL SYSTEM

Om Karmakar[1], Anjan Nandi[1], Jotiraditya Banerjee[1], Dipankar Paul[1] and Nirmalendu Biswas[1,*]

[1]Department of Power Engineering, Jadavpur University, Kolkata 700106, India

[*]corresponding author email id: *biswas.nirmalendu@gmail.com*

ABSTRACT

The shape of the container and the arrangement of the fins have a big impact on how well phase change materials (PCMs) can store thermal energy. This work focuses on enhancing the melting performance of P-58 inside two-dimensional trapezoidal enclosures, characterized by a constant top surface length of 80 mm, while systematically varying the bottom length and depth. Square and rectangular shapes are seen as special examples of the trapezoidal shape, and the overall fin surface area is maintained the same to focus on geometric aspects as shown in Figure 1. A chosen set of computational fluid dynamics (CFD) simulations is executed to provide reference data about melting time, liquid percentage, and thermal efficiency for diverse geometries. A Physics-Informed Neural Network (PINN) is trained using the simulation data. The PINN uses the governing equations for heat transport and phase change as soft physical constraints. This method makes it possible to make accurate and physically consistent forecasts without having to do a lot of CFD calculations. The trained PINN works with the Grey Wolf Optimization (GWO) algorithm to find the best combination of geometric parameters, such as the length, depth, and number of fins on the vessel's bottom that will provide the best liquid-to-time ratio while the heat flow is constant. Sensitivity analysis of the trained model guarantees interpretability and determines the factors that most significantly affect thermal performance. The suggested PINN-GWO architecture provides a quick, physics-consistent, and easy-to-understand way to improve the shapes of PCM enclosures used for storing thermal energy. The model makes very accurate predictions, with $R^2 = 0.998$ and MAE ≈ 0.012.

Key Words: *Phase change materials (PCM), Physics-informed neural networks (PINN), Grey Wolf Optimization, Thermal energy storage.*

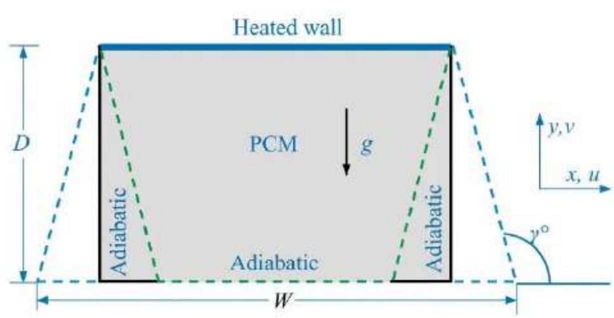


Figure 1: Image of different cases with PCM P-58 at Q =800 W/m^2.

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

COALESCENCE AND NON-COALESCENCE OF DROPLET INSIDE A BIFURCATED MICROCHANNEL UNDER PRESSURE DRIVEN FLOW

Deepanjan Das1,2, Ranjit Barua1 and Nirmalendu Biswas2*
1Department of Mechanical Engineering, OmDayal Group of Institutions, Uluberia 711316, India
2Department of Power Engineering, Jadavpur University, Kolkata 700106, India
*corresponding author email id: *biswas.nirmalendu@gmail.com*

ABSTRACT

Droplet coalescence inside a bifurcated microchannel is an essential tool in lab-on-a-chip applications like digital microreactors and biochemical synthesis. The hydrodynamic of the droplet merging inside the channel under Poiseuille flow are examined in this work. with a focus on the impact of capillary number (Ca), confinement ratio, and initial droplet offset on the coalescence dynamics. To represent interfacial deformation, film drainage, and ultimate coalescence, a 2-D model based on the phase-field formulation of the Cahn–Hilliard and Navier–Stokes equations is created. The combined effect of Ca, viscosity ratio and wettability on droplet merging for low confinement ratio is not studied earlier whish is the main novelty of this paper. The simulations are verified using known experimental data. The findings show that the capillary pressure causing film rupture and the viscous resistance in the thin continuous-phase film separating the nearby droplets which essentially controls the merging process. Because of effective film drainage facilitated by curvature-induced pressure gradients, the droplets approach quasi-symmetrically at low Ca (Ca = 0.01) and quickly combine after nearing the confluence region. Strong viscosity forces and droplet elongation slow down film drainage when Ca rises which prevent the coalescence. The regimes of complete coalescence, near-contact without merging, and hydrodynamic rupture are distinguished by a crucial offset distance between approaching droplets. In extremely confined conditions ($\chi > 0.7$), the confinement ratio ($\chi = D/H$) is observed to enhance interfacial deformation and alter the local pressure field at the junction, hence lowering the critical Ca for coalescence. Again, asymmetric flow partitioning at the Y-junction creates velocity gradients and shear stresses that modify droplet trajectory and collision orientation, as further shown by parametric analysis. When Ca is 0.01 the droplet merged but for higher Ca they are moving separately. So, Ca value is crucial for merging efficiency. By connecting interfacial dynamics with geometric confinement and hydrodynamic pressure, this work offers a thorough physical foundation for comprehending droplet merging in bifurcated microchannels. The results provide design guidance for microfluidic devices such microreactors, biological tests, and emulsion-based material synthesis that require regulated droplet fusion. In order to gain active control over merging events, this work can be extended for future developments of thermocapillary or electrohydrodynamic effects.

Key words: *Coalescence, droplet, microfluidics, phasefield, pressure driven flow.*

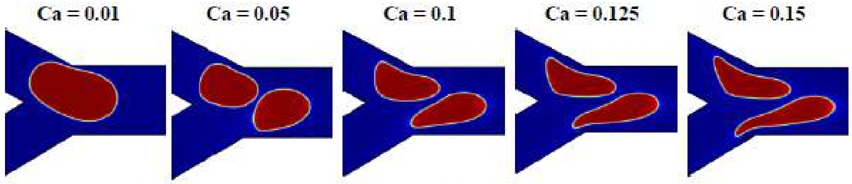


Fig. 1. From volume fraction droplet hydrodynamics for different surface tension at t = 0.5 s, when viscosity ratio = 1, density ratio = 1 and wettability angle = 150° and Re = 0.01.

NUMERICAL SIMULATION OF INCOMPRESSIBLE TWO-PHASE MAGNETOHYDRODYNAMIC FLOWS

Shyam Sunder Yadav

1Department of Mechanical Engineering,

Birla Institute of Technology and Science Pilani, Pilani campus

ss.yadav@pilani.bits-pilani.ac.in

ABSTRACT

In the current work we use the open source flow solver Basilisk to simulate incompressible two-phase magnetohydrodynamic (MHD) flows. The ultimate aim of this work is to simulate the heat transfer from plasma to liquid metals inside Tokamak used in nuclear fusion based reactors. By default, this simulation capability is not available in Basilisk, we have implemented it in the solver. The numerical formulation for the MHD is based on the electric potential which is valid for low magnetic Reynolds numbers. We also implemented the induced magnetic field based MHD formulation which remains valid for arbitrary magnetic Reynolds numbers. The interface between the two phases is tracked with the Volume-of-fluid method which is a mass conservative method. We compare the bubble/drop behavior under the conservative and non-conservative formulations of MHD body forces. We also investigate the effect of Hartman number, Reynolds number on the motion of drops and bubbles under uniform magnetic fields of different strengths. The developed methodology is general purpose in nature and it can be used to simulate the behavior of casting and welding under externally applied magnetic fields. The figure below shows the distribution of current density vectors during MHD duct flow.

Key Words: *Magnetohydrodynamics, Two-phase flows, Basilisk flow solver, Volume-of-fluid method, Low magnetic Reynolds number*

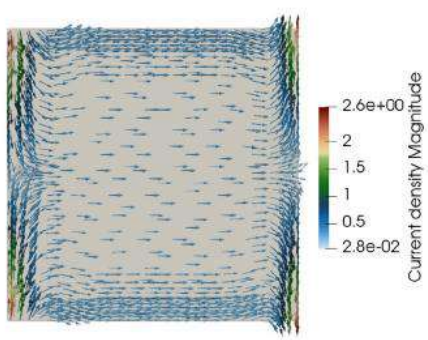


Figure 1: Distribution of current density vectors during MHD duct flow.

NUMERICAL SIMULATION OF THREE-PHASE FLOWS WITH PHASE CHANGE

Deepak Talan1, Shyam Sunder Yadav1

1Department of Mechanical Engineering,

Birla Institute of Technology and Science Pilani, Pilani campus

p20210450@pilani.bits-pilani.ac.in, ss.yadav@pilani.bits-pilani.ac.in

ABSTRACT

In the current work, we present our recently developed numerical technique for simulating three-phase flows with phase change. Specifically, we developed a numerical technique for simulating solidification and melting of a phase change material in presence of a surrounding gaseous medium. The novelty of the technique is that we are tracking the gas-liquid, liquid-solid, gas-solid interfaces explicitly where we include the effect of the respective surface tensions as well. The surface tensions at the three phase contact line govern the shape of the final solidified drop or solidification pattern around gas bubbles. This has been done in very few works previously. This technique has important consequences for the simulation additive manufacturing processes. The technique is based on the open source flow solver 'Basilisk'. We basically solve the equations governing mass, momentum and energy conservation for the three phases. The Navier-Stokes equations are solved with second order accuracy in space as well as time. We capture the interface between the three phases with the help of Volume-of-fluid method. Three different temperature fields are solved corresponding to the three phases. The temperature gradients at the liquid-solid interface are calculated and are used to calculate the mass flux due to phase change. This mass flux then gives the phase change velocity at the liquid-solid interface which is used to move this interface in the normal direction. We present various test cases demonstrating the efficacy of this technique. Our ultimate goal is to use this technique for simulating additive manufacturing processes. The figure below shows the temperature snapshot during the solidification of a drop placed on a cold boundary at bottom.

Key Words: *Three-phase flows, Solidification and Melting, Volume-of-fluid method, Basilisk*

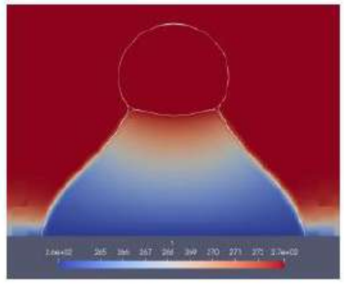


Figure 1: Drop shape and temperature distribution during solidification of the drop.

DIRECT NUMERICAL SIMULATION (DNS) OF EMULSION BEHAVIOR UNDER ELECTRIC FIELD

Akansha Singh[1], Shyam Sunder Yadav[1], Suvanjan Bhattacharyya[1]

[1]Department of Mechanical Engineering, Birla Institute of Technology and Science Pilani, Pilani campus
p20240466@pilani.bits-pilani.ac.in , ss.yadav@pilani.bits-pilani.ac.in, su vanjan.bhattacharyya @pilani.bits-pilani.ac.in

ABSTRACT

Effective separation of stabilised emulsions dispersion droplets in immiscible liquids is crucial across various industries, from pharmaceutical to oil and gas. Still, it continues to be difficult due to strong interfacial films. Present-day computational research lacks thorough Direct Numerical Simulation (DNS) studies of multi-droplet systems under electric-field conditions. This work aims to formulate and validate a DNS framework integrating incompressible Navier-Stokes equations, Maxwell stress tensor and Volume-of-Fluid interface tracking for simulating the droplet dynamics under electric field. We use the open-source flow solver Basilisk for this purpose. The Volume-of-fluid method is used for tracking the interface between the two phases involved. Validation using the established numerical standards guaranteed the computational model's robustness and accuracy. We observe faster emulsion separation under non-uniform electric field. With the application in the oil and gas sector, this model can enhance the efficiency of crude oil-water separation, minimise chemical demulsifier consumption, reduce pipeline corrosion, and optimise operating costs. The figure below shows the typical emulsion configuration studied by us. The current work focuses on the behaviour of multiple droplets under electric field whereas only one or two droplets have been considered in previous studies. Further results will be discussed in the complete paper.

Key Words: *Direct Numerical simulation, Electrocoalescence, Volume-of-Fluid method, Crude oil*

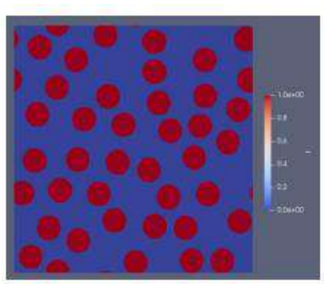


Figure 1: Typical distribution of dispersed and the continuous phases considered in the proposed numerical study.

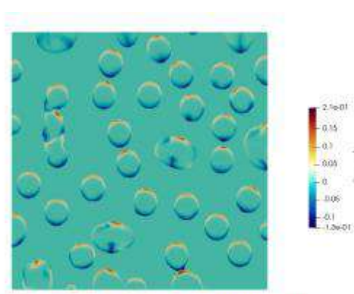


Figure 2: Typical charge distribution inside the dispersed droplets under uniform electric field.

ENSEMBLE AND PHYSICS-INFORMED LEARNING MODEL FOR NEXT-GENERATION PASSIVE MICROMIXER ENGINEERING

Jotiraditya Banerjee[1], Anjan Nandi[1], Dipankar Paul[1], Nirmalendu Biswas[1] *, Dipak Kumar Mandal[2] and Nirmal K. Manna[3]

[1] Department of Power Engineering, Jadavpur University, Kolkata 700106, India

[2]Department of Mechanical Eng., Government Eng. College Samastipur, Bihar, India

[3]Department of Mechanical Engineering, Jadavpur University, Kolkata 700032, India

[*]corresponding author email id: *biswas.nirmalendu@gmail.com*

ABSTRACT

In contemporary microfluidic systems, passive micromixers are crucial, especially in chemical analysis, biomedical diagnostics, and lab-on-a-chip devices where effective microscale mixing is crucial. When compared to traditional T-channel micromixers, pine-tree-inspired architectures have shown impressive advances in hydrodynamic mixing, delivering up to an 82.3% reduction in mixing energy or cost. Despite these developments, the high computing cost of CFD-based simulations still limits the speed at which geometric modifications and flow conditions may be explored in design optimization. In order to get over this bottleneck, the current study presents machine-learning-driven prediction frameworks that can precisely estimate mixing behavior while maintaining compliance with fundamental fluid mechanics principles. This allows for the design of micromixers to be completed more quickly and intelligently.

Five conventional machine learning regression frameworks are compared in this study. Support Vector Regression (R^2 = 0.8503, MAE = 0.078308, RMSE = 0.081754), Random Forest (R^2 = 0.9994, MAE = 0.002658, RMSE = 0.005219), XGBoost (R^2 = 0.9999, MAE = 0.001632, RMSE = 0.002331), K-Nearest Neighbors (R^2 = 0.9640, MAE = 0.016015, RMSE = 0.040088), and Gaussian Process Regression (R^2 = 0.9997, MAE = 0.003818). The study makes use of verified finite element simulation data that includes systematic changes in Reynolds numbers (10–100), Schmidt numbers (25–100), and geometric parameters (base length 0.8L–3.2L, height 0.8L–2.3L). Three crucial parameters are anticipated: mixing cost (MC), pressure drop (ΔP), and mixing index (η). The PiNN architecture clearly maintains the mechanical underpinning controlled by species convection-diffusion and Navier-Stokes equations. PiNNs incorporate partial differential equations as soft constraints, guaranteeing physically consistent predictions in contrast to solely data-driven methods. Comparative analyses show that Support Vector Regression has comparatively poorer generalization while ensemble-based models (XGBoost, Random Forest, and Gaussian Process Regression) attain almost perfect prediction accuracy with little error. This study provides practical recommendations for choosing the best machine learning techniques for microfluidic optimization by quantifying the trade-offs between accuracy, computational complexity, and physical consistency (Fig. 1). The results open the door to expedited and physically grounded microfluidic design frameworks by advancing both passive micromixer technology and physics-informed machine learning methods.

Key Words: *Passive micromixers, Mixing performance, Mixing cost, Physics-informed neural network (PINN).*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

EFFECT OF GRAVITY ON NATURAL CONVECTION OF SUPERCRITICAL CO2 IN A CYLINDRICAL ENCLOSURE WITH AN EMBEDDED HEATER

Niladri Sarkar1, Md Shams Nayeem1, Aranyak Chakravarty2 and Koushik Ghosh1

[1]Department of Mechanical Engineering, Jadavpur University, Kolkata 700032, niladri41099@gmail.com; mdshamsnayeem@gmail.com; koushik.ghosh@jadavpuruniversity.in

[2]School of Nuclear Studies & Application, Jadavpur University, Kolkata 700106, aranyak.chakravarty@jadavpuruniversity.in

ABSTRACT

Natural convection of supercritical carbon dioxide (sCO_2) in confined geometries is a key heat-transfer mechanism in advanced energy, thermal storage, and high-pressure processing systems. The novelty of this study lies in adoption of this physical configuration and is especially relevant due to its application for propellant storage in space environment. This study presents a computational investigation of natural convection of sCO_2 in a closed cylindrical enclosure containing an embedded internal heater (see Fig. 1a), with and without consideration of gravity. The analysis is carried out using ANSYS Fluent and focuses on the complex thermo-fluid behaviour that arises from the strong, temperature-dependent variations in thermo-physical properties, especially near the pseudo-critical region of sCO_2. The thermo-physical properties are modelled using dedicated correlations obtained in-house by non-linear regression of NIST property data, and implemented utilising user-defined functions. Parametric studies are conducted for the above configuration considering a range of heater power – with and without consideration of gravity. The flow and heat transfer characteristics of sCO2 in the cylindrical enclosure is shown for a representative case (27W heater power)– with and without gravity – in Fig. 1(b-d).

Key Words: *Supercritical CO2, Natural Convection, Pseudo-phase transition, Non-linear Regression.*

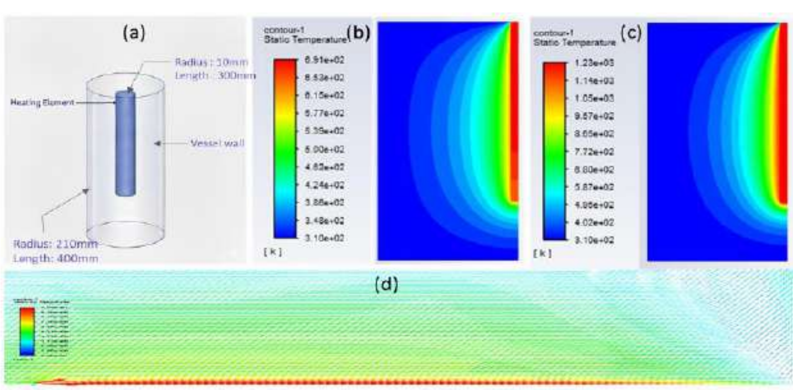


Figure 1: (a) Schematic representation of the geometry, (b) Temperature distribution with gravity, (c) Temperature distribution without gravity and (d) velocity vectors in the vicinity of the heater for a representative case with heater power (27 W)

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

COMPARATIVE ANALYSIS OF BIOLOGICAL TISSUE HEAT TRANSFER DYNAMICS IN MAGNETIC FLUID HYPERTHERMIA

Pratik Roy[1,2], Ranjan Ganguly[1] and Nirmalendu Biswas[1,*]
[1]Department of Power Engineering, Jadavpur University, Saltlake, Kolkata, 700106, India
[2]Department of Electrical Eng., Medinipur Sadar Govt. Polytechnic, Midnapore, 721102, India
*Corresponding author email: *biswas.nirmalendu@gmail.com*

ABSTRACT

Conventional therapeutic approaches for malignancies have often been limited by recurrent challenges, particularly in advanced-stage cancers or after extended treatment exposure. Magnetic Fluid Hyperthermia (MFH) is a novel non-invasive therapeutic approach that has emerged as a promising platform for therapeutic intervention in tumour regression. Magnetic fluid hyperthermia (MFH) induces tumour regression or necrosis by exploiting heat generated from relaxation losses (Néelian and Brownian) of pre-deposited superparamagnetic nanoparticles (SPMNPs) within the tumour tissue when exposed to a high-frequency (~kHz) alternating magnetic field (AMF). *In silico* thermal analysis of MFH utilizes Rosensweig's linear response theory (RLRT) and Pennes Bioheat (PBH) models for temperature evaluation in tissue-tumour models and Arrhenius Thermal damage (ATD) models for necrosis probability assessment. Blood perfusion is reported to have a vital significance in thermal transport in MFH, which varies both spatially and with treatment temperature in the tumoural region. However, most numerical studies investigating thermal transport in MFH are based on simplified assumptions of isotropic perfusion rates across the tumour anatomy, leading to discrepancies between findings *in silico* and those of comparable *in vitro* and *in vivo* investigations, thereby reducing confidence in the procedure's overall reliability and acceptance. The present numerical study investigates thermal transport phenomena and necrosis probability, incorporating heterogeneous perfusion models varying spatially and temporally. In this study, spatial variation is modelled using Radial Gaussian (RGD) and Complementary Radial Gaussian (CRGD) distribution functions, which are found to have close resemblance with blood perfusion patterns as reported in data from Computer Tomography (CT) scans. Temporal variation of tissue temperature is modelled using temperature (TPM) and tissue damage (DPM) dependent perfusion models used in different numerical studies available in the literature. Spatial temperature and tissue damage profiles are also analyzed to ascertain the clinical efficacy. TPM model showed 25 % less heating non-uniformity, with 2°C and 1°C less attained temperature at centre and edge respectively, compared to the DPM models (Fig. 1).

Key Words: *Magnetic fluid hyperthermia, Biological Heat Transfer, Heterogeneous blood perfusion, Finite Elements, Breast cancer.*

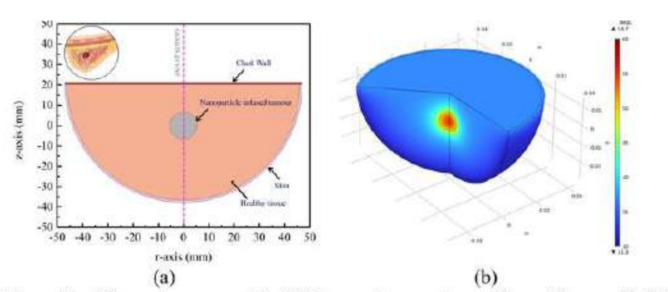


Figure 1. (a) Schematic of tissue-tumour model, (b) Temperature contour of breast tissue with RGD type spatial perfusion distribution.

HEAT TRANSFER PERFORMANCE OF A TUBE HEAT EXCHANGER WITH INLINE SEMI-CIRCULAR WINGLETS IN STRIP INSERTS

Bipin Kumar[1], Anil Kumar Patil[1*], Manoj Kumar[2]

[1]Department of Mechanical Engineering, DIT University, Dehradun 248009, India

[2] Department of Mechanical Engineering, Graphic Era Deemed University, Dehradun 248002, India

Corresponding author email: akpt1711978@gmail.com, dr.anilpatil@dituniversity.edu.in

ABSTRACT

The present experimental study examines the influence of strip inserts equipped with novel inline semi-circular winglets (SIISCW) on the thermo-hydraulic performance of a tubular heat exchanger. Experiments were conducted using strip inserts with semi-circular winglets at a fixed pitch length of 30 mm with each inline winglet oriented at angles of attack of 30°, 45°, and 60° over a Reynolds number range of 10,000–22,000. The thermal and frictional characteristics obtained for the inline winglet inserts were benchmarked against those of a plain tube and a solid strip insert under identical operating conditions. SIISCW (y: 1, α: 60°) is found to be the most efficient insert configuration.

Key Words: *Strip insert, Semi-circular Winglet, Angle of attack, Thermal performance factor.*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

MAPPING THE RESEARCH LANDSCAPE OF SUBCOOLING TECHNOLOGIES IN SUSTAINABLE REFRIGERATION SYSTEMS

Prosenjit Singha[a], Sonam Gopaldasji Rajpuriya[a*], and Mani Sankar Dasgupta[a]

prosenjit.singha@pilani.bits-pilani.ac.in

dasgupta@pilani.bits-pilani.ac.in

[a]Department of Mechanical Engineering, BITS Pilani, Pilani, 333031, India

[*]Corresponding author: sonam.rajpuriya@pilani.bits-pilani.ac.in

Abstract: As HVAC systems shift toward natural refrigerants for sustainability, subcooling has emerged as a key strategy to improve efficiency by lowering refrigerant temperatures and increasing cooling capacity. This study presents a comprehensive review of subcooling technologies, focusing on global research trends from 2015 to 2024. Through systematic literature analysis and bibliometric mapping, the study identifies major research themes, technology trajectories, and gaps. To contextualise these insights, a case study is included that evaluates the thermodynamic, exergy, environmental, and economic performance of R134a, R290, and R1234yf refrigeration systems for a 133-kW milk processing plant. R1234yf, with low latent heat and vapour density, requires the highest refrigerant mass flow, compressor displacement, and work. R290 consistently delivers superior performance. Rising condensing temperatures increase compressor work and heat recovery potential but reduce COP, cycle COP (CCOP), and exergy efficiency. Among the three, R290 offers the highest CCOP and heat recovery; R134a shows slightly better COP, and R1234yf performs the poorest. A 15-year life cycle assessment shows R1234yf has the highest CO_2 and TFA emissions, followed by R134a. R290 leads with the lowest emissions and life cycle cost, confirming its viability as a sustainable and energy-efficient alternative to conventional HFC and HFO refrigerants.

Key words: *Bibliometric analysis; Natural refrigerants; Subcooling; Energy efficiency; Sustainable cooling*.

