International Conference on
Thermo-Fluids and System Design
(*ICTFSD 2024*)

April 04–05, 2024

Abstract Book

**Department of Mechanical Engineering,
Birla Institute of Technology,
Mesra, Ranchi-835215, Jharkhand.**

Preface

Dr. Om Prakash
Convener,
Dept. of Mechanical Engineering

Dr. Lakhbir Singh Brar
Convener
Dept. of Mechanical Engineering

Dr. Arkadeb Mukhopadhyay
Organizing Secretary,
Dept. of Mechanical Engineering

Dr. Saurav Chakraborty
Organizing Secretary,
Dept. of Mechanical Engineering

Dr Sushanta Ghuku
Co-Organizing Secretary, Dept. of Mechanical Engineering

In view of the recent development of various areas of Mechanical Engineering like thermo-fluids and system design around the world, the need for a rigorous discussion on development of the associated science on different applications was felt. With this view this International Conference was planned to bring the experts from academic institutions, research organization and industries, and policymakers involved in this area, on a common platform to facilitate close interactions and to share and exchange their views pertaining to the contemporary issues and future challenges in the field of thermo-fluids and system design.

The objective of the conference was to provide a forum for the free and fruitful exchange of ideas on Thermo-Fluids and System Design. The conference has been organized to promote result oriented, industry related research in academic institutions and research organizations. It will create a network among various institutions and industries for effective collaboration in research and development. This conference will also help to identify different areas which require research and development in the near future.

The International conference on Thermo-Fluids and System Design [ICTFSD-2024] is organized by the Department of Mechanical Engineering, Birla Institute of Technology, Mesra, Ranchi (India) on April 4-5, 2024. All conference papers were reviewed by a technical review committee consisting of professors from reputed institutions. We are grateful to each of the conveners, organizing secretaries and various committee members for their work in managing the review process and in the compilation of this abstract volume of this conference.

We thank all the scholars who responded readily to our call for papers and participated in this conference. All the papers received were sent for blind peer reviews to the reviewers and after their review and recommendation, papers have been accepted for presentation in two different themes of the conference viz. Thermo-Fluids and System Design. We would like to extend our thanks to all the participants for their overwhelming response. We would also like to express our heartfelt thanks to all the reviewers for extending their help and expertise for the timely review of papers.

Also, we record our gratefulness to Dr. Indranil Manna, Vice Chancellor, BIT Mesra and Patron, ICTFSD 2024 for his persistent help and support in facilitating each required task.

Last but not the least, as a witness to the hard work done by our faculty and staff members; we want to congratulate each of them for their immeasurable efforts.

We wish this conference a great success.

Contents

Thermal Management of Microelectronics Using Microchannel Heat Sink with Inclined Geometry and CNT-based Nanofluid

Manoj Kumar[1,2,a)], Sheshang Singh Chandel[1,b)], Sumit Sinha-Ray[3,4,c)*]

[1]School of Mechanical and Materials Engineering, Indian Institute of Technology Mandi, Mandi, HP-175005, India,
[2]Department of Mechanical and Aerospace Engineering, NIIMS University, Jaipur
[3]Department of Textile and Fiber Engineering, Indian Institute of Technology Delhi, Hauz Khas, New Delhi
[4]Department of Mechanical and Industrial Engineering, University of Illinois at Chicago, Chicago, Illinois 60607-
7022, USA

In the present study, the performance of an inclined channel in microchannel heat sink (MCHS) has been investigated through computational approach with aqueous nanofluids and DI Water, where the primary second phase constituents of nanofluids are copper oxide (CuO) and carbon nanotube (CNT). An MCHS of dimension 10 mm x 900 µm x 100 µm (length x height x width) with engraved microchannel of dimension 10 mm x 180 µm x 57 µm is designed with a pre-set inclination (1, 0.66 and 0.33o) and the cooling performance of three coolants, i.e., Water, Water + CuO and Water +CNT is evaluated with different input parameters. Nanofluids are considered as single-phase fluids with simulated and established thermophysical properties. In the first part of this work, the local temperature distribution, local heat transfer coefficient and pumping power are chosen as the parameters for determining the best performance in terms of longevity of a microelectronic device which stems from a specific design while using Water as coolant, and in the subsequent part, the optimized geometry is rendered even more efficient with nanofluid and modified flow parameters. It is observed that Nusselt number increases by 2.4% for mere 1 o inclination as compared to the case of the straight microchannel with Water as a coolant. The efficiency index (h), a parameter governed by flow and thermal resistance, is shown to be highest for 1o inclination with Water+CNT as coolant. The primary reason for enhanced cooling with inclination is induced pumping arising from gravitational drainage, that may assist in reducing overall pumping power. This additional cooling gets augmented due to uniaxial elongation like flow of nanofluid that enhances the local volume fraction of thermally conductive materials, especially for CNT due to its high aspect ratio; a phenomenon that is weaker in CuO-based nanofluid.