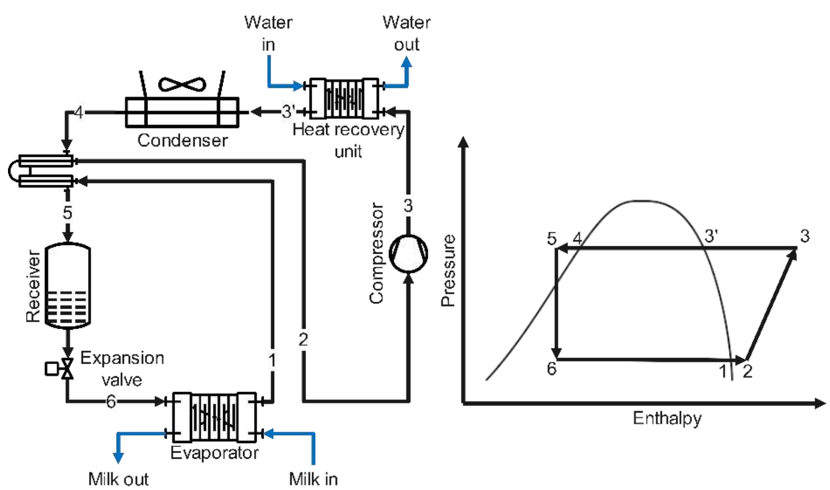


Figure: schematic of (a) VCR system and (b) corresponding p-h diagram

TWO-PHASE HEAT TRANSFER ENHANCEMENT OF OPEN MICROCHANNEL HEAT SINKS

Md Danish Eqbal, Akash Priy, Manabendra Pathak# and Mohd. Kaleem Khan

Sustainable Energy Research Laboratory, Department of Mechanical Engineering, Indian Institute of Technology Patna, Bihar-801106, India

#Corresponding Author, E-mail address- mpathak@iitp.ac.in

ABSTRACT

During flow boiling in conventional microchannel heat sinks, the channel confinement adversely affects the expanding vapour slug, leading to flow boiling instabilities and poor heat transfer performance. The confinement problem of conventional microchannels can be effectively addressed with an open microchannel configuration, where a gap is maintained between the fin's top surface and the cover plate. This work presents a systematic analysis of the thermohydraulic performance of flow boiling in open microchannels and compares it with that of conventional closed microchannels for identical geometric and flow conditions. The experimental findings reveal the distinct flow behavior and heat transfer mechanisms in open channels, providing new inputs for designing next-generation cooling architectures. An open microchannel heat sink has been fabricated with a 0.3 mm gap between the fin and the cover plate. Each microchannel configuration comprised an array of eleven rectangular channels with a hydraulic diameter of 500 µm. The flow boiling experiments were conducted with deionized water at a constant subcooling of 20°C, with varying heat fluxes and coolant mass fluxes of 256 and 536 kg/m²s. A flow visualization study has been conducted using a high-speed camera to examine two-phase flow patterns in open and closed microchannel heat sinks, as well as their impact on heat transfer characteristics and flow boiling instabilities. Experimental observations reveal that the open microchannel heat sink exhibits superior thermal performance with a reduction in wall temperature up to 3.3 °C and a maximum heat transfer enhancement of 20% compared to the closed configuration. Additionally, in the proposed open configuration, the standard deviation of wall-temperature fluctuations reduced from 3.24 °C to 1.32 °C, whereas the standard deviation of pressure-drop fluctuations reduced from 0.88 kPa to 0.14 kPa. The lower amplitudes of wall temperature and pressure drop oscillations observed during slug flow further demonstrate that the open configuration reduces boiling instabilities which is desired for two-phase flow microchannel heat sinks. Additionally, a shorter bubble ebullition cycle time of 0.949 seconds has been observed in open microchannels, compared to 1.19 seconds in closed microchannels, indicating improved flow boiling stability and enhanced vapor removal. Furthermore, compared to the closed microchannel, the average pressure drop was significantly reduced by 60–90% in the open microchannel configuration. The improved thermo-hydraulic performance of the open microchannel could be attributed to the presence of an unrestricted vapor escape path, enhanced liquid–vapor phase stratification, suppression of reverse flow, and augmented heat transfer through the fin's top spacing.

Key Words: *Open microchannels, boiling instabilities, flow boiling*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

NUMERICAL AND DATA DRIVEN INVESTIGATION OF FLUID FLOW PAST A BLUFF BODY.

S.V.H.Nagendra[1], DVS Bhagavanulu[2]

[1]Department of Mechanical Engineering, VGU Jaipur, India,svh,nagendra@vgu.ac.in

[2]Department of Civil Engineering, VGU Jaipur, India, dvs.bhagavnulu@vgu.ac.in

ABSTRACT

This study investigates fluid flow past a bluff body(Cylinder) by comparing simulations from a traditional computational fluid dynamics (CFD) tool with predictions from Machine Learning models. The investigation evaluates the efficacy of models like Random Forest (RF) and Convolutional Neural Network (CNN) as computationally efficient alternatives for predicting total pressure and velocity magnitude. A dataset was generated using commercial CFD tool, employing the k-ε turbulence model, for inlet velocities ranging from 5 m/s to 105 m/s. The trained RF and CNN models were subsequently validated against dedicated simulations for five unseen inlet velocities. The results indicate that the CNN generally excels at predicting the overall velocity fields and pressure profiles. However, the Random Forest model is substantially more accurate in capturing the critical stagnation-point velocity, a region in which the CNN consistently overestimates the value. Predicting the pressure field is challenging for both Machine Learinng models, particularly at higher velocities. These findings suggest considerable potential of machine learning models to predict and also accelrates dynamic fluid-flow simulations, selecting an optimal model depends on the point of interest that is specific flow parameter, it also high lights challenges in achieving uniform accuracy over all parameters.

Key words: *CFD, Random Forest , CNN, Velocity profile , Pressure profile.*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

DEVELOPMENT OF REDUCED CHEMICAL KINETIC MECHANISM FOR COMBUSTION MODELLING

Bijoy Das[1], Mithun Das[2], Amitava Datta [3]

[1]Department of Power Engineering, Jadavpur University, Kolkata 700106, India bijoy.kgec@gmail.com

[2]School of Nuclear Studies and Application, Jadavpur University, Kolkata 700106, India mdas190@gmail.com

[3]Department of Power Engineering, Jadavpur University, Kolkata 700106, India amdatta_ju@yahoo.com

ABSTRACT

The combustion characteristics in a coaxial non-premixed flame are strongly influenced by the chemical kinetic mechanism employed. Global reactions available in Ansys Fluent are widely adopted in combustion simulations, as the detailed kinetic mechanism requires intensive computational demand and extended analysis time. In this study, a reduced kinetic mechanism is developed using the sensitivity analysis framework in ANSYS Chemkin Pro. Reaction pathways and species with dominant contributions to ignition behavior, radical production, and flame stabilization were quantitatively ranked across relevant thermochemical conditions. Low-impact reactions were selectively removed, yielding a compact mechanism with 52 reactions that preserves the essential methane oxidation chemistry. The developed reduced mechanism (84% reduction) was validated against the GRI-Mech 3.0 detailed mechanism as well as simulation result from other researchers employing reduced mechanism reported in the literature, through comparisons of flame temperature, and major species distributions (CO, CO_2, H_2O) under coaxial non-premixed operation. The results demonstrate that the reduced mechanism maintains high fidelity (peak flame temperature <1% variation and species distribution within 3-5%) while substantially lowering computational cost, enabling more efficient and scalable simulations of practical methane combustion. This approach provides a robust pathway for mechanism simplification without compromising physical accuracy in complex non-premixed flame environments.

Key Words: *Sensitivity Analysis, Combustion, Reduced Mechanism, Numerical simulation*

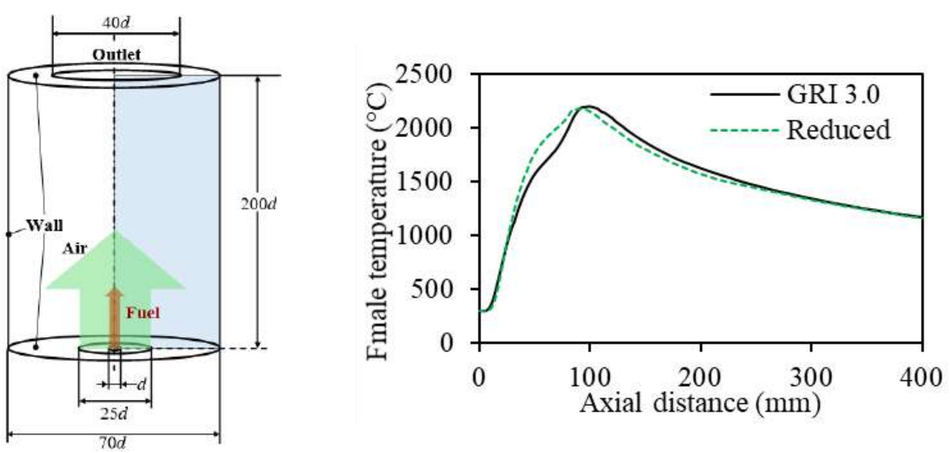


Figure 1: Schematic of coaxial non-premixed flame and CFD result comparison between GRI 3.0 mechanism and reduced mechanism

A NUMERICAL STUDY ON THE EFFECTS OF INTERMEDIATE HYDROPHOBIC BANDS ON THE MIXING PHENOMENA IN A SQUARE WAVE MICROCHANNEL AT LOW REYNOLDS NUMBER FLOW

Rajib Majumder[1], Arindam Bit[1]
[1]Department of Biomedical Engineering, NIT Raipur, Chhhattisgarh, India-492010, E-mail addresses
*Corresponding author: arinbit.bme@nitrr.ac.in

ABSTRACT

The phenomena of mixing has huge applications in biomedical and other engineering fields. An attempt has been made to understand the effects of intermediate hydrophobic bands on mixing for a square wave microchannel. Numerical simulation has been conducted for three configurations: without hydrophobic band (WB), and two configurations with band (B1 and B2). Comsol 5.6a has been used for numerical simulations for Reynolds number (Re) ranging from 0.267 to 50. Water and ethanol were selected as mixing fluids in simulation. The results revealed that there has been very slight increase in the mixing index at the channel outlet with B1 and B2 configurations for lower Re values. This effect is less prominent as Re is increased. However, the mixing at the outlet is almost complete at Re no 15 and above for all the configurations. However, at lower Re values, the effect of the configurations is more prominent having slightly higher values of mixing index for B1 and B2 configurations.

Key Words: *Mixing index, Square wave microchannel, Hydrophobic band, Reynolds number*

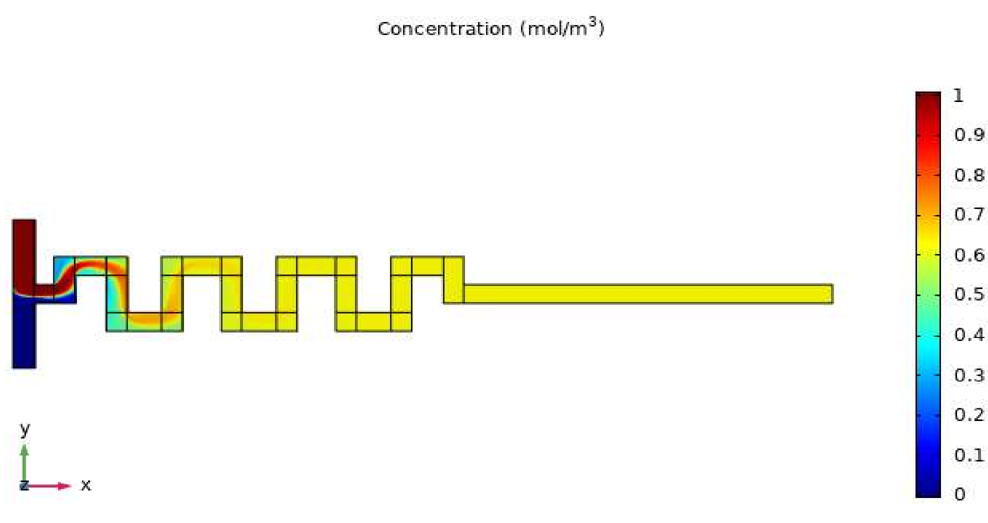


Figure 1: Concentration distribution for the microchannel at Re=50 for WB flow

INVESTIGATION OF THERMAL ENERGY STORAGE MATERIALS FOR SOLAR COOKING

Deepika Kumari1, Sandeep Kumar1 and Sunita Mahavar[1,2*]

[1]Solar Energy Research Laboratory (SERL), Department of Physics, University of Rajasthan, Jaipur 302004, India, smjpr1986@gmail.com

[2]Material Science, Innovation and Modelling Research (MaSIM) Research Focus Area, Department of Physics, North-West University (Mafikeng Campus), Private Bag X2046, Mmabatho 2735, South Africa

ABSTRACT

Energy storage is a crucial technique in the 21st century, particularly due to the depletion of non renewable resources worldwide. The intermittent nature of renewable energy sources highlights the need for effective storage solutions to ensure a reliable and uninterrupted energy supply. Sunlight can be harnessed using solar thermal collectors for various thermal applications, including cooking, water & space heating, desalination, and drying. The use of efficient thermal energy storage material (TESM) can enhance the performance of a solar collector. In this paper, a method is developed to evaluate the performance of different TESMs under identical conditions using an electric backup solar cooker (EBSC). Novel parameters were introduced to evaluate the performance of EBSC under preheating, sensible heat and discharging test conditions with the selected TESMs. Two types of containers were utilized: a usual cooking container (UCC) and a specially designed storage cooking container (SCC). Two sensible materials, glycerine (GY) and sunflower oil (SFO), and four phase change materials (PCMs), namely paraffin wax (PW), stearic acid (SA), palmitic acid (PA), and lauric acid (LA), were selected based on their physical properties. In all observations, the temperature of SCC increased rapidly compared to UCC, which clearly reflects the performance improvement achieved using TESMs. The preheating time of 52 minutes was the lowest for GY compared to other TESMs. The total time taken to reach water at 90 °C in SCC with GY, SFO, PW, SA, PA and LA was 86, 92, 108, 104, 108, and 98 minutes, respectively. In the sensible heating test, the thermal efficiency of GY was found to be the highest, 98%. The performance of PA was also notable, with 85.2%, which was higher than that of others. The discharging rate of GY was found to be 0.241 °C/min, whereas for the others, it varied from 0.275 to 0.342 °C/min. This study is clear evidence of glycerine's superior thermal behaviour for use in solar cooking. The methodology adopted for the testing can be further extended to test different combinations of these selected materials.

Key words: *Thermal energy storage materials, electric backup solar cooker, storage cooking container, glycerine, phase change materials.*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

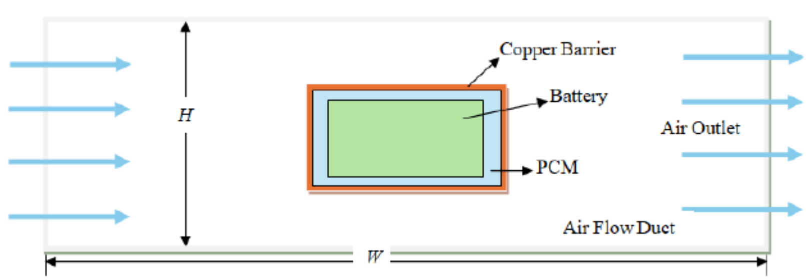
ENHANCED THERMAL REGULATION OF HIGH-RATE PRISMATIC BATTERIES UTILISING HYBRID COOLING TECHNIQUE

Rathin Ghosh[1], Dipankar Paul[1], Anjan Nandi[1], Nirmalendu Biswas[1,*] and and Suvanjan Bhattacharyy[2]
[1]Department of Power Engineering, Jadavpur University, Kolkata 700106, India
[2]Department of Mechanical Engineering, Birla Institute of Technology and Science Pilani, Pilani Campus, Vidya Vihar, Pilani, Rajasthan, India.
**email:* biswas.nirmalendu@gmail.com

ABSTRACT

This research numerically investigated the heat transfer characteristics of a prismatic Li-ion battery operating in a large square cavity when air was blowing past it. The battery module (130.3 × 36.3 × 200.5 mm) was set up in the centre of a large duct cavity, with the fan-induced airflow blowing toward its smallest face, 36 mm wide. The width of this cavity is twice the width of the battery, and its length is four times as long so that crossflow reaches every part thereof. Heat from the battery is dissipated by means of a hybrid passive-active design: the RT-45 phase change material (PCM) encloses it on all but its base, a 2-mm sheet of copper wards off heat and evens internal temperatures. The heat generated by discharging the battery simulates a volume source with efficiency of about 28300W/m^3 (or ~ 26.9W total) modelled at 3C current. Air comes in from the left at 1 to 1.1 m/s and its temperature remains more or less uniform. Comparison of the operating temperatures under air-only cooling and PCM plus copper-assisted cooling: Considering air-cooled operation, flow is essentially laminar and its heat transfer coefficient low. This leads to rapid temperature rise: the battery temperature rises from 50°C to 64–67°C within 1800 s, making safety standards out of reach in extremis. The paraffin RT-45 PCM on the other hand consumes a significant amount of the generated heat, via latent heat melting. This limits peak temperature to something between 42–45°C and is a reduction of 31–35% compared with air-cooling alone. The copper layer further improves temperature uniformity, reducing local hotspots and lowering maximum temperature by a further 8–12%. Under forced airflow, approximately 78–82% of PCM volume enters the melting phase; by comparison, with natural convection only about 45–55% can be expected to melt. This guarantees prolonged thermal stability. With the addition of a copper sheet–encased PCM, the maximum temperature gradient across battery surface is reduced from 12.4 to 3.1°C, contributing directly to more stable electrochemical behavior and higher discharge efficiencies (4–6%), hence a potential life extension of 20–28% for the battery. In conclusion, the integration of RT-45 PCM, copper wrapping, and laminar airflow turns out to be a highly integrated and effective method to manage high-rate prismatic battery systems.

Key Words: *Battery thermal management; Phase change material; RT-45 paraffin; Copper heat spreader; Laminar forced convection; Prismatic battery cooling*



BITS Pilani
Pilani Campus

SPARC
Scheme for Promotion of Academic and Research Collaboration

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

PERFORMANCE ANALYSIS OF ELECTROOSMOTIC MICROMIXER: EFFECT OF MIXING CHAMBER ORIENTATION

Sk Mintajuddin Ahamed[1], Amitava Dutta[1,*], Nirmalendu Biswas[2]

[1] Department of Mechanical Engineering, Aliah University, Kolkata 700160, West Bengal, India
[2] Department of Power Engineering, Jadavpur University, Kolkata 700106, West Bengal, India
**Corresponding author*: amitava_math@rocketmail.com

ABSTRACT

Efficient mixing in microfluidic systems is challenging due to the dominance of laminar flow and slow molecular diffusion at low Reynolds numbers. This work presents a numerically analized electroosmotic micromixer featuring a converging-diverging microchannel with a diamond-shaped mixing chamber driven by *AC*-induced electroosmotic flow (AC-EOF) generated through two pairs of diagonally arranged microelectrodes embedded within microgrooves on the chamber wall. Although electroosmotic micromixers have been widely investigated, the combined use of a diamond-shaped chamber geometry and diagonally actuated microgroove electrodes under AC excitation has not been previously explored, establishing the central novelty of this study. A detailed parametric analysis is conducted to quantify the influence of (i) chamber inclination angle (α = 0°, 20°, 40°, 60°), (ii) average inlet velocity (U_0 = 0.05-0.5 mm/s), (iii) AC actuation frequency (f = 4-16 Hz), and (iv) voltage amplitude (V_0 = 0.05 V – 0.5 V) on mixing enhancement. The microchannel dimensions are maintained at L = 80 µm and W = 20 µm, with a 3 µm microgroove accommodating the electrodes. Two concentration of fluids with initial concentrations of 1 mol/m³ and 0 mol/m³ enter the device through separate inlets under creeping-flow conditions (Re < 1). The electric potential, flow field, and concentration distribution are predicted by solving Laplace's equation, the incompressible Navier-Stokes equations, and the convection-diffusion equation using a finite element solver. Grid independence is ensured prior to the simulations.The results show that the inclination angle of the mixing chamber significantly affects the strength and structure of electroosmotic vortices formed within the mixing chamber region. A chamber orientation of α = 0° facilitates symmetric vortex circulation, resulting in the highest observed mixing efficiency of 98.95% within 1 s at operating conditions U_0 = 0.05 mm/s, f = 8 Hz, and V_0 = 0.5 V. Increasing the inclination angle disrupts the symmetry of the vortical flow, reducing mixing efficiency. The insights gained from this numerical investigation can guide the development of compact, low-energy, high-efficiency micromixers for biochemical analysis, lab-on-chip devices, and other microfluidic applications requiring rapid and uniform mixing.

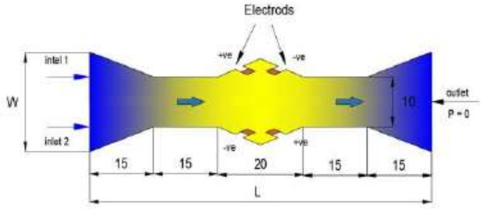


Fig. 1. Schematic diagram of model micromixer

CFD-BASED MULTIPHASE FLOW ANALYSIS OF A MODIFIED LIQUID DESICCANT REGENERATOR WITH CORRUGATED SURFACE AND BAFFLES

Mrinal Pradhan[1], Koushik Das[1] and Rajat Subhra Das[1]
NIT Meghalaya, Meghalaya, pradhanmrinal14@gmail.com, koushik.das@nitm.ac.in, rajatsubhra@nitm.ac.in

ABSTRACT

Liquid desiccant systems have emerged as a promising alternative for energy-efficient dehumidification and cooling applications. A critical component of such systems is the regenerator, where the desiccant solution is re-concentrated by removing absorbed moisture. In this study, CFD technique is employed to investigate the performance of a liquid desiccant regenerator with modified geometrical features. The modifications include the introduction of a corrugated surface and strategically placed baffles, aimed at enhancing fluid mixing and improving mass transfer characteristics. The CFD model incorporates multiphase flow with coupled heat and mass transfer, accounting for the variation of desiccant properties with concentration and temperature. Simulation results demonstrate enhanced moisture removal rates compared to the flat surface regenerator. It is evident from Fig. 1 that the change in the air humidity is gradual in both types of regenerators. However, the change is much rapid in the case of the modified regenerator. The average humidity at the air outlet is 21.45 g/kg and 23.84 g/kg for the plane and the modified regenerator respectively. The velocity contours (Fig. 2) presented for the modified regenerator highlight the influence of the corrugated geometry and baffles in promoting effective mixing and flow distribution. These effects promotes the enhanced diffusion of humidity and multiple localised convective mass transfer in the modified regenerator. These findings confirm that geometrical modifications play a significant role in improving the efficiency of liquid desiccant regeneration. The study contributes to the advancement of liquid desiccant technology by providing insights into the design of regenerators with improved performance. The proposed modifications demonstrate potential for improved regeneration performance and enhancing the viability of liquid desiccant systems in sustainable cooling applications.

Key Words: *Heat and mass transfer, CFD, liquid desiccant regenerator, multi-phase flow*

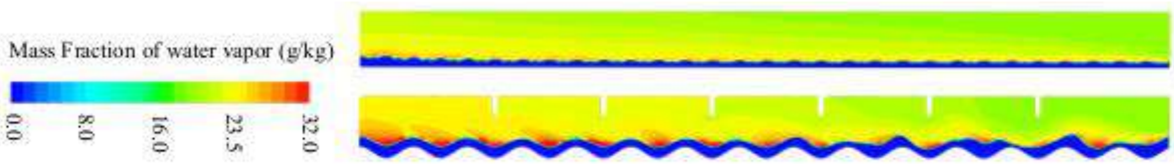


Figure 1: Water vapor distribution inside the flat and modified regenerator

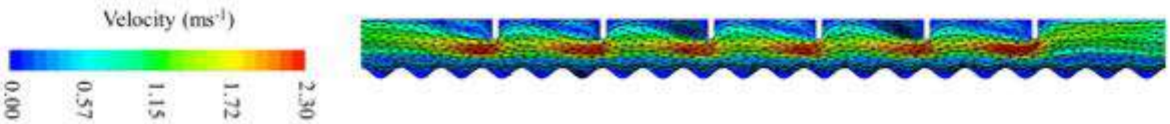


Figure 2: Velocity distribution for regenerator with corrugated surface and baffles

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

IMPACT SAFETY NON-NEWTONIAN FLUID GLOVE: A SHEAR COMPOSITE FOR INDUSTRIAL IMPACT PROTECTION

Yogesh[1], Vaibhav Dubey[1], Shivam Samadhiya[1], Shashank Agarwal[1], Vishal Chaudhary[2], and Dinesh Kumar[1]

[1]Department of Mechanical Engineering, Dayalbagh Educational Institute, Agra, E-mail; yogeshchahar321@gmail.com, vaibhavdubey9306@gmail.com, samadhiyashivam195195@gmail.com, shashankagarwal1024@gmail.com, dinesh.iitkgp19@gmail.com

[1]Department of Mechanical Engineering, National Institute of Technology, Meghalaya, E-mail; ps24.me.002@nitm.ac.in

ABSTRACT

This study presents a novel shear-thickening fluid (STF) integrated glove designed to improve hand safety in heavy-impact industrial machining by offering high flexibility during normal use and rapid stiffening upon impact. The STF, a silica–PEG/TEG composite with low viscosity at rest and a sharp increase under shear, allows the glove to absorb and dissipate impact forces effectively while maintaining ergonomic dexterity. This innovation addresses the limitations of traditional rigid gloves, which often reduce comfort and finger mobility, leading to inconsistent use.This adaptive, durable design addresses limitations of conventional gloves by improving worker comfort and compliance without sacrificing protection, offering a scalable solution for next-generation industrial personal protective equipment. This study introduces an STF-integrated glove for heavy-impact machining safety, utilizing a silica–PEG/TEG composite (density 1.1–1.3 g/cm^3) with low-shear viscosity of 5–50 Pa·s that sharply rises to 1000–10,000 Pa·s above 1–10 s^{-1} shear rates, enabling rapid transition from flexible fluid-like state to rigid protection during impacts. The STF, sonicated for uniform dispersion, is embedded into a 5–8 mm Kevlar composite core via flexible pouches or mesh impregnation, sandwiched between abrasion-resistant Kevlar/Dyneema outer layers and moisture-wicking inner liners to ensure comfort and durability. This lightweight design maintains >90% elastic recovery post-impact, preserving ergonomic flexibility for routine tasks like tool handling while resisting abrasion and repetitive stresses in industrial environments. Impact testing at 2–5 m/s, mimicking tool strikes and debris in machining operations, demonstrates 200–400 N/cm^2 force absorption and 5–15 J/cm^2 energy dissipation—yielding 25–40% superior attenuation over conventional rigid-insert gloves—without hindering finger mobility or increasing user fatigue. The adaptive STF response ensures sustained performance across prolonged cycles, offering a scalable, mass-producible solution that advances ergonomic protective gear for manufacturing floors demanding reliable, long-life hand safety. The key innovation of this study is the dual-mode integration approach, which allows the glove to respond dynamically to varying strain rates. Under normal movement, the glove remains soft and comfortable, while under sudden impact it becomes rigid and enhances protection. This behaviour enables better energy absorption and distribution compared to conventional gloves that rely on static rigid inserts. Quantitative performance results are often missing in low-c st industrial glove designs. The STF-integrated glove offers flexible, lightweight protection that stiffens instantly under heavy machining impacts, absorbing energy from tool strikes and debris while preserving dexterity. Its durable, adaptive composite design ensures long-term resistance to repetitive high-force cycles in industrial settings. This scalable solution advances ergonomic safety gear for manufacturing floors.