Keywords: Microchannel heat sink; Inclination; Nanofluid; Pumping power; Efficiency index.

Performance of the Helical Hydrokinetic Turbine – A Review

Avinash G Shahabadi[a], Abdus Samad[b]

[a] Department of Ocean Engineering, IIT Madras, Chennai, Tamil Nadu, 600036, India.
[b] Department of Ocean Engineering, IIT Madras, Chennai, Tamil Nadu, 600036, India.

Hydrokinetic turbines harness the kinetic energy of water flowing in a river, tidal currents, and many other natural or artificial passages. These turbines can be employed in the perennial river stream and the vast coastline. This helps national interest in self-reliance in the energy sector, as these sources are freely available and have minimal environmental effects. Vertical axis turbines are the most preferred choice as they are more stable and have lesser structural noise than horizontal axis turbines. A helical hydrokinetic turbine (HHKT) is a type of vertical axis turbine with helical airfoil or twisted blades that ensure a constant torque on the blade, thus reducing the power fluctuation at the generator end. However, it can extract a maximum power of 59.23% from free streams of about because of the Betz limit. This paper presents the various design parameters that affect the performance of the HHKT and the various performance enhancement methods proposed for the same.

Keywords: Renewable energy, Hydrokinetic turbine, Tidal energy, Performance parameters.

Heat transfer analysis around a heated tube under cross flow of air

Apoorva Deep Roy[a], S. K. Dhiman[b]

Department of Mechanical Engineering, Birla Institute of Technology, Mesra,
Ranchi-835215, (India), Phone: +91- 0651-2275924

Experiments have been performed on a copper tube placed in a subsonic wind tunnel to calculate local Nusselt number (Nu). Power law relationship: $Nu= C Re^{m}Pr^{n}$, correlates Nusselt, Reynolds and Prandtl numbers around a circular tube in the cross flow of air. Inputs of local Nu data have been taken to estimate constants C and m and subsequently correlations for estimation of overall average Nu have been developed. Matrix least squares method has been used for the estimation of values of constant (C) and power indices (m) through a MATLAB program. Correlations (19) through (22) have been developed for estimation of average Nu on overall tube surface and for different flow regimes. An empirical logarithmic expression (23) has also been proposed for estimating the overall Nu_{av} by curve fitting in MATLAB with minimum residual. Estimates obtained from correlations (19, 20 and 21) have shown a good agreement with reported literature [20] falling within the limit of 4%. The overall Nu_{av} from the present study developed correlations (22 and 23) shows a good agreement with reported literature [2], having minor deviations within 2.75% for correlation (22) and 1.1% for correlation (23) respectively and the estimated values of constant (C) and power indices (m) of power law relationship by using Matrix Least square method also came out to be reasonably acceptable.

Keywords: Correlations around tube; Power-Law; Coefficients and Indices; Nusselt number.

Advances in Striling Engine Engineering: I. Finned Cylinder Walls for Improved Cooling

Dana I. Almansoori[a], Khaled Al-Aribe[a], Osama A. Mohamed[b], Basem F. Yousef[c]

[a]Department of Mechanical Engineering, Abu Dhabi University, Abu Dhabi, UAE
[b]Department of Civil Engineering, Abu Dhabi University, Abu Dhabi, UAE
[c]Department of Mechanical and Industrial Engineering, The American University of Ras Al Khaimah, UAE

A Stirling engine converts a temperature differential into mechanical work. In order to improve the engine's performance, the temperature difference between the hot and cold ends of the cylinder should be increased, and the addition of fins enhances this process. This paper's main objective is to simulate a Stirling engine with finned and unfinned surfaces by primarily using COMSOL Multiphysics. Additionally, manual computations are made in order to verify the outcomes of COMSOL. To determine which fin configuration will work best in practice, fins with various shapes and thicknesses will also be investigated. Data results reveal that adding cylindrical fins to the geometry demonstrate the best Stirling engine performance. The preliminary observations showed an increase in the heat flux up to 32X.