Key Words: *Shear Thickening Fluid, Impact Protection, Kevlar Composite, Industrial Safety, Energy Dissipation*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

DETERMINATION OF HEAT RELEASE RATE RESPONSE TO FLOW PERTURBATIONS IN AN ELECTRICALLY HEATED RIJKE TUBE USING TRANSFER FUNCTION APPROACH

Souranshu Roy Chaudhuri[1], Sourav Sarkar[1] and Achintya Mukhopadhyay[1]
[1]Mechanical Engineering Department, Jadavpur University, Kolkata – 700032, India E-mail addresses: souranshu.ju@gmail.com, souravsarkar.mech@jadavpuruniversity.in, achintya.mukhopadhyay@jadavpuruniversity.in

ABSTRACT

Thermoacoustics is pivotal in applications ranging from combustion stability in gas turbines and rocket engines to sustainable energy generation from low-grade heat sources like solar and waste heat. In these systems, acoustic waves are sustained by periodic heat release, with the coupling between pressure and heat release rate fluctuations being critical. Given the significant disparity in length and time scales—where acoustic waves operate on a much larger scale than the heat release processes—it is practical to simulate these phenomena separately. The acoustic model is typically solved over a large domain with a coarse mesh, while the heat release is resolved in a smaller, finely meshed region. One effective coupling strategy employs transfer functions, where the heat release response to inlet flow perturbations, derived from CFD or experiments, is integrated into the acoustic model. This research systematically investigates how different input forcing functions affect the derived transfer function, highlighting its novelty. The study focuses on a Rijke tube setup—a long, open ended duct with an electrically heated wire gauge. The flow domain includes a cylinder representing the heater. A sinusoidal perturbation is superimposed on a steady inlet flow, with both amplitude and frequency varied to capture nonlinear effects. The convective heat transfer response from the cylinder is analyzed using a transfer function approach. By performing FFTs on the input velocity and output heat transfer rate fluctuations at the forcing frequency, a complex transfer function is obtained. Its magnitude and phase reveal the gain and phase difference, respectively. The overall system shows a gain between approximately 0.451 and 0.462, alongside a phase angle range of -5.0 to -22.5 degrees, indicating consistent signal attenuation and a slight phase lag across the tested frequencies.

Key Words: *Thermo-acoustics, Transfer Function, Rijke Tube, CFD*

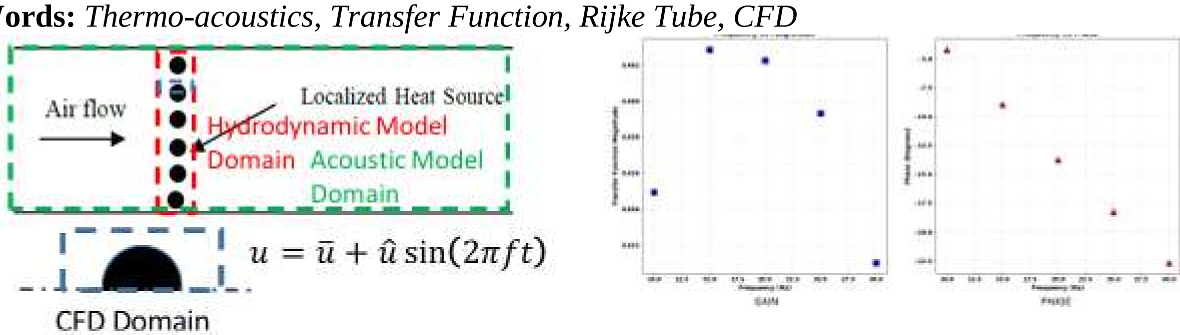


Figure 1: Computational Domain (Left), Transfer Function (Right)

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

THERMAL AND HYDRAULIC PERFORMANCE ASSESSMENT OF JET-PLATE SOLAR AIR HEATERS

Debabrata Roy[1], Rajat Subhra Das[2,*], Nirmalendu Biswas[3]

[1]Department of Mechanical Engineering, NIT Meghalaya, Meghalaya 793108, India, e-mail: *p24me003@nitm.ac.in*

[2]Department of Mechanical Engineering, NIT Meghalaya, Meghalaya 793108, India. e-mail: *rajatsubhra@nitm.ac.in*

[3]Department of Power Engineering, Jadavpur University, Kolkata 700106, India, e-mail: *biswas.nirmalendu@gmail.com*

ABSTRACT

Solar air heater is one of the promising techniques for harnessing the solar energy and to produce hot air. However, due to the lesser thermal performance, solar air heaters have restricted its application. Several researchers has attempedted to improve the the air heater performance by modifying surface roughness artificially. Roughness promoter like arc, ribs, grooves, dimples, baffles, winglets, corrugation etc. are attached with absorber plate to produce eddies and vortices in turbulent flow. These eddies and vortices break the viscous sublayer near the vicinity of heating surface and for this heat transfer augments. But now a days impinging jets are applied on modified surfaces of solar air heater to enhance its thermohydraulic performance. The application of impinging jet prevents the thick boundary layer formation adjacent to heating surface. It has been noticed from the present study that jet diameter and number of jets significantly affect the thermohydraulic performance in traditional solar air heater. In present study a 2-Dimensional CFD model is prepared in ANSYS software using RNG k-ε turbulence model with enhanced wall treatment has been applied to solve the transport equations for turbulent flow and energy dissipation. The impact of non-dimensional parameters is analysed by varying the jet diameter ratio (D_j/D_h) 0.2167 to 0.4334 and jet space ratio 1.78 to 4.16 for Reynolds number (Re) variations from 5000 to 18000 using 1000 W/m^2 heat flux on absorber plate. The Momentum and energy equations was discretized using the second-order upwind scheme. Pressure and velocity coupling were handled by the SIMPLE algorithm and convergence criteria was set with residuals of 10^{-6} for continuity, momentum, energy, k, and ε equations to ensure accurate results. The simulation results indicate significant improvement in Nusselt number (Nu) and thermohydraulic performance parameter (THPP) of this jet plate solar air heater (JPSAH) using cylindrical jet impingement. Maximum THPP is expected in between 1.5 and 2. To elucidate the air flow characteristics in the turbulent zone pressure contours, velocity contours and turbulent kinetic energy contours are provided and discussed. Incorporating artificial roughness in proposed model can bring future scope for further development in jet plate solar air heater.

Keywords: *Solar air heater (SAH), Jet Plate, Thermohydraulic Performance Parameter (THPP),*

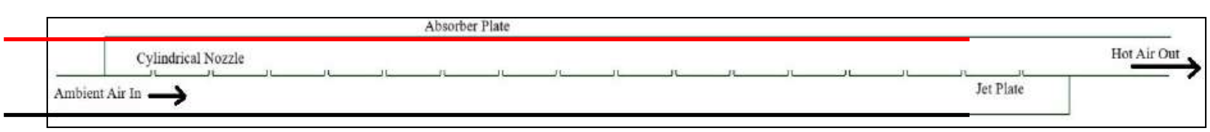


Fig. 1. Schematic of the problem geometry.

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

A COMPARATIVE EVALUATION AND ANALYSIS OF PIPE MATERIALS ON THE THERMAL AND OPERATIONAL PERFORMANCE OF EARTH–AIR HEAT EXCHANGERS

Shashank Saxena[1], Piyush Kumar Jain[1] and Kalpana Sachan[1]
[1]Department of Mechanical Engineering, Bansal Institute of Science and Technology, Bhopal
shashanksaxena7oct@gmail.com

ABSTRACT

Earth-Air Heat Exchangers (EAHEs) use the naturally stable temperature of the soil to cool or heat air, helping buildings reduce electricity use. One of the most important decisions in designing an EAHE is choosing the right pipe material, because it directly affects how well the system transfers heat, how much energy it consumes and how long it lasts underground. This study compares different pipe materials such as copper, stainless steel, aluminum, PVC, HDPE and concrete to understand how their thermal conductivity, durability, corrosion resistance, cost and pressure losses influence EAHE performance. The results show that metals like copper transfer heat very efficiently but are more expensive, while polymer pipes (PVC and HDPE) are cheaper, easier to install and highly resistant to corrosion but offer lower heat transfer. Metals such as stainless steel and aluminum offer superior heat transfer properties but are costly and prone to corrosion. Concrete pipes, while durable, present challenges in handling and installation due to their weight and high frictional resistance. In conclusion, copper's consistently high ratings and superior numerical performance make it an optimal choice for EAHE systems, provided any identified drawbacks are considered for specific cases. The copper pipes have thermal conductivity values around 399 W/m·K, significantly outperforming aluminum and polymeric alternatives. This leads to higher heat transfer rates and improved system efficiency, though at greater initial cost. Longevity and maintenance requirements are also favorably impacted by copper's durability. The Figure shows the experimental setup EAHE made up of PVC material.

Key Words: *Earth to Air Heat Exchanger; Thermal Conductivity; Stainless Steel; heating potential, cooling potential.*

Figure: 1 EAHE setup of PVC material.

DYNAMICS OF GUIDEWIRE-DRIVEN FLOW ALTERATION IN CORONARY ARTERIES

Aditya Kumar[1], Sayan Karmakar[2] Supratim Saha[3]
[1]Department of Mechanical Engineering, Birla Institute of Technology and Science, Pilani, India, asinghania@gmail.com
[2]Department of Mechanical and Industrial Engineering, University of Illinois Chicago, Chicago, USA, skarm5@uic.edu
[3]Department of Mechanical Engineering, Indian Institute of Technology Madras, Chennai, India, supratimiitm@gmail.com

ABSTRACT

We present a fully coupled fluid–structure interaction (FSI) framework to quantify how a pressure guidewire and multi-layer arterial mechanics jointly alter pulsatile coronary hemodynamics. The model represents a two-dimensional axisymmetric left coronary artery (LCA) with radius R = 1.7 mm and length 10R. A cosine-shaped 50% diameter stenosis, extending 2R in length, is embedded in the lumen. The arterial wall is modeled as a three-layer hyperelastic structure. The intima, media, and adventitia namely three laters of the arterial wall, occupy 0.25%, 0.35%, and 0.40% of a wall thickness equal to 20% of the lumen diameter. The fibrous cap follows a Mooney–Rivlin formulation, while the lipid pool is treated as a nearly incompressible linear elastic material with E = 100 KPa and poisson ratio 0.49 . Blood is modeled as an incompressible shear-thinning Carreau fluid with density 1056 kg/m3. Fluid and solid deformation are coupled through an arbitrary Lagrangian–Eulerian (ALE) moving-mesh method. A fully developed pulsatile inlet waveform corresponding to 67 beats per minute (BPM) is prescribed. A constant mean arterial pressure (pmean) is imposed at the outlet. A rigid 0.36 mm diamters guidewire is positioned centrally and extends 8R into the stenotic segment. Its presence introduces an additional geometric constriction and modifies local shear and pressure fields. The framework captures guidewire-induced flow redistribution, plaque–wall deformation, and stenotic jet dynamics, providing new mechanistic insight into pressure-wire-based coronary assessment.

Key Words: *Guidewire, FSI, Coronary stenosis, Hyperelastic modeling.*

THERMOPHYSICAL AND THERMOACOUSTICAL STUDIES OF BINARY LIQUID MIXTURE OF N, N-DIMETHYLACETAMIDE WITH DIMETHYL SULFOXIDE AT DIFFERENT TEMPERATURES

Rolee Verma[*1] and Ranjan Dey[1]
[1,2] Thermophysical Properties Lab, Department of Chemistry, Birla Institute of Technology and Science, Sancoale, Goa, 403726, India
Email[*]: **ranjandey@goa.bits-pilani.ac.in**

ABSTRACT

Industrial solvents are used to dissolve, suspend or extract other materials without chemically changing either the solvents or the other materials. It has various applications in cleaning, degreasing, painting, coating, chemical synthesis, pharmaceuticals, adhesives, extraction process, heat and mass transfer, polymer processing and biochemical systems. N, N-Dimethylacetamide (DMAc) and Dimethyl sulfoxide (DMSO) are most widely used solvents. In present investigation, the thermophysical study of Dimethylacetamide (DMAc) and Dimethyl sulfoxide (DMSO) has been carried out at different temperatures over the entire mole fraction. Thermophysical and allied parameters have been evaluated such as acoustic impedance (z), surface tension (σ), isothermal compressibility (β-T), internal pressure (Pi), non-linearity parameter (B/A) and intermolecular free length (Lf) for the binary liquid mixture of N, N-dimethylacetamide (DMAc) with Dimethyl sulfoxide (DMSO) at different temperatures (298.15 – 313.15K). The novelty of this work lies in its investigation at different temperatures compared to the available data. These parameters are used to interpret the molecular interaction present in the liquid mixtures. The obtained results indicate that there is a strong intermolecular interaction between DMAc and DMSO.

Keywords: *N, N-dimethylacetamide, Dimethyl sulfoxide, and Thermoacoustical*

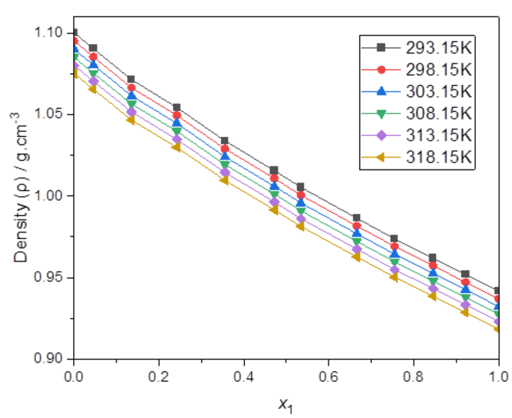


Figure1. The plot of density vs mole fraction at different temperatures for the binary liquid mixture of DMAc with DMSO

CFD SIMULATION APPROACH FOR DRYING PROCESS OF LARGE CARDAMOM USING TRADITIONAL BHATTI

Moumita Roy[1], Prokash C. Roy[2] and Shambhunath Barman[3]
[1,3]Department of Mechanical Engineering, National Institute of Technology Sikkim, South Sikkim 737139, India
[2]Department of Mechanical Engineering, Jadavpur University, Kolkata-700032, India
Email: phme230012@nitsikkim.ac.in

ABSTRACT

Large cardamom, also known as black cardamom, is one of the oldest spices with a unique history, rich flavour, medicinal quality, and diverse uses. The main cultivation of this spice is done in the Eastern Himalayas of India, Nepal, and Bhutan. More than 85% of India's production is from Sikkim. This study aims at the post-harvest drying process for large cardamoms. Large cardamoms are dried in "bhattis," which are ancient curing chambers. The massive volume of firewood needed in this method to produce heat raises environmental problems. Modern modelling and simulation methods are helpful in the development of innovative dryers, energy conservation, and process optimisation. A well-known modelling method that has recently grown in popularity in food operations is computational fluid dynamics (CFD). This paper briefly describes the construction and post-harvest processing of the traditional curing chambers. This paper also examines the numerical simulation of natural convection heat transfer in models with an optimal distribution of a heat source attached to the bottom of the heating chamber. Using a thermal imager, the temperature variations within the bhatti are observed and recorded to validate the result obtained by numerical simulation. Air velocity enhances temperature dispersion, ensuring that temperatures are distributed uniformly. As a result, it is also necessary to investigate the chamber's velocity profile. The results indicate significant thermal stratification and energy loss zones, which have an immediate impact on fuel efficiency and drying uniformity. The knowledge gained provides a solid foundation for enhancing the quality and uniformity of large cardamom curing, improving combustion efficiency, and optimising Bhatti design. All things considered, this study adds fresh, useful information that helps modernise and sustainably enhance conventional drying methods.

Key Words: *Computational Fluid Dynamics (CFD), Thermal Analysis, Natural Convection, Traditional Drying Chambers (Bhatti)*

INTERMOLECULAR INTERACTIONS IN TERNARY MIXTURE OF INDUSTRIAL SOLVENTS (*N*METHYL-2-PYRROLIDONE + DIMETHYL SULFOXIDE + WATER): A THERMOPHYSICAL AND THERMOACOUSTICAL INVESTIGATION

Eshita Kakodkar[1] and Ranjan Dey[1]

[1]Thermophysical Properties Lab, Department of Chemistry, Birla Institute of Technology and Science, Sancoale, Goa, 403726, India. p20240010@goa.bits-pilani.ac.in

ABSTRACT

Thermophysical properties of industrial solvents are essential for determining their usage in various applications such as extraction processes, solvent recovery, separation processes, to help design newer sustainable solvent systems, etc. The experimental determination of various thermodynamic, transport and optical properties such as density, ultrasonic velocity, viscosity, refractive index helps reveal the intricate intermolecular interactions occurring in the liquid mixtures. The estimation of the excess molar volumes (V_m^E), excess Gibb's energy, molar refraction along with isothermal compressibility (β_s), coefficient of thermal expansion(α_p) and their excess counterparts further strengthen our understanding and gives us an insight into the geometrical packing and structural organisations occurring in the binary and multicomponent mixtures. This investigation involves the evaluation of the thermophysical properties like internal pressure, enthalpy of vaporisation, cohesive energy density, etc. and thermoacoustical parameters such as non-linearity parameter, isothermal compressibility, Isochoric acoustical parameter, etc. for the ternary mixture of DMSO + NMP + water for the entire mole fraction range and at varied temperatures ranging 293.15 K-318.15 K for the first time. These properties give us an idea about the non-ideal behaviour and structural changes occurring in these mixtures which helps us in effectively employing them in various industrial and pharmaceutical applications that could help design their alternative greener and sustainable solvent systems.

Key Words: *Industrial solvents, Thermophysical Properties, Molecular interaction*

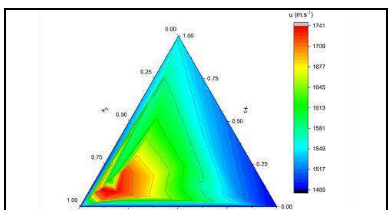


Figure 1. Ultrasonic velocity of ternary mixture of DMSO(x_1) +NMP(x_2) +Water(x_3) at 298.15 K

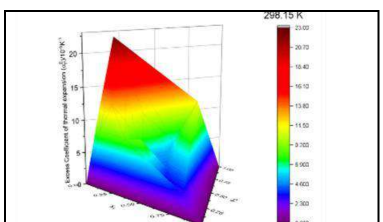


Figure 2. Excess Coefficient of thermal expansion (α_P^E) of of ternary mixture of DMSO(x_1) +NMP(x_2) +Water(x_3) at 298.15 K

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

AI ASSISTED NANOFLUID BOILING AND SURFACE ENGINEERING FOR ENHANCED HEAT TRANSFER PERFORMANCE

Jinendrika Anushi Weliwita[1], Saleimah Alyammahi[1], Sanjeeva Witharana[2]

[1]Department of Mechanical Engineering, Higher Colleges of Technology, Fujairah, UAE. Email: jweliwita@hct.ac.ae

[2]Department of Mechanical Engineering, University of Moratuwa, Sri Lanka, UAE. Email: switharana@ieee.org

ABSTRACT

Effective heat transfer remains a major challenge in high temperature thermal systems, where boiling plays a critical role in dissipating extreme heat fluxes. Nanofluids engineered suspensions of nanoparticles in conventional base fluids have shown potential for enhancing boiling performance, yet their performance remains difficult to predict. This study integrates controlled experimental investigation with physics informed artificial intelligence to understand, quantify, and optimize boiling heat transfer in nanofluid based cooling systems. A series of pool boiling experiments was conducted using TiO_2, Al_2O_3, and carbon nanotube (CNT) nanofluids dispersed in water ethylene glycol mixtures. Experiments were performed on polished and engineered copper heater surfaces under atmospheric conditions to measure heat transfer coefficients (HTC), critical heat flux, bubble nucleation dynamics, and nanoparticle deposition morphology. Multimodal diagnostics including infrared thermography, small angle X-ray scattering, and scanning electron microscopy were used to resolve temperature fields, aggregation behavior, active nucleation site distribution, and multiscale surface fouling structures. These measurements showed that nanoparticle concentration, aggregation state, and evolving surface topology strongly influence boiling enhancement and stability. CNT nanofluids exhibited up to 25% HTC enhancement at 120 kW/m^2, while Al_2O_3 nanofluids showed 8 - 12% deterioration under identical conditions. Measured CHF enhancements ranged from 60% to 300%, depending on nanoparticle type and loading. A framework with AI was developed to detect predictive descriptors governing nanofluid boiling behavior. Convolutional neural networks extracted spatial features associated to bubble distribution and deposition morphology, while recurrent neural networks modelled temporal boiling evolution and heat flux progression. The models were constrained using physically grounded relationships derived from the Young Laplace equation, bubble growth kinetics, and thermodynamic scaling laws. The resulting hybrid model accurately predicted HTC enhancement trends and revealed new insights: AI derived enhancement curves for CNT nanofluids increased from 1.00 to 1.26 across the 60- 120 kW/m^2 range, and feature importance analysis identified aggregation index, deposit layer thickness, and nucleation site density as the most influential parameters. The AI also identified an optimal nanoparticle concentration window of 0.05 to 0.1 wt% that balances enhancement with operational stability. The amalgamated experimental AI approach demonstrates that nanoscale particle structuring and evolving surface morphology play dominant roles in macroscopic boiling performance. The findings provide data driven guidelines for nanofluid formulation and surface engineering strategies, enabling next generation thermal management solutions. By linking nanoscale behavior, engineered surfaces, and AI based predictive modelling, this work establishes a comprehensive pathway for designing adaptive, high performance boiling heat transfer technologies.

Key Words: *Nanofluids, Boiling Heat Transfer, Physics Informed AI, Nanoparticle Aggregation, Thermal Management*

MINERALISATION OF INDUSTRY-RELEVANT SOLVENT POST USAGE FROM WASTEWATER USING DIELECTRIC BARRIER DISCHARGE PLASMA

[1]Eshita Kakodkar and [1]Ranjan Dey

[1]Thermophysical Properties Lab, Department of Chemistry, Birla Institute of Technology and Science, Sancoale, Goa, 403726, India. p20240010@goa.bits-pilani.ac.in

ABSTRACT

The chemical industry is the backbone of the national economy. A variety of chemicals are produced in and transported worldwide. To synthesize such chemicals, industry and process-relevant solvents are employed. In most cases, solvents are reused multiple times via distillation or other separation and purification techniques. However, there is a limit to reusability, as the solvent's properties change, rendering it useless. These chemicals are treated or stored. There is a spillage of chemicals during transportation or treatment, which mixes with soil or water, making them unfit for use. These pollutants are often carcinogenic, hazardous to human health, and can cause severe health problems ranging from mild allergies to cancerous tumours. In the present study, we highlight the importance of dielectric barrier discharge plasma as a medium for degrading contaminants in wastewater. Plasma is a partially ionised gas that produces extremely energetic electrons that transmit energy to atoms via collisions. In this study, we will carry out the degradation of two commonly used industrial solvents, namely dimethyl sulfoxide (DMSO) and N-methyl-2-pyrrolidone (NMP). As per many studies, these pollutants exhibit strong molecular interactions with water, and plasma may act as a driving force to break these interactions, thereby degrading the industrial pollutants. The various parameters, such as plasma irradiation power and time, will be varied, and their effects on pollutant degradation will be explored using spectroscopic analysis.

Keywords: *Industrial solvents, Dielectric barrier discharge plasma, Wastewater*

FEM ANALYSIS OF BACK FACE DEFORMATION ON THE BULLETPROOF VEST

Ravi Kant Rajak[1], Shambhunath Barman[1] and Moumita Roy[1]
[1]National Institute of Technology Sikkim, Ravangla South Sikkim (737139), b220146@nitsikkim.ac.in

ABSTRACT

Lightweight composite materials are widely used in personal protective systems due to their high energy absorption capacity and reduced weight. While preventing projectile penetration is essential, excessive back-face deformation (BFD) can still cause severe blunt trauma, making it a critical parameter in armour design. This study investigates the impact response and back-face deformation of a multilayer composite armour system subjected to high-velocity projectile impact.

The armour model consists of a Kevlar laminate measuring 0.3 m × 0.3 m with a total thickness of 5.5 mm, composed of twenty plies arranged in varying fiber orientations. A Steel 4340 projectile is considered as the impacting threat. An explicit dynamic finite element approach is employed to simulate the impact event, enabling the capture of stress wave propagation, deformation behaviour, and energy absorption mechanisms within the composite laminate.

The simulation results indicate that the laminate effectively dissipates impact energy through fiber stretching, inter-laminar interactions, and localized deformation of the projectile. The predicted back-face deformation remains within acceptable limits, demonstrating the suitability of the proposed configuration for lightweight armour applications. The findings provide valuable insight into the influence of laminate architecture on ballistic performance and contribute toward the design and optimization of advanced composite armour systems.

Key Words: *Back-face deformation (BFD); Composite armour; Kevlar laminate; Ballistic impact; Explicit dynamic analysis; Energy absorption; Finite element method*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

NUMERICAL STUDY OF SAVONIUS TYPE WIND TURBINE USING COMPUTATIONAL FLUID DYNAMICS

Hiranmoy Samanta[1], Sk. Tarif Ali[2], Debajit Saha[1] and Shambhunath Barman[1]

[1]NIT Sikkim, Sikkim, Ravangla, hiru.samanta@gmail.com, debajit.saha1986@nitsikkim.ac.in, shambhunath.barman@nitsikkim.ac.in, Ola Electric, Mumbai, India, sktarifali111@gmail.com

ABSTRACT

The Savonius wind turbine is widely used in small-scale wind energy conversion systems due to its simple construction and ability to operate at low wind speeds. However, its conventional rotor design often results in relatively low efficiency and rotational speed. This study investigates the performance enhancement achieved through a modified Savonius Type-S rotor incorporating rib-based blade variations. The novelty of this work lies in introducing and evaluating rib configurations on the blade surfaces to improve aerodynamic interaction, torque characteristics, and overall energy capture—an area that remains insufficiently explored in existing literature.

A three-dimensional computational fluid dynamics (CFD) analysis was carried out using ANSYS 22, with the rotor model (diameter 1.1 m, height 1.4 m) developed in SolidWorks. Simulations were conducted at wind speeds of 3–5 m/s using the k–ε turbulence model. Grid-independence testing showed stable moment coefficient values, with mesh sizes of 122,060 and 260,876 elements yielding errors of only 0.33% and 0.19%, respectively. A dynamic mesh approach was implemented, and the model was exported to FLUENT for transient flow simulation within a 2000 mm × 1500 mm × 1500 mm computational domain. The inlet velocity was set at 4.39 m/s, with a tip-speed ratio (TSR) of 0.8 and an angular velocity of 6.5 rad/s, consistent with the drag-type nature of the Savonius rotor (TSR < 1.0).

Key performance parameters - including drag coefficient, torque, moment coefficient, and power coefficient - were evaluated for the baseline and rib-modified rotors. The modified design demonstrated measurable improvements in torque generation and power coefficient compared to the standard Savonius configuration, highlighting the effectiveness of rib-induced flow guidance in enhancing rotor performance. These findings provide new insights into geometric optimization for low-speed vertical-axis wind turbines and offer a practical pathway for improving small-scale renewable energy systems. Overall, the study provides clear evidence that rib-based blade modifications can significantly improve the aerodynamic efficiency of Savonius turbines. The findings contribute valuable insights into the geometric optimization of vertical-axis wind turbines and highlight a promising pathway for enhancing small-scale renewable energy systems, particularly in regions with low to moderate wind speeds.

Keywords: *Savonius Wind Turbine; Computational Fluid Dynamics (CFD); Aerodynamic Performance; Rib-Modified Blades; Vertical-Axis Wind Turbine*

NUMERICAL MODELLING OF LI-ION BATTERY THERMAL MANAGEMENT SYSTEM

Hiranmoy Samanta[1], Sk. Tarif Ali[2], Debajit Saha[1] and Shambhunath Barman[1]

[1]NIT Sikkim, Sikkim Ravangla, hiru.samanta@gmail.com, debajit.saha1986@nitsikkim.ac.in, shambhunath.barman@nitsikkim.ac.in,
[2]Sk. Tarif Ali, Ola Electric, Mumbai, India , sktarifali111@gmail.com

ABSTRACT

The primary objective of this study is to evaluate and enhance the thermal performance of an air-cooled lithium-ion battery pack by analysing its heat dissipation characteristics and temperature distribution under a wide range of operating conditions. The novelty of the work lies in the integrated parametric investigation of air velocity, inlet air temperature, and discharge rate—conducted simultaneously—together with a detailed spatial temperature assessment along longitudinal, transverse, and circumferential directions of individual cells. Such a comprehensive analysis offers new insights into the coupled influence of flow conditions and cell loading on thermal non-uniformity in compact air-cooled Battery Thermal Management Systems (BTMS).

The numerical model is developed in ANSYS FLUENT 22, where the Navier–Stokes equations are solved using the Shear Stress Transport (SST) k–ω turbulence model. The simulation framework allows precise control of key thermal-hydraulic parameters, including air velocity from 0.5 to 5.0 m/s, inlet air temperature between 282 and 315 K, and discharge rate ranging from 0.5C to 2.5C. Model validation against experimental data shows strong agreement, with a maximum deviation of 3.12%.

Performance is evaluated using four key indicators: maximum cell temperature, average temperature, maximum temperature difference, and temperature standard deviation within the pack. Results reveal that increasing air velocity from 0.5 to 5.0 m/s can reduce maximum cell temperature by up to 18.6%, while lowering discharge rate from 2.5C to 0.5C decreases peak temperature rise by approximately 12–20 K depending on cell geometry. A single lithium-ion polymer cell exhibits a temperature increase of 5–20 K at a 1°C discharge rate. Furthermore, for a fixed inlet temperature, a distinct thermal gradient develops along the airflow direction, with the inlet-to-outlet temperature difference reaching 22.14 K under extreme conditions and reducing to 0.21 K under mild conditions.

Higher air velocity also improves circumferential temperature uniformity, reducing local temperature variation in hotspot regions by up to 35%. The combined findings confirm that optimized airflow and discharge strategies significantly mitigate hotspot formation and enhance BTMS cooling effectiveness.

Keywords: *Lithium-ion Battery; CFD; Thermal Modelling; Air Cooling; BTMS*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

EXPERIMENTAL STUDY ON THE EFFECTIVENESS OF WAVY FINS FOR AIR-COOLED LITHIUM-ION BATTERY THERMAL MANAGEMENT

Ashish Kumar Saini[1], Bhupendra Kumar Sharma[1], Suvanjan Bhattacharya[2]

[1]Department of Mathematics, Birla Institute of Technology and Science Pilani, Rajasthan, India,
ashishchirawa74@gmail.com, bhupen_1402@yahoo.co.in
[2]Department of Mechanical Engineering, Birla Institute of Technology and Science Pilani, Rajasthan, India,
suvanjan.bhattacharyya@pilani.bits-pilani.ac.in

ABSTRACT

Effective temperature control is essential for lithium-ion batteries (LiBs), as continuous heat generation during charge–discharge cycles can lead to uneven temperature profiles, reduced efficiency, and potential safety hazards. In this work, we experimentally evaluate a battery thermal management system (BTMS) that incorporates wavy fins under forced air cooling to enhance heat removal through extended surface area and flow disturbance. The proposed system is assessed against two reference cases—a battery without air cooling and a battery with fins but without airflow. Experiments are carried out at ambient temperatures of 25 °C, 30 °C, 40 °C, and 45 °C, and at heat generation levels of 10 W and 15 W to represent typical operating scenarios. Throughout the tests, temperature variation is continuously monitored with an emphasis on peak temperature and temperature uniformity across the cell surface. The results show that the battery without cooling reaches high temperatures rapidly, while the fin-only configuration provides partial improvement but is insufficient at elevated ambient conditions. In contrast, the wavy-fin air-cooling system demonstrates a notable decrease in peak temperature and a more consistent temperature field due to enhanced airflow mixing and prolonged heat-transfer contact. These observations confirm that wavy fins combined with air cooling offer a practical and lightweight thermal management solution for stable LiB operation in demanding environments.

Key Words: *Lithium-ion battery; Battery thermal management system ; Air cooling; Wavy fins; Heat dissipation; Forced convection.*

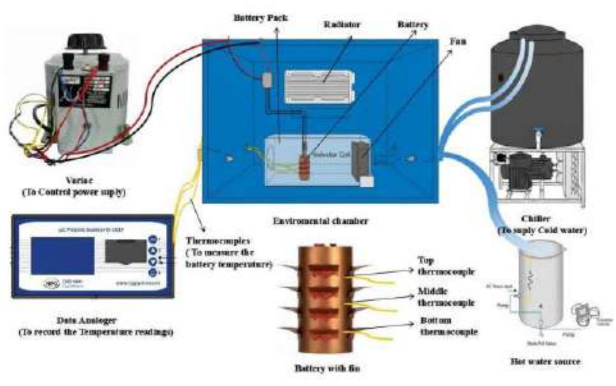


Figure 1: Schematic setup of experiment

DESIGN AND THERMAL SIMULATION OF PASSIVE VAPOR CHAMBER FOR HOT SPOT MITIGATION IN HIGH POWER ELECTRONIC DEVICES

Mayank Mani Pandey[1], Sohan Pattanayak[1],Suvanjan Bhattacharyya[1], Saket Verma[1,2], and Md. Qaisar Raza[3]

[1]Department of Mechanical Engineering, Birla Institute of Technology & Science Pilani, Rajasthan – 333031, India e Affiliation, Postal Address, E-mail addresses- p20210466@pilani.bits-pilani.ac.in, f20220673@pilani.bits-pilani.ac.in , suvanjan.bhattacharyya@pilani.bits-pilani.ac.in

[2]School of Energy Science and Engineering, Indian Institute of Technology (IIT) Guwahati, Assam – 781039, India, E-mail addresses- saket@iitg.ac.in

[3]Department of Mechanical Engineering, National Institute of Technology (NIT) Patna, Bihar - 800005, India, E-mail addresses- qraza.me@nitp.ac.in

ABSTRACT

The thermal management is critical for the reliable operation of high-power electronic devices, where localized heat accumulation can severely impact their performance. This study presents a numerical investigation of a Vapor Chamber (VC) based passive thermal management designed as a heat spreader for electronic cooling. Unlike many existing studies that focus on simplified or experimental evaluations, this work implements a comprehensive 3-dimensional multilayer model incorporating copper plates, wick structures, and a central vapor core. The vapor and wick regions are modeled using effective thermal conductivities to capture the phasechange-driven heat transport, while parametric variations are applied to assess design sensitivity. The model was validated against benchmark data, showing case temperature deviations within 5%, confirming its predictive accuracy. Parametric studies revealed that increasing wick thermal conductivity from 30 W/m°K to 70 W/m°K showed reduction in junction temperature from 59°C to 55°C, against the value of 67°C to 63°C for the benchmark study showing ~12% drop, while optimizing vapor space height further improved thermal spreading. The highest convective heat transfer coefficient studied (55,000 W/m²·K) reduced case temperature by ~8°C. Additionally, increasing the heat flux from 60 W/cm2 to 90 W/cm2 led to a 10-12% reduction in case temperatures. Overall, the VC exhibited substantial lower junction to ambient resistance for variable heat source footprints form 0.27 °C/W to 0.07 °C/W (15% drop) demonstrating its potential as a compact, passive, and efficient solution for nextgeneration electronic cooling.

Key Words: *Electronic-cooling, heat spreader, vapor chamber, thermal management*

BITS Pilani
Pilani Campus

SPARC
Scheme for Promotion of Academic and Research Collaboration

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

ENHANCED CONVECTIVE COOLING OF LITHIUM-ION BATTERIES USING DIMPLED FINS: AN EXPERIMENTAL APPROACH

Ashish Kumar Saini[1], Bhupendra Kumar Sharma[1], Suvanjan Bhattacharya[2]

[1]Department of Mathematics, Birla Institute of Technology and Science Pilani, Rajasthan, India,
ashishchirawa74@gmail.com, bhupen_1402@yahoo.co.in
[2]Department of Mechanical Engineering, Birla Institute of Technology and Science, Pilani, Rajasthan India,
suvanjan.bhattacharyya@pilani.bits-pilani.ac.in

ABSTRACT

Reliable thermal management of lithium-ion batteries (LiBs) is essential for improving performance, ensuring safety, and preventing thermal runaway in high-power applications. Conventional air-cooling methods often struggle to maintain acceptable temperature limits under elevated ambient conditions, leading to efficiency loss and accelerated degradation. To address this limitation, this study experimentally evaluates an air-cooled battery thermal management system enhanced with dimpled fins to increase convective heat transfer. The proposed configuration is compared against two baseline setups—battery without air cooling and battery with fins but without airflow. Experiments are carried out at ambient temperatures of 25 °C, 30 °C, 40 °C, and 45 °C with heat generation levels of 10 W and 15 W to emulate realistic operating conditions. Temperature rise, peak temperature, and temperature uniformity are monitored to assess cooling performance. Results indicate that the no-cooling case shows the highest temperature escalation, while the fin-only system offers moderate improvement but fails to maintain safe thermal limits at higher ambient temperatures. In contrast, the combination of dimpled fins and forced air cooling significantly reduces peak temperature and enhances temperature uniformity due to increased turbulence and boundary-layer disruption. Overall, the findings demonstrate that dimpled fin–assisted air cooling provides a lightweight, cost-effective, and efficient solution for safe and thermally stable lithium-ion battery operation.

Key Words: *Lithium-ion battery; Battery thermal management system,Air cooling; Dimpled fins; Heat dissipation; Forced convection.*

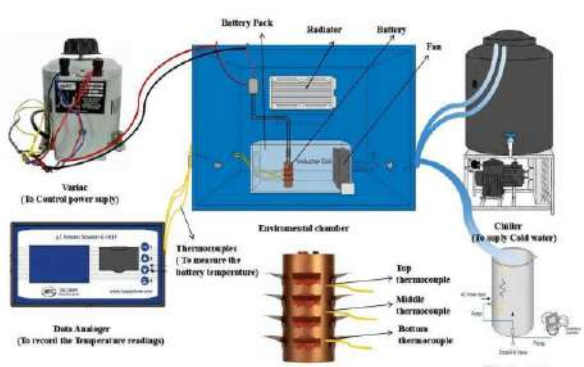


Figure 1: Schematic setup of experiment

INVESTIGATION OF CIRCULAR MINICHANNEL-BASED THERMAL MANAGEMENT FOR ELECTRIC VEHICLE BATTERIES

Nancy Maurya[1], Neerav Krishna[1], Suvanjan Bhattacharyya[1*]

[1]Department of Mechanical Engineering, Birla Institute of Technology and Science Pilani, Pilani Campus, Vidya Vihar, Rajasthan 333031, India.

Corresponsing Email: suvanjan.bhattacharyya@pilani.bits-pilani.ac.in

ABSTRACT

The global shift toward environmental sustainability and the need to reduce the negative impacts of fossil-fuel-based transportation have accelerated the adoption of battery-electric mobility. As electric vehicles gain wider prominence, the development of reliable and highly efficient Battery Thermal Management Systems (BTMS) has become indispensable. Effective thermal regulation is crucial for ensuring operational safety, limiting degradation, and preserving the electrochemical performance of lithium-ion batteries. Without proper heat management, batteries may experience accelerated aging, diminished energy capacity, compromised State of Charge (SoC), and, in severe cases, thermal runaway. This study presents a computational analysis of circular minichannel cooling configurations in combination with advanced coolant media, including nanofluids and Phase Change Materials (PCMs). Owing to their superior thermophysical properties—such as higher thermal conductivity and enhanced heat absorption—these coolants demonstrate strong potential for augmenting heat dissipation compared to conventional water-based systems. The investigation evaluates key performance indicators, including the Nusselt number (Nu), Colburn j-factor (j), friction factor (f), and Thermal Enhancement Factor (TEF), offering comprehensive insight into convective heat transfer behavior and system efficiency. Using an 18650 lithium-ion cell as the reference geometry, three circular minichannel diameters (2.0 mm, 4.0 mm, and 6.0 mm) are assessed under a 0.5 C discharge rate. Simulations examine the influence of coolant type, flow characteristics, and channel dimensions on both thermal and electrochemical parameters, including SoC and Depth of Discharge (DoD). Results show a consistent improvement in heat transfer performance with increasing Reynolds number (Re). Among the configurations studied, the 4.0 mm minichannel yields the most balanced performance, offering notable enhancements in Nu and j-factor while maintaining moderate frictional losses, as reflected in favourable TEF values. Overall, the findings highlight the potential of optimized minichannel geometries combined with advanced coolants to significantly improve BTMS effectiveness, supporting safer and more sustainable electric vehicle operation.

Key Words: *Heat transfer, battery thermal management system, electric vehicle, minichannel, nanofluids*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

THERMAL PERFORMANCE ENHANCEMENT OF EV BATTERIES USING MAGNETO-VIBRATIONALLY ASSISTED CIRCULAR MINICHANNEL COOLING

Nancy Maurya[1], Neerav Krishna[1], Suvanjan Bhattacharyya[1*]

[1]Department of Mechanical Engineering, Birla Institute of Technology and Science Pilani, Pilani Campus, Vidya Vihar, Rajasthan 333031, India.

Corresponsing Email: suvanjan.bhattacharyya@pilani.bits-pilani.ac.in

ABSTRACT

The growing global emphasis on sustainability and the need for cleaner energy solutions have accelerated the transition of the automobile sector toward battery-driven technologies. In this context, effective battery thermal management has become critical to ensure safety, performance, and longevity. The present study computationally investigates the cooling performance of a circular minichannel system using various advanced coolants such as nanofluids and phase change materials (PCM), along with supplementary enhancement techniques including magnetic field application and induced vibrations. These methods are explored as potential avenues to intensify convective heat transfer within the cooling channel. The analysis focuses on key thermal and flow characteristics—Nusselt number (Nu), Colburn j-factor (j), friction factor (f), Thermal Enhancement Factor (TEF), State of Charge (SoC), and Depth of Discharge (DoD)—and evaluates their variation with increasing Reynolds number (Re), comparing performance against a baseline case using water as the coolant. The computational domain consists of a 2D channel with an 18650 Li-ion battery serving as the reference configuration. Additionally, the study examines the effect of varying channel diameters—2.0 mm, 4.0 mm, and 6.0 mm—at a discharge rate of 0.5 C. The results show that combining nanofluids with magnetic field–induced flow manipulation and vibration-assisted convection significantly improve thermal performance. Among the tested geometries, the 4.0 mm channel demonstrates the most substantial enhancement across all evaluated parameters, indicating its superior suitability for integration into high-performance battery packs.

Key Words: *Heat transfer, battery thermal management system, electric vehicle, minichannel, vibration*

THERMAL PERFORMANCE EVALUATION OF NOVEL HYBRID PCM–LIQUID COOLED BATTERY THERMAL MANAGEMENT SYSTEM FOR ELECTRIC VEHICLES

Surjo Das[1,*], Srirama Nrisimha Sai Sripada[1], Penta Jayvardhan[1], Srinjoyee Chakravarty[1], Suvanjan Bhattacharyya[2]

[1,2,*] Department of Mechanical Engineering, Birla Institute of Technology and Science Pilani, Pilani Campus, Vidya Vihar, Pilani, Rajasthan - 333031, India

*Corresponding Author: f20221274@pilani.bits-pilani.ac.in

ABSTRACT

The present study explores a novel hybrid battery thermal management system (BTMS) design integrating phase change materials (PCMs) with a liquid-cooled channel for efficient heat transfer in electric vehicle (EV) batteries. Transient numerical simulations were conducted using commercially available CFD software, ANSYS Fluent 18.1, for the study. A comprehensive literature review guided the selection of organic and inorganic PCMs for the different range of operating temperatures. Model validation was performed using benchmark experimental data from previous studies to ensure numerical reliability. The simulations were conducted on PCMs RT42, RT58, and Lauric Acid for three battery surface temperatures (45 °C, 50 °C, and 60 °C) and four Reynolds numbers (750, 1000, 1500, and 2000) to assess the influence of flow rate and temperature on melting characteristics. Results indicate that both higher inlet temperatures and increased Reynolds numbers significantly accelerate the PCM melting process, enhancing thermal uniformity. Among the materials tested, RT42 exhibited the most desirable performance for medium-temperature BTMS operation, achieving complete melting within 3000–7200 time steps depending on the thermal load, and maintaining stable heat absorption throughout the cycle. Lauric Acid showed a more gradual yet complete phase transition (~7900 steps), suggesting effective and uniform thermal buffering for moderate operating conditions. Conversely, RT58 demonstrated partial melting even after 9000 steps, indicating slower heat absorption but superior long-term temperature regulation under sustained heat loads. Overall, the combined effect of the PCM–liquid cooling system in thermal enhancement has been effectively simulated by the numerical model through various graphical and image illustrations and a temperature-based thermodynamic model, with RT42 emerging as the most suitable PCM for medium-temperature EV battery applications. The present study forms a benchmark for future optimisation of hybrid PCM–liquid cooling systems aimed at achieving energy-efficient, reliable, and sustainable battery thermal management in next-generation electric vehicles.

Key Words: *Phase Change Materials, Battery Thermal Management System, Thermal Energy Storage, Liquid Cooling, CFD, EV Battery*

NUMERICAL INVESTIGATION OF HEAT TRANSFER ENHANCEMENT OF TRIANGULAR PLATE FIN TYPE COMPACT HEAT EXCHANGERS WITH DELTA WINGLETS

Aditya Nair[1], Sri Harsha Dannana[1] and Suvanjan Bhattacharyya[1], Smith Eiamsa-ard[2]

[1]Department of Mechanical Engineering, Birla Institute of Technology and Science, Pilani, Pilani campus, Vidya Vihar, Rajasthan, 333031, India
[2]Department of Mechanical Engineering, Faculty of Engineering, Mahanakorn University of Technology, Bangkok 10530, Thailand
Corresponding Author Email id: *suvanjan.bhattacharyya@pilani.bits-pilani.ac.in*

ABSTRACT

Compact heat exchangers are widely employed in applications requiring high heat transfer rates within limited space, and their design must achieve a balance between enhanced thermal performance and acceptable pressure drop. Increasing heat transfer often incurs a penalty in terms of flow resistance, and this trade-off is central to the design process. Delta winglets have proven effective in improving mixing and disrupting boundary layers, thereby increasing heat transfer without an excessive rise in pressure drop. Understanding the combined influence of geometric parameters such as winglet angle of attack, pitch, and height on the overall performance is therefore essential for optimized design and selection of compact heat exchangers. The present study focuses on developing novel correlations for the Colburn *j* factor and friction factor *f* for a triangular plate fin compact heat exchanger equipped with staggered delta winglets. A detailed parametric analysis is conducted through three-dimensional steady-state computational fluid dynamics (CFD) simulations over a range of Reynolds numbers from 1000 to 10000 (including all flow regimes). The geometric configuration consists of a triangular plate fin array with delta winglets positioned in a staggered arrangement at a constant pitch. The parametric sweep includes systematic variation of winglet height, angle of attack, and pitch, which should provide sufficient data points to develop correlations for the *j* and *f* factors. The performance of the modified geometry is evaluated in terms of the thermal efficiency factor (TEF), defined as the ratio of normalized heat transfer enhancement to the cube root of the corresponding increase in friction factor. The TEF obtained from the simulations exhibits trends consistent with published studies, showing enhancement (TEF > 1) in the laminar and transitional flow regimes, where the heat transfer gains outweigh the associated pressure penalties. At higher Reynolds numbers, the pressure drop becomes more significant, reducing the net benefit of the winglet-induced enhancement, a finding that aligns with available research. The results demonstrate that the use of staggered delta winglets can significantly improve the thermal hydraulic performance of compact heat exchangers in specific flow regimes. The proposed correlations offer a practical means of predicting performance variations due to changes in design parameters, aiding in the preliminary design and optimization of heat exchangers.

Key Words: *Heat Transfer Enhancement, Delta Winglets, Compact Heat Exchangers.*

MULTI-PARAMETER OPTIMISATION OF A TWO-PASS PLATE-FIN HEAT EXCHANGER FOR AIRCRAFT ENVIRONMENTAL CONTROL SYSTEM

Vivek Bharti[1], Chennu Rangnayakulu[2, *] and Manoj Kumar Soni[2, **]

[1]Aeronautical Development Establishment, Bengaluru, India, p20220512@pilani.bits-pilani.ac.in
[2]Biral Institute of Science and Technology, Pilani, India,
[*]c.ranganayakulu@pilani.bits-pilani.ac.in, [**]mssoni@pilani.bits-pilani.ac.in

ABSTRACT

An aircraft Environmental Control System (ECS) ensures the delivery of thermally conditioned and pressurised air required for human comfort throughout the flight envelope. The system should also adhere to the stringent size, weight, and performance constraints. Compact Plate-Fin Heat Exchangers (PFHEs) are widely adopted due to their high surface-area-to-volume ratio and ability to withstand high-pressure and high-temperature environments. This research presents an optimisation-based design approach for a PFHE featuring Offset Strip Fins (OSF) on the charge side and wavy fins on the coolant side, targeting performance enhancement in medium- to high-speed aircraft. A baseline single-pass PFHE from the literature was first re-evaluated under 18 distinct flight phases extracted from a representative flight envelope. The heat exchanger geometry was described using thirteen design parameters, encompassing fin dimensions and core structural characteristics. An optimisation of all 13 geometrical parameters for the single-pass heat exchanger has been carried out. The study was extended to develop and evaluate a two-pass configuration on the charge side, while maintaining identical geometry and inlet conditions for both layouts. Results demonstrate a consistent improvement in effectiveness, yielding an average gain of approximately 11.8% compared to the single-pass design across all flight regimes. A Design-of-Experiments-based D-optimal optimisation strategy was implemented to evaluate thermohydraulic performance, core volume, and weight sensitivity. While several geometric combinations satisfied thermal and pressure-drop constraints, none outperformed the baseline configuration in terms of weight due to lower-limit assumptions on fin thickness. Furthermore, the two-pass configuration was found to be more effective at high Mach cruise and high bleed enthalpy operating regions, where heat duty is critical. Finally, using the same reference flight conditions, optimisation of all 13 parameters for a two-pass heat exchanger has been carried out.The proposed methodology establishes a systematic framework for PFHE optimisation under multi-parameter constraints, which can be extended to alternative fin geometries and multi-pass systems. The presented study contributes toward next-generation ECS thermal management solutions for advanced combat and trainer aircraft.

Key Words: Plate-Fin Heat Exchanger, Two-pass Heat Exchanger, Offset Strip Fins, Wavy Fins

EXPERIMENTAL AND NUMERICAL CHARACTERISATION OF TRANSITIONAL HEAT TRANSFER IN SMOOTH TUBES WITH TWISTED TAPE INSERTS UNDER CONSTANT HEAT FLUX

Sumit Khatri[1*], Tunde Bello-Ochande[2], Suvanjan Bhattacharyya[1]

[1]Department of Mechanical Engineering, Birla Institute of Science and Technology Pilani, Pilani Campus, Pilani 333031, Rajasthan, India.
[2]Department of Mechanical Engineering, University of Cape Town, Rondebosch, Cape Town, South Africa.
[*]Corresponding Author: p20230096@pilani.bits-pilani.ac.in

ABSTRACT

This study investigates the combined experimental and numerical characteristics of forced convection heat transfer in a smooth horizontal circular tube equipped with twisted tape inserts and subjected to varying heat flux conditions. Understanding the interaction between free and forced convection is essential for improving the thermal performance of heat exchangers, especially in systems operating within the transitional flow regime. Such regimes are critically important in industrial applications, including process cooling, chemical processing, and power generation, where thermal reliability and energy efficiency depend strongly on the nature of flow transition. To address this, a comprehensive experimental and CFD simulation was carried out over a broad range of operating conditions. The test section consisted of a smooth horizontal tube of 2000 mm length and 20 mm inner diameter with twisted tape inserts, using water as the working fluid. Both experiments and simulations were performed for Reynolds numbers ranging from 1000 to 8000, covering laminar, transitional, and early turbulent regimes. Four different constant heat flux levels, from 1 kW/m² to 4 kW/m², were applied to evaluate the influence of thermal loading on transition and heat transfer development. The experimental data were rigorously validated against established correlations in the literature as well as the CFD results to ensure accuracy and consistency. The investigation focused on identifying, characterising, and quantifying the transitional flow regime along the tube length. The flow domain was divided into distinct regions to analyse the evolution of local heat transfer coefficients and the interaction between buoyancy and inertia forces. Results show that increasing Reynolds number and enhanced forced convection advance the onset of laminar-to-turbulent transition. This shift significantly affects the local heat transfer distribution, with substantial enhancement observed in regions where free convection influences remain prominent. The presence of twisted tape inserts further accelerates thermal development and intensifies heat transfer across all regimes. Overall, the findings provide valuable insights into the mechanisms governing heat transfer in horizontally oriented tubes with inserts under varying thermal loads. These outcomes contribute to the optimised design of compact heat exchangers and other thermal management systems operating in transitional flow conditions, enabling improved energy efficiency and operational performance in industrial applications.