Keywords: Stirling engine; finned Stirling engines; enhanced Stirling engine; waste heat; finned Stirling engines, waste heat, cooling fin

Numerical investigation on thermal performance of PCM based triple tube heat exchangers using different types of fins

Bhim Kumar Choure[a] , Tanweer Alam[a] , Rakesh Kumar[a]

Mechanical Engineering Department, Indian Institute of Technology (Indian School of Mines), Dhanbad 826004, India

A numerical investigation has been made in the present research work to evaluate the thermal performance of phase change material (PCM) based triple tube heat exchanger (TTHX). A two dimensional (2-D) model of PCM based TTHX has been developed. Various shape of fins (rectangular, tree shape and double V-shape) were used to analyze the thermal performance of RT82 PCM based TTHX. The grid and time independency test has been performed. The developed model has been validated with experimental and numerical result. The variation in liquid fraction and average temperature of PCM were determined with the help of ANSYS-FLUENT 2021 R2 software. The rectangular fins were seen the most effective one. The reduction in melting time for rectangular and tree shape fins were found to be 19.50%, 16.32% compared to double V-shape fins to achieve the same temperature. The reduction in melting time for rectangular and double V shape fins were found to be 52%, 44% respectively compared to tree shape fins to achieve same liquid fraction. The effect of fin length penetration into the PCM in concentric radial direction have been seen to be more effective one compared to extending fin length in any other angular direction for same perimeter of fin length.

Keyword: Phase change material, triple tube heat exchanger, melting performance, fins

Study on different grouting materials for efficiency enhancement of geothermal energy systems

Anand Kumar[a], Tanweer Alam[a]

Mechanical Engineering Department, Indian Institute of Technology (Indian School of Mines) Dhanbad-826004, India

In recent times, there has been a significant surge in energy consumption, necessitating high-scale energy production to meet the escalating demand. Presently, approximately 80% of energy is derived from fossil fuels, contributing to the emission of greenhouse gases and various harmful toxins, thereby posing environmental threats. Researchers have shifted their focus to alternative energy sources, particularly renewable energy, as a means to mitigate carbon emissions. Geothermal energy, a form of renewable energy, holds promise in meeting future energy demands. Geothermal energy systems, employing Ground Source Heat Pumps (GSHPs) and Borehole Heat Exchangers (BHE), play a crucial role in heat production and power generation. The performance of GSHPs and BHE is augmented by the use of surrounding grout materials. This study assesses various types of grouting materials, comparing their thermo-physical properties and performance. Grout materials such as cement, bentonite, and silica sand are commonly used. However, contemporary practices involve the use of novel and enhanced grout materials. Controlled Low Strength Material (CLSM) and Phase Change Material (PCM) are notable examples employed to elevate the heat transfer rate. Literature indicates that CLSM and PCM emerge as optimal choices for grout materials due to their superior thermo-physical properties compared to conventional options.

Keywords: Geothermal energy systems; Grouting materials; Controlled Low-Strength Material; Phase Change Materials

Two phase flow modelling of a bubble impact on flat plate

Sanyukta Palatkar[a], Swapnadip De Chowdhury[b]

[a]Department of Ocean Engineering and Naval Architecture,Indian Institute of Technology Kharagpur,Khragpur,West Bengal 721302,India
[b]Department of Ocean Engineering and Naval Architecture,Indian Institute of Technology Kharagpur,Khragpur,West Bengal 721302,India

We conduct two phase numerical simulations to investigate the impact pressures on flat plate due to a rising bubble in a fully bounded fluid domain. The purpose of these investigations is to explore the feasibility in simulating the impact of the cavitating bubble while collapsing on plates which may be kept to prevent the bubble signatures on free surface. This may add to the stealth capabilities of high-speed marine crafts. Two standard benchmark problems are considered to verify the two-phase numerical simulations: for the Rayleigh Taylor instability problem, we compare the results with fully incompressible SPH simulations and the collapsing of a cavitating bubble in an unbounded fluid domain where we compare the numerical solution against the analytical solution predicted by the Rayleigh-Plesset equation. Numerical simulations for the bubble impact on the plates by the verified CFD model reveals that the rising bubble may impart impact type loading at the smallest initial depth of the bubble below the plate.