Key words: *Heat transfer, Transition flow regime, Twisted tape inserts, Heat exchangers, Critical Reynolds number, CFD simulations*

EXPERIMENTAL ANALYSIS OF FORCED CONVECTION HEAT TRANSFER DEVELOPMENT IN A ROUGH HORIZONTAL TUBE UNDER TRANSITIONAL FLOW CONDITIONS

Sumit Khatri[1*], Tunde Bello-Ochande[2], Suvanjan Bhattacharyya[1]

[1]Department of Mechanical Engineering, Birla Institute of Science and Technology Pilani, Pilani Campus, Pilani 333031, Rajasthan, India.
[2]Department of Mechanical Engineering, University of Cape Town, Rondebosch, Cape Town, South Africa.
[*]Corresponding Author: p20230096@pilani.bits-pilani.ac.in

ABSTRACT

This study presents a detailed experimental investigation into the influence of forced convection on the development of local heat transfer characteristics in a rough horizontal circular tube subjected to varying heat flux conditions. Understanding the interaction between free and forced convection is crucial for enhancing thermal performance and ensuring the efficient operation of heat exchangers, particularly in systems operating within the transitional flow regime. Transitional flows are of significant interest in many industrial processes where thermal reliability and energy consumption depend heavily on the evolution of flow and heat transfer behaviour. To examine these effects, a comprehensive experimental setup was designed and fabricated to evaluate heat transfer behaviour across a wide range of operating conditions. The test section consisted of a rough horizontal tube, 2000 mm in length with an inner diameter of 20 mm, using water as the working fluid due to its favourable thermal properties. Experiments were performed for Reynolds numbers between 1000 and 8000, thus covering laminar, transitional, and early turbulent flow regimes. Four constant heat flux levels, ranging from 1 kW/m² to 4 kW/m², were applied to investigate the influence of thermal loading on the transition process and heat transfer development. Experimental measurements were validated through comparison with established correlations and findings reported in the literature to ensure accuracy and consistency. The investigation focused on identifying and characterising the transitional regime along the tube length. The flow was analysed into distinct regions, allowing a detailed assessment of local heat transfer coefficient variations and the interplay between buoyancy and inertia effects. Results indicated that increasing Reynolds number, surface roughness, and stronger forced convection shifted the onset of laminar–turbulent transition. This shift significantly modified the local heat transfer distribution, with substantial enhancement observed in regions where buoyancy effects remained influential. This study provides valuable insights into the mechanisms governing heat transfer development in rough horizontal tubes under transitional flow conditions. These outcomes support improved design and optimisation strategies for compact heat exchangers and thermal management systems aimed at achieving higher energy efficiency and operational performance.

Keywords: *Surface roughness, Heat transfer, Transition flow, Constant heat flux, critical Reynolds number, heat exchanger*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

SIMULATION AND THERMODYNAMIC ANALYSIS OF A COUPLED AIR COMPRESSION–VAPOR COMPRESSION REFRIGERATION SYSTEM USING ECO-FRIENDLY REFRIGERANT R1234YF WITHOUT AIR HEATER CONDITIONS IN ECOSIMPRO/PROOSIS

Neeraj Kumar Sharma[1], Chennu Ranganayakulu[1] and Suvanjan Bhattachryya[1]

[1]Department of Mechanical Engineering, Birla Institute of Technology and Science Pilani, Pilani Campus, Vidya Vihar, Pilani 333031, Rajasthan, India.
p20230068@pilani.bits-pilani.ac.in, c.ranganayakulu@pilani.bits-pilani.ac.in, suvanjan.bhattacharyya@pilani.bits-pilani.ac.in

ABSTRACT

This study presents the modeling and simulation of a coupled air compression and vapor compression refrigeration (VCR) system employing the eco-friendly refrigerant R1234yf, using the EcosimPro/PROOSIS simulation platform. The developed schematic integrates a closed-loop refrigeration cycle without air heater loop thermally coupled through heat exchangers, enabling analysis of thermodynamic behavior and energy exchange under zero-heater-power conditions. The refrigeration subsystem, consisting of a compressor, condenser, expansion valve, and evaporator, operates within an evaporator temperature range of 263–273 K, a condenser temperature of 313 K, and a degree of superheating varying from 0 to 5 K. The air loop, modeled as a Brayton-type flow path, provides dynamic boundary conditions for studying the stability and performance of the coupled system. Simulation results reveal that the Coefficient of Performance (COP) increases with both evaporation temperature and compressor efficiency. For a fixed condenser temperature of 313 K, the COP improves significantly from approximately 2.5 to above 5.0 as compressor efficiency rises from 0.6 to 0.9 and evaporation temperature approaches 273 K. These findings confirm that optimizing compressor performance and operating temperature levels can substantially enhance system efficiency. The use of R1234yf, a low-GWP refrigerant, ensures reduced environmental impact while maintaining satisfactory thermodynamic performance. Overall, the study validates EcosimPro/PROOSIS as an effective tool for modeling hybrid, environmentally sustainable thermodynamic systems. The developed model provides a foundation for further research on energy-efficient cooling, waste heat recovery, and integrated cooling–power generation applications.

Key Words: *COP, Compressor Power, GWP, ODP, Waste heat recovery*

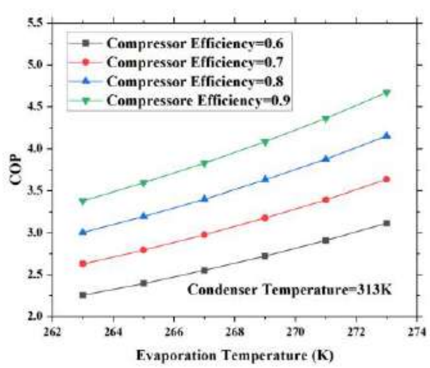


Figure 1: Effect of evaporation temperature on COP

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

EXPERIMENTAL AND ANALYTICAL EVALUATION OF A SOLAR-ASSISTED MILK COOLING SYSTEM EMPLOYING DUAL ECO-REFRIGERANTS R1234YF AND CO_2 FOR ENHANCED ENERGY EFFICIENCY

Neeraj Kumar Sharma[1] and Suvanjan Bhattachryya[1]

[1]Department of Mechanical Engineering, Birla Institute of Technology and Science Pilani, Pilani Campus, Vidya Vihar, Pilani 333031, Rajasthan, India.
Email: p20230068@pilani.bits-pilani.ac.in, suvanjan.bhattacharyya@pilani.bits-pilani.ac.in

ABSTRACT

This study presents a comparative performance and energy analysis of a solar-powered milk-cooled chiller operating on a vapor compression refrigeration (VCR) cycle. The system utilizes two eco-friendly refrigerants R1234yf, a low-GWP hydrofluoro-olefin, and CO_2 (R744), a natural refrigerant to evaluate their suitability for sustainable milk cooling applications. The analysis was performed under an evaporator temperature range of 263–273 K and a condenser temperature of 313 K, considering both superheated and non-superheated conditions. Solar energy is employed to drive the compressor, ensuring energy-efficient and off-grid operation for rural environments. The results reveal that solar power consumption decreases with increasing evaporator temperature for both refrigerants, while the coefficient of performance (COP) increases correspondingly. Among the two, R1234yf consistently demonstrates superior performance, with lower solar power consumption (7.68–12.42 kW) and higher COP values (ranging from approximately 3.36 to 5.43) compared to CO_2, which exhibits higher power consumption (13.42–20.71 kW) and lower COP values (2.01 to 3.11) across the same temperature range. These differences are primarily attributed to the higher operating pressure and compressor work associated with CO_2. The integration of a heat exchanger further enhances the refrigeration effect and reduces the compressor's workload, thereby improving overall efficiency. In conclusion, the comparative results confirm that the solar-assisted VCR system using R1234yf and CO_2 offers an environmentally friendly and energy-efficient solution for milk cooling, with R1234yf demonstrating superior thermodynamic and energy performance, while CO_2 remains a viable natural refrigerant alternative under sustainable operating conditions.

Key Words: *Chiller,Refrigeration effect, COP, ODP, Global warming potential*

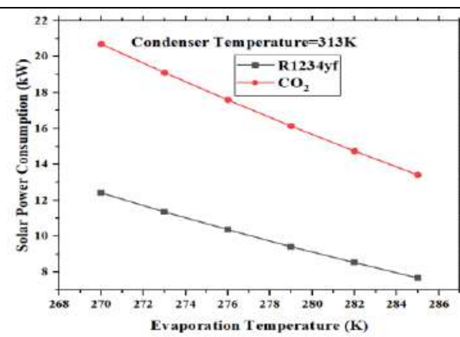


Figure 1: Effect of evaporation temperature on solar power consumption

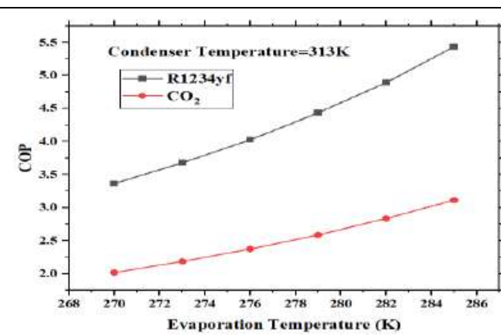


Figure 2: Effect of evaporation temperature on COP

PERFORMANCE EVALUATION OF COW DUNG–BASED COMPOSITE BIOMASS BRIQUETTES FOR THERMAL ENERGY APPLICATIONS

Naveen Kumar Sain[1], Chandan Kumar[2], and Ashish Nayyar[2]

[1]Department of Mechanical Engineering, Birla Institute of Technology and Science, Pilani, Pilani, India-333031, P20242202@pilani.bits-pilani.ac.in,
[2]Swami Keshvanand Institute of Technology, Management & Gramothan, Jaipur, India-302017, chandan.kumar@skit.ac.in, ashish.nayyar@skit.ac.in

ABSTRACT

Cow dung–based briquettes are promising for decentralized thermal energy, but their adoption is limited by poor handling strength and process challenges associated with high-moisture feedstocks. This study's original contribution is the design of a small-scale composite briquetting system specifically configured for cow dung composites, integrating moisture-drainage provision, a conical hopper with vertical spiral agitator for uniform mixing, and interchangeable dies (cylindrical/rectangular/triangular) followed by quantitative performance benchmarking of multiple waste-derived blends. The developed machine uses a 5 hp drive with practical attachments (including venturi length variants of 6 in and 7.5–8 in) to support stable feeding and compaction of sticky biomass mixtures. Composite briquettes were produced from cow dung blended with coal powder (lignite), wood dust, dry leaves, and crop residues, and evaluated for moisture content, calorific value, and compressive strength. Measured moisture content across representative samples ranged from 44.3% to 60.7%, reflecting the inherent wet nature of dung-based feedstocks. Despite this, the briquettes delivered net energy performance in the range of 15.09–15.64 MJ/kg, with the cow dung–wood dust blend achieving the highest calorific value (15.638 MJ/kg), followed by cow dung + coal powder (15.472 MJ/kg) and cow dung + dry leaves (15.272 MJ/kg). Mechanical integrity was demonstrated by a cow dung–wood briquette of size 2.5 × 2 × 2 in, which withstood a compressive load of 2.4 kN, indicating adequate strength for handling and transport. The system achieved a production rate of ~250 kg/h, and open-sun drying required 6–7 days under typical conditions. The results demonstrate that a purpose-built briquetting setup can convert locally available waste into composite briquettes with quantifiable strength and energy output, providing a low-cost solid fuel option for thermal applications while supporting residue management and rural value generation.

Key Words: *Biomass briquettes, agro-waste utilization, cow dung, calorific value, solid biofuel*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

EFFECT OF REYNOLDS NUMBER AND STREAM CONFIGURATION ON MIXING TIME IN A MULTI-INLET VORTEX REACTOR

Sharmi Sarkar[1], Mithun Das[2] and Nirmalendu Biswas[3]

[1]Department of Chemistry, Prasannadeb Women's College, Jalpaiguri 735101, India sharmisarkar333@gmail.com
[2]School of Nuclear Studies and Application, Jadavpur University, Kolkata 700106, India mdas190@gmail.com
[3]Department of Power Engineering, Jadavpur University, Kolkata 700106, India biswas.nirmalendu@gmail.com

ABSTRACT

Uniformly sized functional nanoparticles are important in pharmaceutical and agricultural products, but producing them efficiently is still a challenge. The multi-inlet vortex reactor (MIVR) is a promising option because it can create very fast mixing, which is required for flash nanoprecipitation. This work examines how the mixing time in the MIVR depends on Reynolds number, inlet velocity, fluid properties and reactor geometry. Reynolds numbers from 500 to 2000 are studied for a fixed chamber diameter. All inlets supply steady, uniform streams. Two opposing inlets deliver an acid solution (A), while the other two supply a base solution (B) that carries the chemical reagent (D) as shown in Figure 1. The study shows that mixing performance changes noticeably with flow conditions. At a Reynolds number of 1000, the mixing effectiveness is about 0.78, while at 2000 it drops to around 0.35 after 0.1s, indicating stronger segregation at high flow rates. The detailed analysis and supporting data will be presented in the full paper.

Key Words*:* *multi-inlet vortex mixer, microreactor, CFD, Precipitation, Chemical reagent transport*

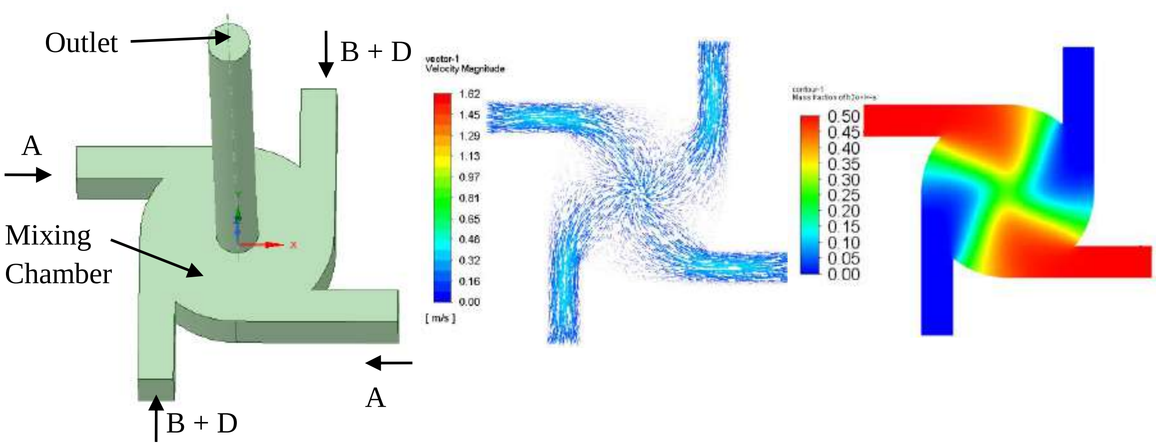


Figure 1: Schematic of the reactor geometry with mid-plane velocity distribution and reagent concentration at Re = 2000

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

PAYLOAD-DEPENDENT THERMAL ENDURANCE OF PCM COLD BOXES: AN EXPERIMENTAL-COMPUTATIONAL FRAMEWORK

Tapasvi Bhatt[1], Hamid Saeedipour[2] and Eddie Ng Yin Kwee[1]

[1]School of Mechanical and Aerospace Engineering, Nanyang Technological University (NTU), 50 Nanyang Avenue, Singapore, 639798, tapasvi001@e.ntu.edu.sg, mykng@ntu.edu.sg
[2]School of Engineering, Republic Polytechnic (RP), 9 Woodlands Avenue 9, Singapore, 738964, hamid_saeedipour@rp.edu.sg

ABSTRACT

Reliable passive cooling for perishable commodities remains a critical challenge in last-mile cold chain logistics, particularly in tropical regions where ambient temperatures and humidity accelerate thermal ingress. This study investigates how biological payloads, specifically strawberries and apples affect the thermal endurance of an unpowered PCM-based cold box, and develops a numerical PCM melting model in ANSYS Fluent to predict the internal temperature evolution with high fidelity. While conventional passive boxes maintain effective cooling for only 3–4 days, our combined experimental–computational framework demonstrates sustained sub-0 °C conditions for 6–7 days depending on payload characteristics and internal insulation configuration. Experiments were conducted under controlled tropical profiles (32–34 °C; 70–85% RH) for 168 h. Payloads with higher moisture content and respiration heat influenced thermal gain differently: strawberry-loaded boxes maintained 0°C at 103 h and rose gradually to 0.5°C at 168 h, while apple-loaded boxes exhibited slower heat gain, stabilising at 0.2°C after 168 h. In both cases, the internal relative humidity remained consistently high (92.4–95.1%), partially buffering thermal fluctuations. Compared to an empty-box baseline, the presence of biological payloads reduced temperature stability by 14–22 h due to their latent moisture buffering effect.

Key Words: Cold chain logistics, Heat Transfer, Cold Box, Strawberries.

Figure 1: Experimental Setup and numerical model development.

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

CURRICULUM-DRIVEN GRAPH U-NET FRAMEWORK FOR MHD NANOFLUID CONVECTION IN COMPLEX ENCLOSURES

Shubham Das[1], Sandip Sarkar[1], Nirmalendu Biswas[2*], Dipak Kumar Mandal[3], Nirmal K. Manna[1]

[1]Department of Mechanical Engineering, Jadavpur University, Kolkata 700032, India
[2]Department of Power Engineering, Jadavpur University, Kolkata 700106, India
[3]Department of Mechanical Engineering, Government Engineering College Samastipur, Bihar, India
[*]Corresponding author email id: *biswas.nirmalendu@gmail.com*

ABSTRACT

The advancement of modern thermal management technologies, spanning systems designed for electronic component cooling to those employed in nuclear reactor thermal regulation, increasingly depends on the combined influence of engineered nanofluids and externally controlled magnetic fields. Within this context, Magnetohydrodynamic (MHD) natural convection has been the subject of substantial investigation through classical Grid-Based numerical methodologies, including but not limited to Finite Element Analysis (FEM). Although such techniques remain foundational within computational heat transfer, they often encounter notable performance challenges when applied to domains characterized by non-rectangular or otherwise intricate geometrical features. These challenges typically manifest as elevated computational costs, difficulties associated with generating high-quality meshes, and reduced robustness when resolving sharply varying physical fields. Physics-Informed Neural Networks (PINNs) have emerged as a promising class of mesh-independent approaches capable of directly embedding governing equations into the training objective. Nevertheless, standard PINN architectures built upon conventional Multi-Layer Perceptrons (MLPs) frequently experience limitations in accurately approximating steep gradients within irregular regions. Moreover, they are susceptible to convergence breakdowns when confronted with tightly coupled multi-physics scenarios, especially under conditions corresponding to high Rayleigh numbers, where flow behaviour becomes strongly convection-driven. To address these limitations, the present study develops a Deep Learning framework specifically designed to compute the coupled Navier–Stokes and Energy equations for a CuO–water nanofluid contained within a complex circular–trapezoidal enclosure. Departing from common PINN formulations that depend on global coordinate mappings, the proposed method employs a specialized Graph U-Net architecture. This architecture interprets the computational domain as an unstructured graph, thereby enabling Graph Neural Network (GNN) mechanisms to inherently accommodate irregular boundaries without requiring spatial coordinate transformations. Within the encoder, Graph Attention Layers are utilized to capture fine-scale geometric and local physical interactions, while the decoder employs Graph Convolutional Layers to reconstruct coherent global solution fields. To overcome the spectral bias and initialization sensitivities typically observed in PINN training at high Rayleigh numbers, a Curriculum Learning strategy is implemented. The model is initially trained on conduction-dominated regimes and subsequently guided toward convection-dominated configurations through progressive refinement. In parallel, boundary conditions are enforced using a discrete masking mechanism, ensuring strict boundary adherence throughout the entire training process rather than relying on soft-penalty formulations. Furthermore, adaptive loss weighting is incorporated to appropriately balance the contributions of the fluid dynamic and thermal energy equations. The resulting framework integrates geometry-independent graph processing, physics-guided learning, and explicitly enforced boundary treatment to produce a stable and scalable methodology for simulating complex multi-physics environments.

Key words: *Physics-informed neural network (PINN), Graph neural network (GNN), Magnetohydrodynamics (MHD), Nanofluid, Curriculum learning.*

SUSTAINABLE BIODIESEL FROM WASTE ANIMAL TALLOW: PRODUCTION, ENGINE PERFORMANCE, EMISSION CHARACTERISTICS, AND ECONOMIC ASSESSMENT

Deepak Tanwar[1*], Kunj Bihari Rana[1], Chandan Kumar[2]

[1]Department of Renewable Energy, Rajasthan Technical University, Kota–324010, India.
[2]Mechanical Engineering Department, Swami Keshvanand Institute of Technology, Management & Gramothan, Jaipur–302017, India.
[*]Corresponding author email– deepakrspcb@gmail.com

ABSTRACT

The growing global energy demand and depletion of fossil fuel reserves have intensified the search for sustainable and renewable energy sources. Biodiesel has emerged as a promising alternative due to its renewability, biodegradability, and lower environmental impact compared to conventional diesel. This study focuses on the production, performance, emission, and economic analysis of biodiesel derived from waste animal tallow, referred to as Transesterified Tallow Oil (TTO).The TTO biodiesel was synthesized through citric acid–catalyzed transesterification, converting tallow waste triglycerides into fatty acid ethyl esters (FAEEs). The fuel exhibited suitable physicochemical properties with improved cetane number, viscosity, and flash point, ensuring compatibility with diesel engines. Four fuel blends—D–TTO5, D–TTO10, D–TTO15, and D–TTO20—were tested on a single-cylinder, four-stroke, water-cooled CI engine under varying loads.The production cost of TTO biodiesel was estimated. These results confirm that biodiesel from animal waste feedstock offers a sustainable, economical, and eco-friendly solution to meet future energy challenges while promoting waste valorization and emission reduction.

Key words: Animal Tallow; Transesterified Tallow Oil; CI Engine; Performance; Exhaust Emissions; Economic Analysis

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

PHYSICS-INFORMED NEURAL NETWORKS (PINNS) FOR DATA-EFFICIENT RECONSTRUCTION OF THERMOFLUID FLOW FIELDS IN CLASSICAL CAVITY SYSTEMS

Ayush Ganguly[1], Sandip Sarkar[2], Nirmalendu Biswas[3*], Dipak Kumar Mandal[4], Nirmal K. Manna[2]

[1]Department of Civil Engineering, Jadavpur University, Kolkata 700032, India
[2]Department of Mechanical Engineering, Jadavpur University, Kolkata 700032, India
[3]Department of Power Engineering, Jadavpur University, Kolkata 700106, India
[4]Department of Mechanical Engineering, Government Engineering College Samastipur, Bihar, India
[*]Corresponding author email id: biswas.nirmalendu@gmail.com

ABSTRACT

A Physics-Informed Neural Network (PINN) framework is employed for the simulation, reconstruction and analysis of three of the most classical fluid & thermofluid systems, namely the pure lid-driven cavity flow, natural convection in a differentially heated square cavity and a mixed convection combining the effect of lid motion and buoyancy. The approach is based on the ability of PINNs, which many times are often referred to as a substitute for Partial Differential Equations (PDEs), to work without or with very little labelled data. The governing physics of the system are directly embedded into the loss functions of the Neural Network, giving it the guidance in order to learn the effective physics of the system. The primary idea of the study is to obtain temperature and flow fields in the absence of dense training data. The case of pure cavity flow is completely reconstructed without any labelled data whatsoever, with the training solely guided by the PDE residue of the governing equation, that is, the 2D incompressible Navier Stokes Equations. The other two cases involving natural and mixed convections are where the sparse temperature measurements are provided at specific Rayleigh numbers (Ra). For the simulation of natural and mixed convection cases, highly sparse temperature observations across the domain are provided, obtained by downsampling temperature fields from numerical simulations. These sparse temperature samples are incorporated as soft constraints during training, along with the 2D Incompressible Navier Stokes Equations, the 2D heat transfer equation and Dirichlet and Neumann boundary conditions, while no other flow field information is supplied. The primary goal of this study is to evaluate the effectiveness of a PINN in capturing the multiphysics associated with the three different thermofluid systems from extremely sparse thermal measurements. Analyses are conducted by inducing noise in the temperature data and by evaluating the model's training and generalizability across different Ra (in the range of 10^3 to 10^6) for the natural and mixed convection cases. Across all three configurations, the capability of PINNs to reproduce complex thermofluid interactions is demonstrated. The results demonstrate the ability of PINNs to produce physically consistent solutions involving actual flow fields across different parameter settings without dense labelled datasets. The solutions highlight the robustness of the physicsguided learning when there is no data available or when there is very sparse, low-quality experimental data. The model is evaluated on the PDE and the temperature reconstruction residuals. Most importantly, the velocity and the temperature fields are evaluated based on numerical simulation of the same system. The outcomes of this study suggest that PINNs can serve as effective data-efficient solvers for such problems and may be further extrapolated to complex geometries and data-driven surrogate modellingms.

Key words: *Physics-informed neural networks, Thermofluid flow reconstruction, Sparse data simulation, Lid-driven cavity flow, Natural and mixed convection.*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

FLUID-STRUCTURE INTERACTION OF PULSATILE BLOOD FLOW IN ASYMMETRIC STENOSED ARTERIES

Sayandeep Sain[1], Anubhav Chowdhury[2], Nirmalendu Biswas[3*], Dipak Kumar Mandal[4] and Nirmal K. Manna[5]

[1]Department of Chemical Engineering, Jadavpur University, Kolkata 700032, India
[2]Department of Civil Engineering, Jadavpur University, Kolkata 700032, India
[3]Department of Power Engineering, Jadavpur University, Kolkata 700106, India
[4]Department of Mechanical Eng., Government Eng. College Samastipur, Bihar, India
[5]Department of Mechanical Engineering, Jadavpur University, Kolkata 700032, India
[*]corresponding author email id: *biswas.nirmalendu@gmail.com*

ABSTRACT

Arterial stenosis with asymmetric plaque formation and compliant hyperelastic walls creates complex fluid-structure interaction phenomena governing hemodynamic parameters critical for cardiovascular disease progression, thrombosis risk assessment, and intervention planning. This investigation examines pulsatile blood flow through asymmetrically stenosed arteries with deformable walls using coupled fluid-structure interaction analysis, systematically evaluating effects of arterial wall elasticity (E = 2.5–7.5 MPa), cardiac cycle duration, and stenosis severity on wall shear stress distribution, wall pressure patterns, and structural deformation characteristics. The computational domain represents a stenosed arterial segment with length L = 12R, diameter D = 2R = 3.4mm, and asymmetric stenotic constrictions positioned on upper and lower hyperelastic arterial walls [1]. Governing Navier-Stokes equations for incompressible non-newtonian blood flow are coupled with structural mechanics equations for hyperelastic wall deformation through arbitrary Lagrangian-Eulerian formulation, solving bidirectional fluid-solid coupling at deforming interfaces [2]. Hemodynamic analysis reveals wall shear stress distribution exhibiting peak magnitudes of 22–25 Pa at stenosis throat locations (x-coordinate = 11–13 mm) with maximum values increasing systematically from 22.5 Pa at E = 2.5 MPa to 25 Pa at E = 7.5 MPa, demonstrating that arterial stiffening intensifies shear stress concentration. Wall pressure distributions show maximum values of 100–113 Pa at stenosis entrance (x-coordinate ≈ 11 mm) with elevated stiffness producing 113 Pa compared to 100 Pa for compliant walls, indicating that reduced arterial compliance amplifies pressure loading during systolic phase [3]. Post-stenotic regions demonstrate dramatic pressure recovery to 35–40 Pa, creating significant adverse pressure gradients conducive to flow recirculation. Compliant arterial walls (E = 2.5 MPa) exhibit greater deformation reducing peak shear stresses through geometric accommodation, while stiffer arteries (E = 7.5 MPa) maintain rigid boundaries intensifying stress magnitudes. Results establish that arterial wall elasticity critically modulates hemodynamic stresses, with atherosclerotic stiffening exacerbating wall shear stress concentrations that promote endothelial dysfunction and plaque progression [4]. Findings guide clinical risk stratification for stenotic lesions by incorporating wall compliance measurements, inform computational planning for endovascular interventions considering patient-specific arterial stiffness, and support development of compliant stent designs minimizing flow disturbances through mechanical compliance matching. Fluid-structure interaction framework demonstrates that coupled analysis capturing wall deformability is essential for accurate hemodynamic prediction in compliant stenosed arteries subjected to pulsatile physiological loading conditions.