Keywords: Cavitation Bubbles; VOF; Rayleigh-Taylor instability; Rayleigh-Plesset; Plate

Analysis of Combustion Characteristics of Can-Type and Annular-Type Combustion Chamber

D. Smitha[a] and M. R. S. Satyanarayana[a]

Department of Mechanical Engineering, Gitam University, Visakhapatnam 530045, Andhra Pradesh, India

High performance is always desired for the propulsion system and the assortment of a better combustion type is also a vital part of the propulsion system. The analysis of different designs of combustion chambers and comparison of their combustion characteristics. In this paper, the assessment of Can-type and Annular-type combustion chambers and the design developed with SOLIDWORKS software and analysis done with ANSYS FLUENT. Parametric analysis is used to acquire the sizing of the combustion chamber.

Keywords: Can-Type combustion chamber, Annular-Type combustion chamber, ANSYS, SOLIDWORKS

Numerical Studies of Solid Rocket Propellant PMMA-PBAN-ALF_3-Nitrocellulose-Difluoramine

Sachin Srivastavaa*, G. Hamshitha[b] , V. Jagannath[b] , Gaurav Kothenty[b] , Subham Haldar[c] , Ajith Raj Rajendran[d]

[a]Department of Aerospace Engineering, Uttaranchal Institute of Technology, Uttaranchal University, Dehradun 248007, Uttarakhand, India
[b]Department of Aeronautical Engineering, MRCET, Hyderabad-500100, Telangana, India
[c]Department of Aerospace Engineering, University of Pisa, Pisa, Italy
[d]Division of Aerospace Engineering, Karunya Institute of Technology & Sciences, Coimbatore, Tamilnadu

The aim of the new investigation is the evaluation of solid rocket Propellant. In which performing numerical analysis of solid rocket propellant and calculating the constraints involved as derived from thrust i.e. velocity involved in the usage of solid rocket motor. The purpose of this paper is to show the viability of solid rocket propellant for modern and future applications. Solid rocket motors are simple in design and fabricating. The need for a propellant that produces the same specific impulse as another chemical rocket engine. So it is needed to have a solid rocket motor with an enhanced combustion characteristic at a reasonable O/F ratio. For a healthy ecology, a low-cost, highly effective, and environmentally friendly propellant is constantly desired. As a superior alternative among the already used propellants in the sector, the paper suggested fuel PolyMethyl MethAcrylate (PMMA), PolyButadiene AcryloNitrile (PBAN) CoPolymer, and Aluminum hydride (AlH3), and oxidizers such as Nitrocellulose and Difluoramine. The desired O/F ratio for the technical criteria was found to result in a high combustion characteristic. With the features discovered through numerical and analytical research, we have suggested a design for the SRM that includes post-combustion, which enhances the reaction and creates the innovative elements of the existing SRM.

Keywords: Solid Rocket Motor, PolyMethyl MethAcrylate, PolyButadiene AcryloNitrile, Aluminum Hydride, Nitrocellulose, Difluoramine.

Estimation of energy available in rain droplets from meteorological rainfall data

Anushka Gupta[a], Vimalraj S R[a], Visakh Vaikuntanathan[a*]

[a]Department of Mechanical Engineering, Shiv Nadar Institution of Eminence (deemed to be University), Old NH-91, Tehsil Dadri, Greater Noida 201314, India *Corresponding author: visakh.v@snu.edu.in; visakh@gmail.com

Even though the amount of rainfall is well documented in the reports of the India Meteorological Department (IMD), the rainfall energy available in different parts of India is unknown to the best of our knowledge. This paper reports a theoretical framework to convert the meteorological rainfall data to rainfall energy. The amount of rainfall over the south-west Monsoon period of 2022 is first converted to an average rainfall rate. Using the information on rain droplet size distribution for a rainfall of a given rainfall rate, the total energy contained in the rainfall is derived. In this, both the kinetic and surface energies of rain droplets of different sizes are considered. Based on the developed theoretical framework, rainfall map of India is converted to rainfall energy map showing states/union territories with maximum and minimum rainfall energy. Furthermore, for selected states in India, the districts showing maximum and minimum rainfall energy are also highlighted. The framework developed in the present study would help attempts to develop efficient rain energy harvesting systems.