Key words: *Fluid-structure interaction, Pulsatile blood flow, Arterial stenosis, Hyperelastic walls, Wall shear stress, Hemodynamics, Arterial stiffness.*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

THERMO-MAGNETIC CONVECTION AND ENTROPY GENERATION IN T-SHAPED CAVITY

Sayantan De[1], Sayandeep Sain[2], Nirmalendu Biswas[1*], Dipak Kumar Mandal[3], Nirmal K. Manna[4] and Suvanjan Bhattacharyya[5]

[1]Department of Power Engineering, Jadavpur University, Kolkata 700106, India
[2]Department of Chemical Engineering, Jadavpur University, Kolkata 700032, India
[3]Department of Mechanical Eng., Government Eng. College Samastipur, Bihar, India
[4]Department of Mechanical Engineering, Jadavpur University, Kolkata 700032, India
[5]Department of Mechanical Eng., Birla Institute of Technology and Science Pilani, Rajasthan, India
[*]Corresponding author email id: biswas.nirmalendu@gmail.com

ABSTRACT

T-shaped cavities with perpendicular branching channels create distinctive convective flow structures and thermal transport characteristics relevant to electronic cooling systems, microfluidic mixers, and building ventilation applications. This investigation examines magnetohydrodynamic natural convection of CuO-water nanofluid in an inverted T-shaped enclosure subjected to differential heating using finite element methodology. The study systematically analyzes effects of Rayleigh number (Ra = 10^3–10^6), Hartmann number (Ha = 0–70), radiation parameter (Rd = 0–2), Darcy number (Da = 0.0001–0.01), and magnetic field inclination angle (λ = 0°–150°) on flow patterns, temperature distributions, heat transfer performance, and entropy generation characteristics. Governing equations incorporating buoyancy forces, Lorentz force effects, porous medium resistance via Darcy-Brinkman model, and thermal radiation through Rosseland approximation are discretized using Galerkin weighted residual technique [1]. CuO nanoparticles at 2% volume fraction enhance base fluid thermophysical properties following validated correlations. The inverted T-geometry produces flow bifurcation at the junction with primary circulation cells developing in both horizontal base and vertical stem sections, creating complex multi-cellular flow structures where perpendicular flows interact [2]. Flow analysis reveals maximum velocity magnitude reaching 41.2 at Ra = 10^6 with Ha = 30, λ = 90°, and Da = 0.01, indicating strong buoyancy-driven convection modulated by magnetic damping and permeability effects. Heat transfer quantification yields Nusselt number Nu = 4.29 at reference conditions, reflecting geometric influence on convective transport through the branched configuration. Streamline topology (ψ = -3.09 to 3.09) exhibits symmetric dual vortices in horizontal arms with vertical stem circulation, demonstrating flow partition at the T-junction where horizontal and vertical branches meet. Heat line visualization (Π = -0.657 to 2.44) illustrates energy pathways through perpendicular branches with thermal flux redistribution occurring at junction regions. Viscous irreversibility (NS_{Vmax} = 1.39E-7) concentrates at junction corners and branch entrances where flow redirection creates elevated velocity gradients [3]. Parametric investigation confirms inverted T-geometry fundamentally alters convective patterns through branch coupling effects, with circulation strength controlled by buoyancy while magnetic fields provide flow regulation capability. Results guide thermal management strategies for T-shaped heat sinks dissipating thermal loads through branched pathways, microfluidic devices employing T-junctions for stream mixing with magnetic field control, and HVAC systems using T-shaped ducts for optimized air distribution.

Key words: *Natural convection, MHD, Nanofluid, Inverted T-cavity, Porous-media, Thermal radiation, Entropy generation, Finite element method.*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

THERMO-MAGNETIC CONVECTION OF NANOFLUID IN AN ASTROID-SHAPED POROUS ENCLOSURE WITH THERMAL RADIATION

Soham Bhattacharya[1], Sayandeep Sain[2], Nirmalendu Biswas[1*], Dipak Kumar Mandal[3], Nirmal K. Manna[4] and Suvanjan Bhattacharyya[5]

[1]Department of Power Engineering, Jadavpur University, Kolkata 700106, India
[2]Department of Chemical Engineering, Jadavpur University, Kolkata 700032, India
[3]Department of Mechanical Eng., Government Eng. College Samastipur, Bihar, India
[4]Department of Mechanical Engineering, Jadavpur University, Kolkata 700032, India
[5]Department of Mechanical Eng., Birla Institute of Technology and Science Pilani, Rajasthan, India
[*]corresponding author email id: biswas.nirmalendu@gmail.com

ABSTRACT

Astroid-shaped enclosures with four-cusped stellate geometry generate distinctive convective flow patterns and thermal transport mechanisms compared to conventional polygonal cavities, relevant for advanced heat exchangers, solar thermal collectors, and microelectronic cooling systems. This investigation examines thermo-magnetic convection of CuO-water nanofluid in an astroid-shaped porous enclosure subjected to differential heating and thermal radiation using Galerkin finite element technique. The work systematically evaluates effects of Rayleigh number (Ra = 10^3–10^6), Hartmann number (Ha = 0–70), radiation parameter (Rd = 0–2), Darcy number (Da = 0.001–0.1), and magnetic field inclination angle (λ = 30°–150°) on velocity distribution, temperature fields, streamline topology, heat transfer performance, and entropy generation characteristics. Governing equations incorporating Lorentz forces, porous resistance through Darcy-Brinkman formulation, and radiative heat flux via Rosseland approximation are solved numerically with CuO nanoparticles at 2% volume concentration modifying thermophysical properties according to established mixture theory [1]. The astroid configuration produces quadrant flow cells with primary circulation in each stellate lobe and secondary vortices near cusp regions where geometric singularities occur. Flow analysis indicates maximum velocity magnitude reaching 163 at Ra = 10^6 with Ha = 30, λ = 90°, and Da = 0.01, demonstrating buoyancy-driven convection intensified by permeability effects and controlled through magnetic damping [2]. Heat transfer quantification yields Nusselt number Nu = 13.0 under reference conditions, reflecting geometric influence on convective transport through the multi-lobed structure. Streamline patterns (ψ = -14.7 to 14.7) reveal symmetric tetrad circulation with flow acceleration along cusp walls and stagnation zones at lobe centers where perpendicular flows interact. Heatline distributions (Π = -8.73 to 6.04) demonstrate energy pathways converging toward cusps and diverging through lobes toward isothermal boundaries, with thermal flux channeling modified by cavity morphology. Viscous irreversibility (NSv_{max} = 8.22E-7) concentrates at cusp vertices and lobe-wall intersections where velocity gradients intensify due to sharp geometric transitions and flow redirection [3]. Parametric examination confirms astroid geometry fundamentally alters convective behavior through cusp-lobe interactions, with enhanced buoyancy strengthening multi-cellular circulation while magnetic fields provide directional damping capability dependent on field orientation. Elevated radiation parameters supplement conductive-convective transport at higher Rd values, particularly in optically participating nanofluids. Results guide design optimization for star-shaped heat sinks dissipating thermal loads through multiple lobed pathways, microfluidic mixers employing astroid channels for multi-stream merging with controllable magnetic stirring, porous thermal energy storage units utilizing stellate geometries for increased surface contact, and architectural ventilation systems incorporating star-shaped ducts for enhanced air circulation. Comprehensive analysis establishes that astroid enclosures create complex convective structures through cusp-induced flow focusing and lobe-generated recirculation zones, with irreversibility generation peaking at geometric singularities where both viscous and thermal entropy production mechanisms reach critical values under combined buoyancy, magnetic, and porous resistance effects.

Key words: *Thermo-magnetic convection, Nanofluid, Astroid cavity, Porous media, Thermal radiation, Entropy generation, Finite element method.*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

EFFECT OF TEMPERATURE ON PRESSURE DROP IN LAMINAR AND TURBULENT PIPE FLOW: A COMPREHENSIVE ANALYTICAL, NUMERICAL, AND COMPUTATIONAL STUDY

Soham Vaishampayan[1], Bhavik Yelhekar[2], Neal Asalkar[3], Manish Bagul[4], Vyankatesh Narhare[5], and Nitin Borse[6]

[1–6] Vishwakarma Institute of Technology, 666 Kapil Nagar, Kondhwa Budruk, Pune, Maharashtra 411048.
soham.vaishampayan24@vit.edu

ABSTRACT

This study investigates the effect of temperature on pressure drop in laminar and turbulent flow regimes for water flowing through a circular pipe using a novel three-method validation approach. Temperature variations significantly alter fluid properties, particularly dynamic viscosity (68% reduction) and density (4% reduction) between 25°C and 100°C, which directly influence the Reynolds number and pressure losses. A test pipe (L = 1 m, D = 0.02 m) was analyzed for laminar flow (Re = 224 at 25°C, Re = 684 at 100°C) and turbulent flow (Re = 44,809 at 25°C, Re = 136,857 at 100°C). Analytical calculations employed Hagen-Poiseuille equation for laminar flow and DarcyWeisbach equation with Blasius correlation for turbulent flow. MATLAB simulations performed temperature sweeps with 0.1°C resolution using validated property correlations. ParaView provided computational visualization of analytical velocity and pressure fields. Laminar flow pressure drop decreased from 0.712 Pa to 0.224 Pa (68.5% reduction) while turbulent flow decreased from 2168.16 Pa to 1575.92 Pa (27.3% reduction) across the 25-100°C range. Reynolds number increased approximately threefold in both regimes. All three methods demonstrated excellent agreement (<0.5% deviation), confirming that viscosity-dominated laminar flows exhibit higher temperature sensitivity compared to inertia-dominated turbulent flows. Originality: Unlike previous studies examining temperature effects at discrete points, this work integrates exact analytical solutions with high-resolution continuous numerical analysis (751 temperature points) and computational visualization. The methodology uses accessible tools (MATLAB, open-source ParaView) without requiring expensive CFD licenses, enabling reproducibility and straightforward extension to other fluids and similar operating conditions.

Key Words: *Reynolds number, Hagen-Poiseuille equation, Darcy-Weisbach equation, MATLAB simulation*

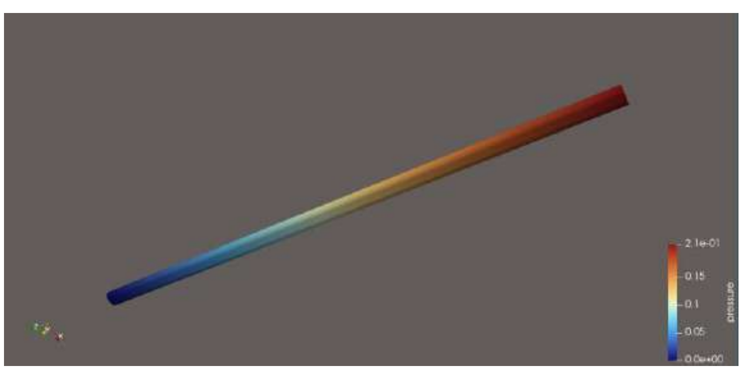


Figure 1: Pressure drop visualization in laminar 100 degrees celsius case

EFFECT OF DIVERSE INSERTS AND NANO FLUID ON THERMAL PERFORMANCE OF CONCENTRIC TUBE HEAT EXCHANGER HAVING SPIRAL TUBE

[1]Shivasheesh Kaushik and [2]Satyendra Singh

[1]Department of Mechanical Engineering, Shivalik College of Engineering, Dehradun, Uttarakhand, India, skaushik@sce.org.in

[2]Department of Mechanical Engineering, B.T.K.I.T., Dwarahat, Uttarakhand, India, ssinghiitd@gmail.com

ABSTRACT

A comparative study is presented to acknowledge the combined effect of diverse geometry of insert and fluid domain used in concentric tube exchanger (CTE). The combination of newly developed stable hybrid nanofluid with geometrical inserts enhances the overall surface area which provides a noble feature to this study compared with existing literature. Here, helical-shape, cubical-shape, cuboidal-shape, cylindrical-shape, conical-shape, and hemispherical-shape insert designs were considered for analysis. These were equipped over spiral tubes to analyse the thermal characteristics and its performance with diverse nanofluid. These fluids are developed through utrasonication technique and the combination strategy of diverse nanoparticle integration such as ZnO with Al_2O_3, and Al2O3 with CuO etc. at a size range of 10 nm and 0.05% volumetric fraction. The exchanger performance was examined under laminar-turbulent region for counter fluid flow situation. During experimentation the hot working fluid (H2O) moves in upper side of the exchanger shell. It is kept constant at 333 K with 4.02 L/min, whereas, the cold nanofluid is flow at ambient temperature of 304 K with a range of 0.72–2.95 L/min. The cold nanofluid moves inside the spiral tube. The geometrical and fluid domain were assessed for their impact on thermal performance, friction factor, Nusselt number, and effectiveness. The results reveals that the helical-shape and cubical-shape insert provides superior performance, it improves the thermal transfer rate by 38.5%, 25.2%, compared to the other inserts by 21.5%, 18.5%, 15.5% and 11.5%, respectively. Further, investigation has explored for nanofluids to examine the effect of these fluids inside the exchangers, installed with superior helical-shape insert. Al_2O_3 integrated CuO nanofluid presented the high and optimum thermal performance compared with other fluids..

Key Words: *Concentric Tube Exchanger (CTE), Geometrical Domain, Fluid Domain, Thermal Behaviour, Exchanger Performance..*

MHD THERMAL CONVECTION AND ENTROPY GENERATION IN A NANOFLUID-FILLED CURVILINEAR CAVITY WITH CENTRAL COLD OBSTRUCTION UNDER THERMAL RADIATION

Swagato Dhara[1], Sayandeep Sain[2], Nirmalendu Biswas[1*], Dipak Kumar Mandal[3], Nirmal K. Manna[4] and Suvanjan Bhattacharyya[5]

[1]Department of Power Engineering, Jadavpur University, Kolkata 700106, India
[2]Department of Chemical Engineering, Jadavpur University, Kolkata 700032, India
[3]Department of Mechanical Eng., Government Eng. College Samastipur, Bihar, India
[4]Department of Mechanical Engineering, Jadavpur University, Kolkata 700032, India
[5]Department of Mechanical Eng., Birla Institute of Technology and Science Pilani, Rajasthan, India
[*]corresponding author email id: biswas.nirmalendu@gmail.com

ABSTRACT

Square cavities with centrally positioned hourglass-shaped obstructions exhibit unique convective and thermal transport characteristics critical for heat exchanger design, microfluidic devices, and electronic cooling applications. This study investigates magnetohydrodynamic natural convection of CuO-water nanofluid in a differentially heated square enclosure containing a central constriction under thermal radiation effects using finite element methodology. The research systematically analyzes effects of Rayleigh number ($Ra = 10^3$-10^6), Hartmann number (Ha = 0-70), radiation parameter (Rd = 0-2) and Darcy number (Da = 0.001–0.1) on hydrodynamic behavior, isothermal patterns, energy transport mechanisms, and irreversibility characteristics. The hourglass geometry creates flow recirculation with dual symmetric vortices in upper and lower chambers separated by the constricted throat region [1]. Flow analysis reveals maximum velocity magnitude reaching 167 at $Ra = 10^6$ with Ha = 30 and $\lambda = 90°$, indicating strong buoyancy-driven circulation modulated by magnetic damping. Thermal performance evaluation yields Nusselt number Nu = 15.8 under reference conditions, demonstrating enhanced convective transport through the narrowed passage [2]. Streamline topology (ψ = -12.8 to 12.8) exhibits symmetric circulation cells with flow acceleration through the constriction zone influencing global heat redistribution. Heatline visualization (Π = -7.51 to 4.63) reveals energy pathways bifurcating around the central obstruction with thermal flux channeling through lateral regions toward cold boundaries. Viscous irreversibility (NSv_{max} = 2.16E-6) localizes at constriction walls and upper/lower chamber corners where velocity gradients intensify due to geometric restrictions and flow reversal. Parametric investigation confirms hourglass configuration significantly modifies convective patterns through geometric confinement effects, with circulation strength controlled by buoyancy forces while magnetic fields provide flow regulation capability. Results inform thermal management strategies for compact heat sinks utilizing constricted passages, microreactor designs employing hour-glass channels for enhanced mixing with magnetic field control, biomedical devices incorporating constricted flow paths for particle separation, and HVAC systems using shaped obstacles for flow distribution optimization. The study establishes that hourglass obstructions fundamentally alter convective transport through flow bifurcation and recombination mechanisms, with entropy generation concentrated at geometric transitions where irreversibilities from viscous dissipation and thermal gradients reach maximum values.

Keywords: *Natural convection, MHD, Nanofluid, Hourglass cavity, Thermal radiation, Entropy generation, Finite element method.*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

MATHEMATICAL MODELING OF ROLLING CONTACT BEARINGS UNDER DIFFERENT STAGES OF LUBRICATION

Kunwar Chandra Singh[1], Dr. Arun Kumar Jalan[2],Dr. Reza N. Jazar[3] and Dr. Sharad Shrivastava[4]

[1]Research Scholar, Department of Mechanical Engineering, BITS PILANI, Pilani campus 333031,
p20240800@pilani.bits-pilani.ac.in
[2]Associate Professor, Department of Mechanical Engineering, BITS PILANI, Pilani campus 333031,
arunjalan@pilani.bits-pilani.ac.in
[3]Professor, Department of Mechanical Engineering, RMIT Melbourne, Australia
reza.nakahiejazar@rmit.edu.au
[4]Associate Professor, Department of Mechanical Engineering, BITS PILANI, Pilani campus 333031,
sharad_shrivastava@pilani.bits-pilani.ac.in

ABSTRACT

A bearing is an evident element that carries the machine load and is a critical element in ensuring safe and efficient operation. A rolling contact bearing (RCB) exhibit a complex vibration system as its major components including rolling elements, inner raceway, outer raceway and cage interact to generate complex vibration signals.Vibration analysis and acoustics emissions (AE) are widely used to plan and create the predictive maintenance schedules and envisage the catastrophic faults. Conventional conditional monitoring studies illustrates a dominant diagnostic insights but voids robustness under complex vicinity scenarios often encountered in rolling contact bearings.To address the diagnostic void, a digital twin framework for rolling contact bearing is proposed that alludes physics based models to the data driven approach that exemplifies intelligent prognostics and health management (PHM).

The proposed framework demonstrates the transformation paradigm in condition monitoring and the interconnectivity between the three layers of the system namely sensors inculcated data acquisition unit (DAQ) with physical system to capture vibration signals under different load and speed conditions, a virtual domain of mathematical modelling for non linear vibration simulation that illustrates stiffness variation ,lubricant regimes and friction force effects, Archard's wear mechanics, Hertz contact mechanics considering contact area as circle and semi ellipsoid and the analytical layer which inculcates deep learning algorithms like neural networks and variational auto-encoders (VAE) that provides feature extraction. A Bi direction communication path that provides fidelity between physical and virtual domain.

This study contributes to the framework that illustrates the wear initiation and propogation for the bearing using wear cage and a color code mechanism that provides visual insights for the life cycle management and optimization.The study further offers a scalable and adaptive approach for planning maintenance schedules of the assets.

Key Words: *Digital Twin, Vibration, Wear, Physics based models.*

EFFECT OF MULTI CHANNELS MICROCHANNEL EQUIPPED PERIPHERAL RECTANGULAR INSERTS (PRI) AND HYBRID NANOFLUID ON THERMAL PERFORMANCE CHARACTERISTICS OF CONCENTRIC TUBE MICROCHANNEL EXCHANGER

Shivasheesh Kaushik[1]*, Vinay Sati[2], Sohan Lal Sharma[3]*, Satyendra Singh[4], Shailesh Kumar Rajan[5], Anil Kumar Chhotu[6], Manish Kumar[7], Rahul Kumar[8], Dinesh Kumar Rao[9], S Rahul Bharath[2], Vishnu R Nair[2], Ashwarya Raj Paul[10], Samriddhi Vashisth[1], Shubham Singh Karki[1]

[*1] Department of Mechanical Engineering, Shivalik College of Engineering, Dehradun, India
[2] Department of Mechanical Engineering, B.I.T.S. Pilani, Rajasthan, India
[*3] Department of Mechanical Engineering, THDC, IHET, Tehri, 249124, U.K., India
[4] Department of Mechanical Engineering, BTKIT, Dwarahat, India
[5] Department of Mechanical Engineering, Motihari College of Engineering, Motihari, India
[6] Department of Civil Engineering, Motihari College of Engineering, Motihari, India
[7] Department of Mechanical Engineering, Bakhtiyarpur College of Engineering, Bakhtiyarpur, India
[8] Department of Mechanical Engineering, Government Engineering College, Jamui, India
[9] Department of Mechanical Engineering, Institute of Engineering and Technology, Dr.Ram Manohar Lohia Avadh University Faizabad, Ayodhya, Uttar Pradesh, India
[10] Department of Mechanical Engineering, V.I.T. Vellore, India
*Corresponding Author Email ID: skaushik@sce.org.in, sohansh@nith.ac.in

ABSTRACT

Elevated heat generation is a result of the high performance of electronic components. When it comes to boosting efficiency and maintaining stability, heat dissipation becomes a major problem. This work experimentally investigates the performance (ηtpf, ε, N.T.U), fluid-flow (Nu, mfr), thermal (Q, U) attributes, and economical aspect (f, ΔP, pumping power, EC cost) of hybrid nano fluid composition (Al_2O_3+ZnO+D_2O and CuO+ZnO+D_2O) flowing inside Concentric Tube Microchannel Exchanger (CTMCE) with Peripheral Rectangular Inserts (PRI). The objective of this research is to develop a nanofluid-based microchannel cooling system with varying channel capacities (6, 12, and 18) to effectively handle the thermal transfer problems that arise in various kinds of electronic devices. Under steady state, counter-flow conditions, 303K and 323K were the operating temperatures of the hybrid nanofluid and hot D_2O, and their flow rate range of 9-375 ml/min and 46 ml/min respectively, during the experiments. A mixture of 0.5 ml of CTAB surfactant and various hybrid nanofluid compositions containing 10–20 nm-sized nanoparticles at 0.01% and 0.03% were used in the cooling system. In this study, three distinct techniques CFD, Regression, and ANN, are used to confirm the various features through experimental investigation. According to the results, CuO+ZnO+D_2O at 0.03% volume fraction exhibits superior thermal behavior, fluid flow, and performance at the turbulent zone; nevertheless, it is more expensive than Al_2O_3+ ZnO+D_2O at 0.01 and 0.03 % volume fraction from an economic standpoint. CuO+ZnO+D_2O hybrid nanofluid with a smaller range of operating parameters worked effectively beneath the laminar zone, both technically and economically.

Key Words: *CRBMC, ERI, Hybrid Nano-Fluid, Electronic Deception, Regression, ANN*

BIO-INSPIRED LEADING-EDGE TUBERCLES FOR SPORTS APPLICATIONS: A COMPREHENSIVE REVIEW

Ankitha Gupta Surisetty[1], Navya Vijay[2] and Shashank Khurana[3]

[1,2,3] Department of Mechanical Engineering, Birla Institute of Technology and Science, Pilani, Dubai Campus, Dubai International Academic City, Dubai, United Arab Emirates,

Email: f20230333@dubai.bits-pilani.ac.in, f20230250@dubai.bits-pilani.ac.in and skhurana@dubai.bits-pilani.ac.in

ABSTRACT

Tubercles are rounded protrusions along the leading edges of humpback whales flippers where tubercles play a key role in passive flow control. These protrusions generate consistent streamwise vortices that energize the boundary layer, which helps the flow overcome adverse pressure gradients. Tubercles can delay separation which reduces stall, increases lift and reduces drag in post-stall conditions. Reported stall-delay angles range from approximately 4° to 6° compared to baseline smooth leading-edge configurations, with peak lift coefficients increasing by up to 16.7% in several experimental studies. Experimental and numerical studies report lift-to-drag ratio improvements of up to 10.9- 17.6% and drag reductions of approximately 30% in post-stall and bluff-body flow conditions, depending on tubercle geometry and operating angle of attack. Experimental and numerical studies on airfoils and hydrofoils with tubercle-inspired design have demonstrated significant gains in lift-to-drag ratio at moderate to high angles of attack, highlighting the potential of tubercles as low-energy flow-control devices. The design of tubercles found in nature have been used in engineering systems including wind turbine blades, marine propellers, hydro turbines, aircraft wings, and ventilation fans where tubercles improve efficiency, increase the usable angle-of-attack range, and decrease tonal noise.

Tubercle-inspired designs have also been adapted into sports products such as surfboards, swim fins and sailing foils to improve stability and maneuverability. Tubercles energize the flow around the surface and help reduce turbulent separation, this allows athletes to maintain smoother trajectories and reduce drag energy loss. This review focusses on the principles that affect tubercle performance with emphasis on sports applications equipment since small improvements in stability and efficiency can translate into significant performance gains. This study consolidates quantitative performance trends, geometric parameters like ratios of amplitude and wavelength, and operating regimes relevant specifically to sports engineering applications, unlike previous reviews that have primarily focused on flow physics or single engineering domains. The analysis shows the advantages such as improved lift, reduced drag and enhanced stability and investigates the limitations, which are dependent on geometry, operating conditions, and material constraints. Finally, research opportunities are identified, including optimization of tubercle designs, integration with other passive and active flow-control strategies, and exploring broader applications. The results offer a framework for further numerical, experimental, and product-focused research to incorporate bio-inspired tubercle concepts to enhance sports performance.

Key Words: *Bio-inspired design, Leading-edge tubercles, Hydrodynamic performance, Aerodynamic optimization, Sports engineering*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

DESIGN AND DEVELOPMENT OF PCM-THERMAL ENERGY STORAGE SYSTEM INTEGRATED SOLAR PV & SOLAR PTC-ASSISTED FLUIDIZED BED GRAIN DRYING SYSTEM FOR INDUSTRIAL APPLICATION

Shivasheesh Kaushik[1]*, Vinay Sati[1], Srinivasan Periaswamy[2], Srikanta Routroy[2]

[1]Research Scholar, Department of Mechanical Engineering, Birla Institute of Technology and Sciences, Pilani, Rajasthan, India

[2]Professor, Department of Mechanical Engineering, Birla Institute of Technology and Sciences, Pilani, Rajasthan, India

*Corresponding Author Email ID: p20210049@pilani.bits-pilani.ac.in

ABSTRACT

The rising demand for energy-efficient and sustainable agro-processing technologies emphasises the development of advanced thermal management solutions for continuous grain drying operations. This study provides a sustainable solution towards the consistency of energy supply and management for industrial grain drying application. A grain drying operation is developed by integrating the thermal energy storage system with solar PV & solar Parabolic Trough Collector (PTC) assisted Fluidized Bed Dryer (FBD) unit. With such an integration strategy, the extra energy developed by the solar PTC is stored in the Phase Change Material (PCM) bed, additionally the waste heat discharging from FBD chamber is also stored in the PCM bed. Thus, the waste heat recovery reducing the energy demand by both means, while discharging the PCM stored energy during the night. This investigation was performed experimentally and numerically to determine optimum operating condition. Analysis was performed with different geometrical parameters with constant velocity of 3 m/s. Larger the volume of the annular space, large energy can be stored leading in higher energy storage during the 8 hours charging. Maximum 11.47% of the energy consumption was reduced in the nights with the larger design available and preheating the air through the PCM bed. Similarly, the heating load is reduced by 5.84% through a waste heat recovery system equipped with PCM-based energy storage bed. A total of 17.30% energy has been recovered in a day by such an integrated system. Overall, the proposed integrated solar-assisted PCM-based fluidized bed drying system demonstrates substantial potential for energy conservation, operational reliability, and sustainable industrial grain drying, offering a viable pathway toward low-carbon agro-processing technologies.

Key Words: *Solar PV, Parabolic Trough Collector (PTC), Fluidized Bed Dryer (FBD), Phase Changer Material (PCM), Energy Storage System*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

UNSTEADY MIXED CONVECTION IN A TWO-SIDED LID-DRIVEN CAVITY WITH A POROUS ELLIPTICAL CYLINDER

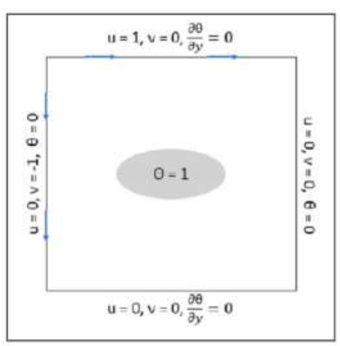
Jiban Chowdhury[1] and Y.V.S.S. Sanyasiraju[1]

[1]Department of Mathematics, IIT Madras, Chennai – 600036, ma21d004@smail.iitm.ac.in

ABSTRACT

A numerical investigation is conducted on the unsteady, two-dimensional, laminar mixed convection flow of an incompressible Newtonian fluid within a cavity featuring two-sided lid motion and an internally embedded hot, porous, elliptical cylinder. The opposing lid-driven motions generate the flow within the cavity, while the high-temperature cylinder has a significant influence on the flow dynamics. Flow through the porous cylinder is modeled using the Brinkman–Forchheimer–extended Darcy formulation, which accounts for porous medium effects. The porous medium is assumed to be isotropic and homogeneous, with constant permeability κ(= Da.L2) and porosity ξ, and identical thermal diffusivity is assumed for both the solid and fluid phases. In the absence of a porous medium, ξ→1 and κ→∞. The simulations employ a local radial basis function (RBF)-based meshless technique to accurately capture the flow behavior throughout the cavity. Computed results are analyzed for various cavity inclination angles and ellipse orientations across a range of key parameters, including the Richardson number ($0.01 \le Ri \le 100$), Darcy number (10-6 $\le Da \le$ 10-2), and Prandtl numbers (Pr = 0.71 for air and Pr = 6.9 for water). To examine the impact of the chosen parameters, the results are presented in the form of streamlines, isotherms, velocity profiles, and plots of local and average Nusselt numbers. The numerical results suggest that, at a fixed Grashof number (Gr), an increase in Richardson number (Ri) decreases the average Nusselt number (Nu), while an increase in Darcy number (Da) increases Nu, with the cavity inclination angle having a minimal effect on the convection rate.

Key Words: *Solar Mixed Convection, Porous elliptic cylinder, Brinkman and Forchheimer-corrected Darcy's model, Nusselt number*



International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

NATURAL CONVECTION IN A PARABOLIC-DIP POROUS CAVITY SUBJECTED TO SINUSOIDAL BOTTOM WALL HEATING

Akash Yadav[1] and Sanyasiraju V S S Yedida[1]

[1]Indian Institute of technology Madras, Chennai – 600036, mr.akashkhola@gmail.com

ABSTRACT

Natural convection in porous enclosures is strongly governed by boundary geometry and the imposed thermal conditions. Various studies have been done on natural convection in square cavity with rectangular dip on the top wall. In the present study, steady two-dimensional natural convection in a porous cavity with a parabolic-dip top wall is numerically investigated under a sinusoidally varying temperature applied along the heated bottom wall. The enclosure is assumed to be saturated with a incompressible Newtonian fluid and modeled using the Darcy–Brinkman–Forchheimer formulation in conjunction with the Boussinesq approximation. The governing streamfunction, vorticity, and energy equations are solved using a local radial basis function collocation method, which facilitates accurate discretization of the curved boundary without mesh generation. The effects of the Rayleigh number (*Ra*), Darcy Number (*Da*) and the amplitude of the sinusoidal temperature distribution are systematically examined. Flow and thermal fields are analyzed through streamline and isotherm contours, along with local and average Nusselt numbers along the heated wall. The results indicate that sinusoidal thermal modulation substantially modifies the flow structure by inducing localized convection cells and enhancing heat transfer near temperature maxima, while the presence of the parabolic dip intensifies thermal gradients in the upper region of the cavity, leading to strongly non-uniform heat transfer patterns. It is further observed that a uniformly heated bottom wall yields a higher average Nusselt number than the sinusoidal heating case, suggesting that non-uniform thermal forcing redistributes rather than maximizes convective transport. The study provides detailed physical insight into the coupled influence of boundary curvature and spatially varying heating on natural convection in porous enclosures and offers guidance for the thermal design of porous systems.

Key Words: *Natural Convection, Parabolic-Dip, Porous medium*

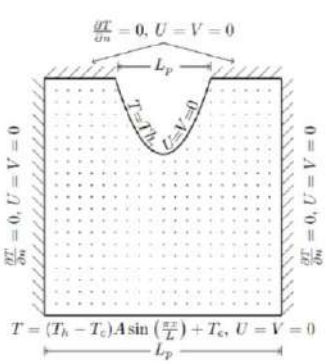


Figure: Schematic domain

EXPERIMENTAL INVESTIGATION OF THERMAL CONDUCTIVITY IN SOLID MATERIALS USING TRANSIENT PLANE SOURCE METHOD

Akshay S Bhat[1], Dr. G B Krishnappa[1], Dr. Srinivasan Kasthurirengan[2] and Dr. Govinda Rao Yenni[3]

[1]Vidyavardhaka College of Engineering, P.B. No. 206, Gokulam III Stage, Mysuru - 570 002, Karnataka, India, akshaybhat@vvce.ac.in, gbk@vvce.ac.in

[2]Centre for Cryogenic Technology, Indian Institute of Science, Bangalore - 560 012, Karnataka, India, ksrinivasan@iisc.ac.in

[3]Thermal Systems Group, U R Rao Satellite Centre, ISRO, Old Airport Road, Vimanapura Post, Bengaluru - 560 017, Karnataka, India, ygrao@ursc.gov.in

ABSTRACT

The precise characterization of thermophysical properties is quite important in the present-day material engineering. Recent decades have seen a paradigm shift toward transient methods, which offer rapid and non-destructive testing and simultaneous determination of multiple thermal transport properties. They use the hot strip and the hot disc sensors and are known as Transient Plane Sources (TPS). TPS method ("Hot Disk" method), utilizes a bifilar spiral sensor, made of nickel and shielded by Kapton film, serves simultaneously as a heat source and a resistance thermometer. This spiral sensor is placed in between identical planar samples. Time evolution of the sensor temperature, when supplied with a constant power is measured to obtain thermal conductivity of the sample under study. Theoretical foundations are well established, which models the sensor as a planar heat source embedded in an infinite medium. Although as on date, commercial instruments may be imported from many vendors at exorbitant costs, we have made a sincere attempt to develop the above experimental system indigenously. A thin double-spiral metallic silver film of 15 mm radius and 5-μm thickness is screen printed on a Kapton film of 125 μm and again insulated on its top by a similar Kapton film. The design of the sensor is kept similar to that of commercial system. The spiral sensor is placed in between the two identical slab shaped samples and is operated using a Keithley constant current source (Model 6220) which can supply a maximum of 100 mA. The sensor voltage measured by the Keithley DMM (Model 2000) is directly recorded using LabVIEW based DAQ program along with an IEEE-USB interface. Based on the temperature coefficient of resistance of the silver film, ΔT, the rise in temperature of the film is obtained. By plotting ΔT versus $D(\tau)$, we obtain the inverse of thermal conductivity of the material under study. At room temperature, preliminary experimental studies have been performed on non-metallic samples such as paper, wood-plastic composite, marble and granite as well as metallic samples such as stainless steel, brass and aluminium. The measured TC values for non-metallic samples are within ± 5% error, whereas those of metallic samples, the errors are much higher. This is due to fact that metallic samples need much higher heater powers for obtaining significant temperature rise. Suitable modifications of the setup are in progress to address the above issue. The novelty of the work lies in the development of an indigenous TPS experimental setup and the sensor development using screen printing of silver film, although considerable improvements of the setup is still in the offing.

Key Words: *Transient Plane Source (TPS) Method, Double-Spiral Silver Sensor, Thermal Conductivity*.

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

A NOVEL COMPACT RADIAL MULTIZONE DRYER: A SOLUTION FOR DRYING HEAT SENSITIVE COHESIVE PARTICLES

Santanu Dey[1], Thomas Tourneur[1], Axel de Broqueville[1], Juray De Wilde[1]

[1]Université catholique de Louvain, IMMC-IMAP, Place Sainte Barbe 2, 1348 Louvain-la-Neuve,

Belgium. E-mail address: mesantanudey@yahoo.com

ABSTRACT

A novel compact Radial Multizone Dryer (RMD) has been developed and tested at a large pilot scale. The RMD allows for efficient production of powders from organic solutions (Maltodextrin), specifically addressing the challenges of drying heat-sensitive and cohesive particles. Furthermore, applying multi-zone operation and process intensification through high-G operation in a centrifugal force field, the RMD is about an order of magnitude smaller than conventional spray dryers, for given capacity. This makes the device at least 20% more efficient than the conventional dryer. Also the unit's reliance on electricity for both compression and heating is no longer viewed as an operational disadvantage, but as a strategic alignment with future-proof, low-carbon industrial standards. The device employs a two-stage drying approach. Fast initial drying occurs in a central zone where hot air is fed counter-current with the atomized solution. Under the action of the centrifugal force, the initially dried particles are rapidly evacuated to the periphery where drying is completed using milder temperature air. The high-G operation enables a millisecond residence time in the central zone, high slip velocities intensifying mass/heat transfer, and separation and segregation of dried powders collected via various solid outlets. The device is currently at the pilot scale; however, finding suitable operating conditions under a specific geometry for long-duration operation remains challenging. In this article, the author discusses the spray drying of a 40% maltodextrin solution for approximately 5 minutes across two operating conditions where the height of the main spray drying chamber is varied. In Case 1, the height is 554 mm, and in Case 2, it is 724 mm. The shorter the chamber, the stronger the G-force, but the residence time decreases. Therefore, a trade-off between these parameters is essential. The article further discusses the amount of dry powder collected from the various collecting ports.

Key Words: *Radial multizone dryer (RMD), Vortex chamber, Spray drying, and Process intensification*

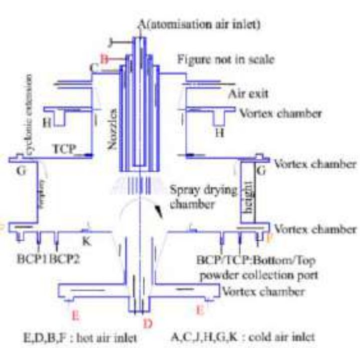


Figure: Schematic digram of the experimental set up

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

DESIGN, DEVELOPMENT AND ANALYSIS OF CROSS FLOW HEAT EXCHANGER FOR THE AIRCRAFT ENGINE OIL COOLING APPLICATION

[1]Giridhara Babu Y., [1]Felix J., [1]Sanmuganantham M., [1]Poonima N., [1]Janaki Rami Reddy M., [1]Kishor Kumar

[1]CSIR-National Aerospace Laboratories, Belur, Bangalore- 560037, giris.nal@csir.res.in

ABSTRACT

An Air-Cooled Oil Cooler (ACOC) is used in the aircraft engine oil cooling system for the removal of heat from engine bearings, gearboxes, propeller control systems etc. ACOC is a compact heat exchanger, which removes heat from hot oil by transferring its heat to ambient ram air (temperature of ambient air is cooler than the oil temperature). The oil is used in a closed loop circuit for the removal of heat from the engine components of these bearings, gearboxes, propeller control systems etc. The aircraft environment makes air cooling as a practical and reliable solution. Pertaining studies on heat exchanger mostly consider random temperature and mass flow rate of fluids in the heat exchanger. Further to enhance the heat transfer, mostly complex shapes of fins are employed which increases pumping power requirement for coolant flow and also possess manufacturing constraint. Also, experimental investigation on such problems is limited in the open literature.

In the present study, an in-house development of a cross flow heat exchanger is carried out along with the experimental analysis. It consists of series of oil channels and air channels, considering the oil flow perpendicular to the direction of air flow. Here, the oil channel encompasses lanced offset fins and the air channel encompasses the plain fins for better heat transfer and size miniaturization. The experimental investigation is performed at a fixed oil mass flow rate of 150 Lbs/min at varying air flow rates corresponding to the real flight conditions of aircraft. As per the engine conditions the corresponding inlet temperature conditions of air and oil are maintained at 37°C and 107°C respectively. Coriolis flow meter is used in the oil line to maintain the oil mass flow with the aid of inlet bypass valves. Orifice flow meter is employed in the air line to maintain the mass flow rate of air with aid of inlet and bypass valves.

The temperature and pressure drop of air across the ACOC are measured using RTDs and pitot tubes respectively. The oil temperature and pressure drop are measured using the RTDs and differential pressure sensor across the inlet and outlet ports of oil line respectively. The pumping power requirements are estimated for both fluids. The desired temperature of fluid is achieved by installing the required capacity heating units before the inlet ports of both the fluids. The heat loads and pressure drop data across the oil cooler for both the fluids are measured and compared with HW-LORI data. A good agreement is found between the inhouse developed ACOC experimental results and HW-LORI heat exchanger data.

Key Words: *Air cooled oil cooler, Compact Heat Exchanger, Crossflow, Engine conditions*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

RESIDUAL TIME-BASED CONTROL OF HETEROGENEOUS COOLING LOADS FOR DEMAND RESPONSE

Krishna Kumar Saini*, Hitesh Datt Mathur

Department of Electrical and Electronics Engineering

Birla Institute of Technology and Science Pilani, Pilani, Rajasthan, India

* Corresponding Author: p20210071@pilani.bits-pilani.ac.in

ABSTRACT

Heating, Ventilation, and Air Conditioning (HVAC) loads account for a significant portion of electricity demand, offering substantial potential for demand response (DR) services. However, their effective utilization remains challenging due to the inherent heterogeneity in thermal characteristics, operating conditions, and occupant comfort constraints across individual units. Existing DR control strategies often rely on simplified aggregate models or static priority rules, which fail to capture real-time thermal states and may lead to comfort violations or unstable aggregate power responses during DR events. A critical challenge, therefore, lies in dynamically sorting and prioritizing heterogeneous cooling loads in real time, while simultaneously ensuring user comfort and maintaining a reliable system. To address this challenge, this paper proposes a residual time-based control strategy for the coordinated management of heterogeneous cooling loads during DR events. The core concept of the proposed approach is the introduction of a residual time metric, defined as the estimated duration for which an individual cooling unit can remain in an off state without breaching predefined thermal comfort limits. This metric provides a physically meaningful and real-time indicator of flexibility, enabling the dynamic ranking and selection of loads for control actions. By continuously updating the residual time of each unit based on its thermal state, the strategy allows for intelligent prioritization of loads for both upward and downward regulation, thereby enhancing the responsiveness and reliability of aggregate demand modulation. The proposed control framework is supported by a second-order thermal model that accurately captures the dynamic behaviour of individual cooling loads, including thermal inertia and heat exchange processes. This modelling approach facilitates realistic aggregation of heterogeneous units and improves the fidelity of flexibility estimation at the system level. Simulation results demonstrate that the proposed residual time-based strategy effectively coordinates large populations of cooling loads while respecting individual comfort constraints. The proposed approach achieves smoother aggregate power responses, reduces rebound effects, and enhances overall DR reliability. The findings highlight the potential of residual time-based prioritization as a scalable and occupant-centric solution for integrating heterogeneous cooling loads into future demand response programs.

Key Words: *HVAC, Demand Response, Residual Time.*

INTELLIGENT IMPEDANCE CONTROL AND FORCE ESTIMATION FOR SURGICAL TOOL-TISSUE INTERACTION USING NEURAL NETWORK ADAPTIVE MODELING

Manmahendra Singh Daksh, Puneet Mishra

Department of Electrical and Electronics Engineering

Birla Institute of Technology and Science, Pilani - Pilani campus

Vidya Vihar, Pilani, Rajasthan, INDIA - 333031

*p20230041@pilani.bits-pilani.ac.in

* Corresponding Author

ABSTRACT

This proposed work introduces a simulation-based framework of a virtual single-degree-of-freedom (DOF) robotic system, designed to emulate a surgical robot with integrated haptic feedback. The robot is tasked with following a prescribed trajectory while interacting with biological tissue of varying stiffness. To achieve safe and adaptive interaction, a neural-network-driven adaptive impedance control scheme is implemented. Unlike conventional impedance control, which employs fixed stiffness and damping gains and may suffer performance degradation under variable dynamics or noise, the proposed approach incorporates a neural network that continuously tunes impedance parameters in real-time. The neural network accepts tracking error and its derivative as inputs and outputs adaptive impedance parameters constrained within predefined stability bounds. A robust nonlinear feedback term is incorporated to enhance disturbance rejection, while an observer estimates interaction forces in the presence of sensor noise. The overall scheme ensures bounded control torques and a safe interaction between the surgical robot and the environment (i.e., the tissue's reaction force). Simulation experiments are conducted using a sinusoidal reference trajectory to assess the controller's capabilities in trajectory tracking, force regulation, and adaptive behaviour. As a result, the achieved IAE and RMSE values are 0.001409 and 2.58×10^{-8}, respectively, for the considered test-case reference trajectory, indicating a negligible tracking error. The results confirm that the proposed control strategy delivers precise trajectory tracking, stable haptic force feedback, and enhanced robustness to noise and external disturbances when compared with fixed-parameter impedance schemes. Beyond surgical robotics, the neural-network-based adaptive impedance framework holds promise for broader applications, including human-robot interaction, rehabilitation devices, and haptic interfaces, where compliance and adaptability are essential.

Key Words: *Adaptive impedance control, Neural network control, Haptic feedback, Surgical robot, Trajectory tracking*

DESIGN AND THERMAL ANALYSIS OF CERAMIC BEARING

Mishra Dinesh Kumar Rao[1]*, C.K. Kaithwas1, Kshitij Pandey[2], Shivasheesh Kaushik[3]*, Navdeep Singh[4,5], Virendra Kumar Sharma[1], Rahul S Bharath[6], Amit Kumar[7]

[1]Mechanical Engineering Department, Dr.Ram Manohar Lohia Avadh University Faizabad, Ayodhya, Uttar Pradesh, India, Pin Code 224001
[2]Department of Mechanical Engineering, Uttaranchal Institute of Technology, Uttaranchal University, Dehradun, India
[3*] Department of Mechanical Engineering, Shivalik College of Engineering, Dehradun
[4]Department of University Centre for Research & Development (UCRD), Chandigarh University, Mohali–140413, Punjab, India.
[5]Department of Business and Communication, INTI International University, Nilai, Negeri Sembilan, Malaysia
[6]Department of Mechanical Engineering, BITS Pilani, Rajasthan
[7]Department of Mechanical Engineering, Motihari College of Engineering, Motihari
*Corresponding Author Email: erdinesh555@gmail.com, skaushik@sce.org.in

ABSTRACT

A bearing is a very small component of a machine that serves the purpose of reducing the amount of friction that occurs between moving parts and limiting relative motion to just the motion that is needed. One of the most important parts of ball bearing design is the selection of the material. This is due to the fact that the material influences such a wide range of bearing qualities. This article has demonstrated the considerable departures from the M standard bearing analysis that occur when rolling parts are manufactured from either of two unique materials. These findings have been presented in the context of the M standard bearing. Ball-bearing rings are often made of steel because it is the most common material utilised in their fabrication. In this investigation, ceramics will be the focus of the comparison, while steel will serve as the standard against which the comparison will be made. Bearings' primary purpose in an application is to carry loads while also making it possible for relative motion to occur. Bearings are able to facilitate relative motion in addition to their other functions. It is possible for the loads to originate either inside or outside of the bearing. These are the two probable places. When the application includes high speeds, it is probable that significant centrifugal loading will be formed on the outer raceway. At the present, the bearing is made out of stainless steel because that is the material that was available. The ceramic material is being subjected to the exact same load and put through its paces. After that, a thermal analysis is performed to determine the total heat flow that is present in the ball bearing that is currently in use for the temperature parameters that have been provided. This is done in order to determine whether or not the ball bearing can be used safely at the higher temperatures. putting into effect the temperature that is acting on the inner surface of the bearing, and then utilising the results not only to calculate the total heat flux for a specific material, but also to put into effect the temperature that is acting on the inner surface of the bearing.

Key Words: *Bearing, Stainless Steel, Thermal, Analysis*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

RELIABILITY ENHANCEMENT OF ROTATING EQUIPMENT USING VIBRATION ANALYSIS AND THERMOGRAPHY AS A TOOL

Dinesh Kumar Rao1*, C.K. Kaithwas1, Navdeep Singh3,4, Ruby Pant5, Rohit Kumar Singh Gautam2, Shivasheesh Kaushik6*, Premantusha Sahay7

[1]Department of Mechanical Engineering, Institute of Engineering and Technology, Dr. Ram Manohar Lohia Avadh University Faizabad, Ayodhya, Uttar Pradesh, India.
[2]Department of Mechanical Engineering, Teerthanker Mahaveer University, Moradabad, Uttar Pradesh, India.
[3]Department of University Centre for Research & Development (UCRD), Chandigarh University, Mohali–140413, Punjab, India.
[4]Department of Business AND Communication, INTI International University, Nilai, Negeri Sembilan, Malaysia.
[5]Department of Mechanical Engineering, Uttaranchal Institute of Technology, Uttaranchal University, Dehradun, India
[6*]Department of Mechanical Engineering, Shivalik College of Engineering, Dehradun
[7]Department of Mechanical Engineering, Govt Polytechnic Lucknow
[*]Corresponding Author Email: erdinesh555@gmail.com, skaushik@sce.org.in

ABSTRACT

With the industrial revolution, the range of rotating machinery's applications in daily life and for industrial use in manufacturing and processes, such as cement, automobiles, oil and gas refineries, etc., has greatly expanded. Furthermore, catastrophic failure is a substantial challenge for all of these rotating machines since defects brought on by bearing problems are a major factor in machine failure when it comes to fatigue. The condition monitoring vibration approach is well known for providing an accurate evaluation of the state of the rotating machinery when compared to other maintenance methods. These faults' vibration qualitative analysis is specific to the various vibration analysis techniques examined. In this study, bearing failures in rotating machinery are experimentally explored using a variety of vibration analysis approaches that are time-domain. In order to get the spectrogram, the signal from the rotating machinery's rolling element bearings—needle bearings with inner and outer race defects—is analysed in relation to the fast Fourier transform (FFT) and inverse fast Fourier transformation (IFFT). And the outcome of this experimental inquiry will be an accurate evaluation of the bearing problems that contribute to the failure of rotating machinery.

Key Words: *FFT, IFFT, Rotating Machinery, Spectra, and Bearing Faults*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

PARAMETRIC OPTIMIZATION OF EDM PROGRESSION FOR OVERCUT ON MACHINING OF INCONEL 600 BY TUNGSTEN CARBIDE TOOL WITH TAGUCHI TECHNIQUE

Sushil Kumar Choudhary[1], Dinesh Kumar Rao[2*], Navdeep Singh[3,4], Shivasheesh Kaushik[5*], Prabhakar Bhandari[6]

[1] Guest Faculty, Department of Mechanical Engineering, MCAET, Ambedkar Nagar U.P
[2*]Department of Mechanical Engineering, Dr. Ram Manohar Lohia Avadh University, Ayodhya, UP, 224001, India
[3]Department of University Centre for Research & Development (UCRD), Chandigarh University, Mohali–140413, Punjab, India.
[4]Department of Business AND Communication, INTI International University, Nilai, Negeri Sembilan, Malaysia.
[5*] Department of Mechanical Engineering, Shivalik College of Engineering, Dehradun
[6]Department of Mechanical Engineering, School of Engineering & Technology, K. R. Mangalam University, Gurugram, Haryana 122103, India
*Corresponding Author Email: erdinesh555@gmail.com, skaushik@sce.org.in

ABSTRACT

EDM is a sophisticated machining technology used to cut hard materials. EDM is useful because it can machine any electrically conductive metal alloy, regardless of its hardness. EDM is broadly utilized to shape or manufacture innovative technological materials that must withstand harsh environments. This research explores how input parameters of EDM impacted the cutting efficiency of Inconel-600 alloy. Tungsten Carbide (WC) was employed as the tool, while EDM oil was used as the dielectric solution. The Taguchi approach was used to determine the overall effect of input parameters such as Ton, Ip, and Vg for output response over-cut (OC) on the machining of Inconel 600 alloy. The investigation was carried out at three levels using an orthogonal-array (OA) L9. The S/N-(signal-to-noise ratio) and The ANOVA technique was used to examine the die sinking EDM performance aspects. Using the statistical programme MINITAB-17, the Taguchi approach was applied to enhance the input and performance variables. During the EDM process, the amount of overcut grows in a linear relationship with the amount of peak current. As the gap voltage rises, so does the amount of overcut. OC reduces somewhat as Ton grows, then increases as Ton increases. At the first level of Ip, first level of Vg, and second level of Ton, the overcut (OC) is the lowest. The Ip and Vg have a substantial influence on overcut, according to ANOVA. Peak current (30 A), gap voltage (90 Volt), and pulse on time (100-sec) are the best settings for overcut. The 95 percent confidence level predicted ideal range of overcut (OC) was the optimum findings obtained, which were confirmed by performing confirmation trials, where the overcut was produced as 0.012 mm.

Key Words: *EDM, Tungsten Carbide, Over Cut, Inconel 600, ANOVA, Taguchi method*

SEMG FINGER MOVEMENT CLASSIFICATION USING DEEP LEARNING MODELS

Pooja Maurya[1], Kunj Goel[1], Swarit Kachroo[1] and Sujan Yenuganti[1]

[1]Dept. of Electrical and Electronics Engineering, Birla Institute of Technology and Science Pilani, Rajasthan, India 333031.
*p20210429@pilani.bits-pilani.ac.in, f20220817@pilani.bits-pilani.ac.in, f20221150@pilani.bits-pilani.ac.in
yenuganti.sujan@pilani.bits-pilani.ac.in

ABSTRACT

Surface Electromyography (sEMG) signals play a crucial role in understanding neuromuscular activity and have been extensively applied in prosthetic control, rehabilitation engineering, and human machine interaction systems. Accurate classification of finger movements using sEMG signals remains a challenging task due to signal nonlinearity, noise, and inter-subject variability. In this study, sEMG signals were recorded from 9 healthy subjects while performing 11 distinct finger movement tasks to develop a robust finger movement recognition system using deep learning techniques. The recorded sEMG signals were subject to preprocessing steps, including filtering, normalization, and segmentation, to enhance signal quality and extract meaningful information. To effectively classify finger movements, two advanced deep learning models were designed and evaluated a Hybrid Convolutional 1D Bidirectional Long Short-Term Memory (Conv1D-BiLSTM) model and a Deep 1D Convolutional Neural Network (Deep 1D-CNN) Model. The Conv1D layers were utilized to automatically extract discriminative spatial features from the sEMG signals, while the Bidirectional LSTM layer captured long-term temporal dependencies in both forward and backward directions. In parallel, the Deep 1D-CNN model was implemented to learn hierarchical feature representations directly from raw sEMG signals, eliminating the need for manual feature extraction. Comprehensive experimental evaluations demonstrate that both models successfully classify multiple finger movements with high reliability. Notably, the Hybrid CNN-BiLSTM model achieved superior classification accuracy compared to the Deep 1D-CNN model, indicating its enhanced capability to learn complex spatial temporal patterns inherent in sEMG data. The novelty of this work lies in the spatial sequential modeling of sEMG signals using a Conv1D BiLSTM architecture for accurate classification of 11 distinct finger movements, outperforming a deep 1D-CNN approach. This study highlights the effectiveness of deep learning-based approaches for sEMG signal classification and provides valuable insights for developing advanced prosthetic control systems and intelligent rehabilitation technologies.

Key Words: *sEMG Signal, Finger Movement, Deep Learning, CNN, BiLSTM, Feature Extracion.*

ENHANCING MILK SHELF LIFE THROUGH INSULATING MATERIALS: A SYSTEMATIC REVIEW OF STORAGE TECHNOLOGIES

Akhilesh Kumar[1], Srikanta Routroy[2], Subhasis Pradhan[3], Sushil Yadav[4] and Suvanjan Bhattacharyya[5]

[1]Research Scholar, Department of Mechanical Engineering, BITS Pilani, Pilani Campus,333031, India, p20241009@pilani.bits-pilani.ac.in
[2]Professor, Department of Mechanical Engineering, BITS Pilani, Pilani Campus,333031, India, srikanta@pilani.bits-pilani.ac.in
[3]Assistant Professor, Department of Civil Engineering, BITS Pilani, Pilani Campus,333031, India, subhasis.pradhan@pilani.bits-pilani.ac.in
[4]Senior Veterinarian, Department of Pharmacy, BITS Pilani, Pilani Campus,333031, India, sushil.yadav@pilani.bits-pilani.ac.in
[5]Assistant Professor, Department of Mechanical Engineering, BITS Pilani, Pilani Campus,333031, India, suvanjan.bhattacharyya@pilani.bits-pilani.ac.in

ABSTRACT

With the rapid growth of the global population, the demand for milk has increased substantially. Milk is a nutritionally complete food that fulfills many essential dietary requirements; however, it is highly perishable and requires storage at 2–4 °C to maintain quality and extend shelf life. Temperature fluctuations during storage and transportation under ambient conditions accelerate microbial growth, leading to rapid deterioration of milk quality. Consequently, effective thermal insulation is critical for maintaining temperature stability in milk storage and cold-chain systems. Thermal insulation plays a vital role in minimizing heat ingress, stabilizing internal temperature fields, and improving the overall energy efficiency of refrigeration systems. This systematic literature review examines insulating materials used in milk storage and refrigeration systems with the objective of reducing temperature fluctuations. The thermal performance, durability, moisture resistance, and practical applicability of commonly used and emerging insulation materials are critically analyzed. Conventional insulation materials, such as polyurethane foam, expanded polystyrene, and extruded polystyrene, are widely used due to their low cost and ease of manufacture; however, their insulating performance degrades over time due to moisture absorption and ageing. Advanced insulation technologies, including vacuum-insulated panels and aerogel-based composites, offer exceptionally low thermal conductivity and reduced thickness requirements, but their widespread adoption is limited by high costs, mechanical fragility, and long-term vacuum loss. Recent studies highlight the integration of phase change materials to mitigate temperature fluctuations during power outages and logistical disruptions. Additionally, this review explores environmentally sustainable alternatives and novel hybrid insulation concepts, including crop-residue-based insulating materials. The review identifies key research challenges, including improving long-term durability, reducing costs for decentralized and rural storage, and developing sustainable and eco-friendly insulation solutions. The findings of this study provide valuable guidance for selecting appropriate insulating materials for milk storage, transportation, and cold-chain logistics.

Keywords: *Milk Supply Chain, Insulating Materials, Phase Change Material, Shelf-Life Enhancement of Milk*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

THERMAL ANALYSIS DURING MICRO-DIMPLE FABRICATION IN MECHANICAL MICROMACHINING OF TI-6AL-4V

Abhishek Tandey*, Anuj Sharma, Suvanjan Bhattacharyya

Department of Mechanical Engineering, Birla Institute of Science and Technology, Pilani, Pilani Campus, Pilani 333031, Rajasthan, India.

*Corresponding Author: Abhishek Tandey

E-mail address: p20241001@pilani.bits-pilani.ac.in

ABSTRACT

Mechanical micromachining of titanium alloys is a crucial process for precision material removal, with significant applications in aerospace, biomedical, defence and microelectromechanical systems (MEMS) devices. Despite the ability to create highly precise features such as micro dimples and holes, the machinability of Ti-6Al-4V remains challenging due to its low thermal conductivity and high chemical reactivity. Because of the electron transport characteristics, intense heat is trapped at the tool tip. This characteristic makes it difficult to accurately record the temperature of the workpiece during machining, as the heat generated in the cutting zone is not quickly dissipated through the bulk material and accumulates. The deformation causes a highly localized temperature increase at the tool and workpiece interface. These temperature changes affect the instability of cutting forces, the progression of tool wear, the formation of chips and burrs and the resulting surface integrity. Although we are processing at the microscopic scale, nanoscale deformation mechanisms influence material removal. The thickness of the uncut chip is comparable to the cutting radius, resulting in significant dimensional effects. Therefore, thermal and mechanical phenomena are strongly coupled and exhibit different behaviour from conventional machining. This study presents a comprehensive experimental investigation of the cutting force and temperature changes in the cutting zone of harder-to-cut material utilising thermomechanical interactions during the mechanical micromachining of Ti-6Al-4V. Real-time cutting forces are measured using a dynamometer, while temperature evacuation is performed through an integrated K-type thermocouple and a thermal image-based measurement approach, enabling the estimation of both local and spatial temperature distributions during micro-milling. To gain clearer insight into process effects, kinematics and temperature behaviour are studied independently in up-milling and down-milling processes. This strong correlation emphasizes that the processing temperature is a reliable indicator of tool condition and process stability during micro-milling. Results make the difference in ideas to optimize process parameters, extend tool life and increase machining stability, while supporting sustainable micromachining by reducing tool consumption, rework, and machining waste of material and energy.

Key Words: *Mechanical Micromachining, Titanium Ti-6Al-4V alloy, Cutting-zone temperature.*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

FROM PERIODIC OSCILLATIONS TO CHAOS IN A SINGLE-BRANCH PULSATING HEAT PIPES SYSTEM

Bilash Thakur[1], Alok Kumar[*2], Dipankar Das[*1], Suneet Singh[*1]

[1]Fluid Flow Systems Simulation Lab, Department of Energy Science and Engineering,
Indian Institute of Technology Bombay, Mumbai, 400076, India
[2]Stephen B. Klein Faculty of Aerospace Engineering,
Technion-Israel Institute of Technology, Haifa, 3200003, Israel
Author email: bilash.thakur@iitb.ac.in; alok.kumar@campus.technion.ac.il; dipankar.das@iitb.ac.in;
suneet.singh@iitb.ac.in

ABSTRACT

Nonlinear oscillatory behavior for the single-branch pulsating heat pipes (SBPHPs) plays an important role in determining their thermal performance and stability. In this study, we investigate the route to chaos in a reduced three-dimensional dynamical model of an SBPHP by systematically varying the dimensionless frictional coefficient while keeping the phase change parameter fixed. The governing equations couple with frictional forces, viscous dissipation, phase change processes, and thermal interaction, capturing the essential physics for self-sustained oscillations. A bifurcation diagram is shown by extracting local maxima of the oscillatory state variable, showing clear identification of qualitative changes in the system dynamics. As frictional coefficient is varied, the system exhibits a sequence of period-doubling bifurcations, progressing from stable periodic oscillations to multi-periodic states and ultimately to chaotic motion. The results demonstrate that fluid damping acts as a critical control parameter governing nonlinear transitions in pulsating heat pipes. The presented bifurcation analysis provides quantitative insight into the onset of complex oscillations and establishes a foundation for further studies involving Lyapunov exponents, and thermal performance optimization. These findings contribute to a deeper understanding of nonlinear flow–thermal coupling in pulsating heat pipe systems and may guide the design of more stable and efficient thermal management devices.

Key Words: *Heat Transfer, PHP, Period double bifurcation.*

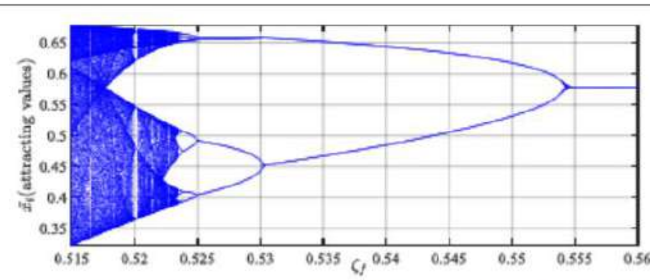


Figure 1: Period-doubling bifurcation structure of the reduced SBPHP model, highlighting the transition from periodic to chaotic oscillations as the frictional parameter is varied at fixed phase change parameter.

BIFURCATION ANALYSIS OF TWO-PHASE FLOW INSTABILITIES IN A SINGLE HEADTED CHANNEL

Dipankar Das[1], Bilash Thakur[1] and Suneet Singh[1]

[1]Fluid Flow Simulations Lab., Department of Energy Science and Engineering Indian Institute of Technology Bombay, Mumbai-400076, India dipankar.das@iitb.ac.in, bilash.thakur@iitb.ac.in, suneet.singh@iitb.ac.in

ABSTRACT

Understanding flow instabilities in boiling heated channels is essential for the safe and reliable operation of many thermal systems. Over the years, different types of flow instabilities have been linked to specific bifurcation mechanisms. Dynamics two-phase instabilities such as pressure-drop oscillations (PDO) and density-wave oscillations (DWO) are commonly associated with Hopf bifurcations, whereas static instabilities like Ledinegg instability are related to pitchfork or saddle-node bifurcations. Recent studies using co-dimension-two bifurcation analysis (where two system parameters are varied simultaneously) have identified three distinct instability regimes in the operating parameter space: Ledinegg instability, flow excursion with compressible volume (FECV), and pressure-drop oscillations. While these regimes have been individually characterized using time-domain responses and phase-space representations, the mechanisms governing transitions between them remain poorly understood. In this work, we address this gap by performing a systematic co-dimension-one bifurcation analysis (where a single control parameter, the inlet restrictor coefficient K1, is varied). Bifurcation analysis is carried out using the MATLAB-based continuation tool MATCONT. The results reveal that saddle-node (limit point) bifurcations govern the transition from the FECV regime to pressure-drop oscillations, whereas pitchfork (branch point) bifurcations mark the shift from Ledinegg instability to the FECV regime. Numerical simulations and phase-space analyses at the identified bifurcation points further clarify the underlying nonlinear dynamics and transition behavior.

Key Words: *Two-phase flow, Ledinegg Instability, Pressure Drop Oscillations, Bifurcation analysis, Hopf Bifurcation*

SOLAR ENERGY INTEGRATION IN SYNTHESIS OF SUPPLEMENTARY CEMENTITIOUS MATERIALS FOR INDIAN CEMENT INDUSTRY

Ankur Mittal[1] and Manoj Kumar Soni[1]

[1]Birla Institute of Technology and Science, Pilani (Raj), p20220301@pilani.bits-pilani.ac.in

ABSTRACT

The cement manufacturing units are classified under very high energy intensive sectors which requires around 725 kCal/kg of thermal energy and 80 kWh/MT electrical specific energy consumption, respectively. Nowadays cement industries are focusing to reduce and trying to replace traditional fossil fuel consumption by adopting the use of energy from renewable resources such as wind mills, solar energy systems. Solar thermal technology can be used in numerous ways in the cement industry like to provide warm water for processes, hot air for drying or calcining the raw materials etc. The Indian cement industry is facing a shortfall of Supplementary Cementitious Materials (SCMs) and thus relying on the import of such materials. On the other hand, Waste/by-product of other industries, like wet fly ash, red mud, phosphogypsum, lime sludge, etc., are abundant in huge quantities that can be used in cement plants with drying and medium temperature calcination process. A suitable green and clean energy option can be explored to raise the temperature for calcination. A reliable solar thermal energy system can raise temperature of heating medium (air in case of present work) to directly transfer heat to the material to be dried in a dryer vessel. If the sufficient amount of hot air is present, then waste/by-product calcination can also be acknowledged. The working application can be categorized among low temperature application (up to 100 ^{o}C and to heat water), medium temperature application (upto 400 ^{o}C and meeting like calcination of SCMs). A feasibility concept of using Solar Thermal Energy is discussed in this paper. A successful demonstration of using solar thermal technology is also briefed here towards thermal conditioning of phosphogypsum to concert impurities in inert form, making the material competent to replace mineral gypsum in cement plants.

Key Words: *Cement, Solar Thermal Energy, Supplementary Cementitious Material*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

CFD ANALYSIS OF A RANQUE-HILSCH VORTEX TUBE: BRIEF REVIEW

Pankaj Kumar Sagar[1] and Akhilesh Arora[2]

[1,2] Department of Mechanical Engineering, Delhi Technological University, New Delhi-110042.
Email*: pankajkumarsagar_23phdme02@dtu.ac.in

ABSTRACT

The Ranque-Hilsch vortex tube (RHVT) is a thermo-fluidic device that has no moving part and utilizes compressed air/gas as its working medium. The energy and flow separation in RHVT totally depends on the size of the nozzle, number of nozzles, length and diameter of the RHVT, inlet and outlet pressures, control valve, hole size of the diaphragm, and hot and cold mass fractions. This review highlights advancements in computational fluid dynamics (CFD) analysis and experimental investigations on the energy and temperature separation phenomena in RHVT. The review presents various geometrical modifications, working fluids, and operational parameters to enhance the efficiency of RHVT. Many researchers carried out CFD analysis of the RHVT using various turbulence models like RANS, k–ε, k–ω, and BSL k-x to predict the energy separation. Some researchers used an artificial neural network (ANN) method for the CFD analysis in RHVT and gave suggestions to improve the performance of the Ranque-Hilsch vortex tube. Advanced turbulence flow models and thermodynamic analyses provide deeper insights into flow behaviors, optimizing energy separation and enhancing the coefficient of performance (COP). These available results indicate the potential of CFD and experimental methodologies in advancing the design and efficiency of vortex tubes for residential, commercial, and industrial applications.

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

PERFORMANCE ENHANCEMENT OF A REFRIGERATION SYSTEM BY UTILIZING ITS WASTE HEAT IN VAPOUR ABSORPTION REFRIGERATION SYSTEM

[1*]Ksheerja Arora, [1]Swastika Singh and [2]Pramendra Kumar Bajpai
[1,2] Netaji Subhas University of Technology, New Delhi, 110078
Email*: ksheerja.arora.ug22@nsut.ac.in

ABSTRACT

The climate change due to the issue of Global Warming related to the HFC refrigerants is a pertinent problem. Most of the refrigerants which are used now a days belong to HFC category. However HFC refrigerants have high Global Warming potential (GWP) in comparison to CO2 for which GWP is 1. Limiting the use of HFC refrigerants and using natural refrigerants seems to be a feasible solution to mitigate Global Warming. The natural refrigerant CO2 is a preferred refrigerant which is finding an increased interest in refrigeration systems for different applications. The use of CO2 as a refrigerant is feasible in transcritical vapour compression cycle since the temperature in many Indian cities goes beyond 45ºC. The utilisation of Vortex tube (VT) in the transcritical cycle with CO2 has shown to enhance the COP and cooling capacity in comparison to transcritical vapour compression cycle without VT. However, major drawback of both these cycles is high temperature of the CO2 gas leaving the compressor due to which the gas cooler rejects large amount of high temperature heat to cooling media.

This study introduces a novel cycle in which a single stage water lithium bromide Vapour absorption refrigeration (VAR) system is operated by the waste heat of the gas cooler of a Vortex Tube integrated single stage trans-critical vapour compression refrigeration (VT-SS-TC-VCR) cycle. Since no external source of energy is utilised for operating the VAR cycle hence it is best way of utilising the heat going waste from the gas cooler thereby effectively mitigating the global warming associated with the production of CO2 due to burning of either fossil fuels or oil required for producing heat for the operation of VAR cycle.

This study examines the performance analysis of the conceived cycle (VT-SS-TC-VCR + VAR) under different operating conditions such as evaporator temperature and gas cooler outlet temperature. A code in Engineering Equation Solver is developed based on mass and energy conservation for the analysis of this cycle. Initially the optimum pressure and temperature at inlet to Vortex tube are computed for the maximum COP of the VT-SS-TC-VCR cycle corresponding to different evaporator and gas cooler outlet temperatures. The cooling capacity obtained is in the range of 130 kW to 155 kW. The computed results show that the heat going waste from gas cooler is in the range of 1.61 to 2.24 times that of the cooling capacity of the VT-SS-TC-VCR cycle and the average entropic temperature of gas cooler is in the range of 60ºC to 110ºC for the gas cooler outlet temperature varying between 34ºC to 50ºC and evaporator temperature varying between -35ºC to 0ºC. The gas cooler waste heat is utilised to operate VAR cycle from which additional cooling capacity upto 230 kW can be achieved depending upon different operating conditions.

POWERING HEALTHCARE WITH GREEN HYDROGEN: A SOLAR–WIND HYBRID PATHWAY FOR RURAL HOSPITALS IN JAISALMER

Kavya S, Abhinav Kamath, Srikanta Routroy, Suvanjan Bhattacharyya*

Department of Mechanical Engineering, Birla Institute of Technology and Science Pilani, Pilani Campus, Vidya Vihar, Pilani, Rajasthan - 333031, India.
*Corresponding Author: suvanjan.bhattacharyya@pilani.bits-pilani.ac.in

ABSTRACT

Reliable and uninterrupted energy supply is critical for healthcare delivery, particularly in rural and remote regions where grid instability and diesel dependency compromise medical services. In arid regions such as Jaisalmer, Rajasthan, rural hospitals face persistent energy challenges due to high ambient temperatures, frequent power outages, and limited access to clean backup power. This study proposes and evaluates a solar–wind hybrid green hydrogen–based energy system as a sustainable and resilient pathway to power rural healthcare infrastructure in Jaisalmer. The proposed system integrates photovoltaic (PV) arrays and wind turbines to exploit the region's abundant solar irradiance and favorable wind potential. Surplus renewable electricity is utilized for water electrolysis to produce green hydrogen, which is stored and later converted to electricity using fuel cells during periods of low renewable generation or peak hospital demand. A comprehensive techno-economic, environmental, and reliability assessment is conducted to evaluate system feasibility under real climatic and load conditions representative of rural hospitals in western Rajasthan.
Hourly solar radiation, wind speed, and temperature data are used to model renewable generation, hydrogen production, storage dynamics, and power dispatch. Key performance indicators include system efficiency, hydrogen yield, loss of power supply probability (LPSP), levelized cost of energy (LCOE), and greenhouse gas emission reduction compared to conventional diesel-based backup systems. Results demonstrate that the hybrid solar–wind configuration significantly enhances energy reliability, achieving near-zero unmet load for critical hospital operations. The green hydrogen subsystem provides long-duration energy storage, overcoming the intermittency limitations of standalone renewable systems and ensuring continuous power for essential medical equipment, cold storage of vaccines, diagnostic facilities, and emergency services. From an environmental perspective, the proposed system achieves substantial reductions in carbon dioxide and particulate emissions, contributing to improved local air quality and alignment with India's National Green Hydrogen Mission and sustainable healthcare goals. Economically, while the initial capital investment is higher than diesel-based systems, long-term operational savings, fuel cost avoidance, and emissions benefits make the system financially viable over its lifecycle, particularly under policy incentives and declining costs of electrolyzers and fuel cells. This study highlights the transformative potential of green hydrogen as a clean energy vector for rural healthcare, especially in resource-rich but infrastructure-constrained regions like Jaisalmer. The findings provide a scalable and replicable framework for deploying renewable-hydrogen energy systems in rural hospitals across arid and semi-arid regions, strengthening healthcare resilience, energy security, and climate sustainability.

Key Words: *Green hydrogen, solar and wind energy, fuel cell, sustainable healthcare, oxygen prodution.*

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

FLOW SUDY OF CLAY MATERIAL IN LDM (LIQUID DEPOSITION MODELING) 3D-PRINTING TECHNOLOGY

Tarun Kanti Pal[1*], Dipankar De[2], Asit Kumar Singh[2], Suvanjan Bhattacharyya[3], and Anjan Bhunia[1]

[1*,1] Department of Mechanical Engineering, College of Engineering & Management, Kolaghat, KTPP Township, West Bengal 721 171, India
[2]Centre for Development of Advanced Computing (C-DAC), Kolkata
[3]Department of Mechanical Engineering, Birla Institute of Technology and Science, Pilani Rajasthan, India
[1*]Corresponding Author E-mail: tarunkantipal@gmail.com

ABSTRACT

Liquid Deposition Modeling (LDM) — paste extrusion or direct ink writing (DIW) of clay-based pastes — is an increasingly important route for ceramic manufacturing, architectural components, and sustainable fabrication. Clay pastes are complex, particle-filled fluids exhibiting yield stress, shear-thinning, and often thixotropy and weak viscoelasticity. Predicting the flow structure (plug formation, shear layers, die-swell and exit behavior) inside barrels/nozzles and during deposition is essential to control printability and final part quality. This study synthesizes analytical and experimental approaches for modeling clay flow in LDM: constitutive models (Bingham, Herschel– Bulkley), regularization techniques (Papanastasiou), canonical axisymmetric nozzle solutions, rheometry, ram extrusion, high-speed imaging. The study focus on summarize representative parameter ranges, identify key physics often omitted (thixotropy, particle migration, drying/shrinkage coupling), and propose benchmark tests and best practices for modelers and experimentalists.

Key Words: *Liquid Deposition Modeling, clay paste, yield-stress fluids, Herschel–Bulkley, Papanastasiou regularization, nozzle flow, die-swell, ram extrusion, rheology, computational modelling*

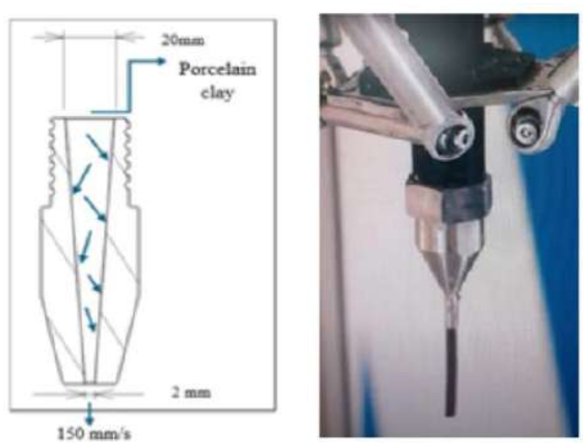


Figure: Crosssection view LDM Clay Printer Nozzle and actual view of nozzle flow

SPONSOR

SPARC

Aimil

International Conference on
Fluid and Thermal Engineering (ICFTE 2026),
January 19-21, 2026,
BITS Pilani, Pilani Campus

PREVIOUS SPARC SPONSORED EVENT HOSTED BY THE DEPARTMENT OF MECHANICAL ENGINEERING, BITS PILANI, PILANI CAMPUS.