Keywords: rainfall energy; rainfall rate; IMD; raindrop size; terminal speed; droplet energy

2D wake transition of NACA 0012 airfoil in soap film for low Reynolds number

Harish Wathore[a] , Kamal Poddar[a] and Alakesh Chandra Mandal[a]

[a]Department of Aerospace Engineering, IIT Kanpur, Kanpur-208016, India

This study presents experimental investigations conducted on the flow past a NACA 0012 airfoil within a horizontal soap film tunnel. Soap film experiments were performed over a range of angles of attack (0^o to 20^o) at Reynolds numbers ~ 3000 and 4500. The flow was visualized using monochromatic light and a DSLR camera employing the interference technique. The research focuses on elucidating the two-dimensional wake transition of the NACA 0012 airfoil under different Reynolds numbers and angles of attack. Changes in the trailing flow were primarily influenced by variations in the Reynolds number and angle of attack. The visualization process revealed various wake states and structures. Vortex shedding occurred in an alternating shedding mode at smaller attack angles, transitioning to alternate pair vortex shedding at intermediate angles. These vortex forms became more turbulent and chaotic, shedding at greater angles of attack. The findings indicate that wake transition occurred at lower angles of attack and earlier in time with increasing Reynolds numbers.

Keywords: Soap film tunnel; monochromatic light; wake transition; PIV; alternate pair vortex shedding; interference technique.

Comparative Study on DI Diesel Engine Performance and Emissions using HCP microalgae biodiesel, Mouha oil biodiesel, Rubber seed oil biodiesel and Petroleum diesel

Jagadev kumar and A Patil

Department of Mechanical Engg. Faculty of Engineering and Technology exclusively for women, Sharnbasva University, Kalaburagi (Karnataka) , India

Energy consumption of the world is increasing hence the prices of fuels are increasing. Fossil fuels are the major source of energy but they are depleting in nature and they may exhaust in future. So we cannot depend on these sources. So biofuels can be considered as substitute for fossil fuels, hence lot of researchers are doing extensive research on biofuels. But the difficulty associated with biofuels is that they leads to competition with food. First generation biofuels includes the plants which are consumable so they directly leads to food competition. Second generation biofuels are non consumable but they requires fertile land for their growth thus they indirectly leads to food crisis and third generation biofuels like algae doesn't require fertile land for their growth they can be grown in water bodies like pond,river, sea etc and also can be grown in municipal sewage water and industrial waste outlet water. Another problem with the usage of biofuels is their lower performance compared to diesels. Hence it is important to study the performance of different biofuels to find suitable one in terms of food crisis and better engine performance. Accordingly in this study the performance and emissions characteristics of Rubber seed oil, Mohua oil and microalgae oil biodiesels are studied. The results of this comparison study carried with HCP-B20, MO-B20 and RSO-B20 biodiesel blends indicates that the performance of HCP-B20 blend is better and emissions are lower compared to MO-B20 and RSO-B20 blends.

Keywords: Biodiesel; Biofuel ;Heterotrophic; Load; Microalgae.

Numerical Investigation of Droplet Spreading on Curved Surface Using Lattice Boltzmann Method

Ganesh Sahadeo Meshram[a], Gloria Biswala[a]

[a]Mechanical Engineering, IIT Kharagpur, Kharagpur 721302, India

Using a two-dimensional (2D) pseudo-potential multiphase lattice Boltzmann method with a D2Q9 model, we present a numerical investigation of the impact of solid-fluid interaction strength on the wettability and hydrophobicity of curved surfaces for a range of solid-fluid interaction strengths, from -1.30 to -2.90. To assess the correctness of the current numerical model and determine the contact angle, a simulation of the equilibrium state of a water droplet on a flat surface is first taken into consideration for different interaction parameters. Contact angles have been subjectively confirmed with the prior findings for a range of interaction strength values. The semi-circular concave surfaces with varying curvature radii were then superimposed. Within the range of 200x200 lattice units, the droplets' radii vary from 30 to 60 lu. The study demonstrates that a significant increase in the solid-fluid interaction parameter of the curved surfaces results in a larger contact area between the solid walls and water droplets, which in turn enhances the hydrophobicity. By calculating the contact angle between the solid-liquid and fluid-vapor interface, the hydrophobicity is examined.

Keywords: Lattice Boltzmann method; D2Q9; Droplet spreading; contact angle.

Effect Of Temperature On Low Rotational Speed Performance Of External Gear Pump

Nadakumar P S[a] , Rohini kumar B[a]

[a]Department of Mechanical Engineering, National Institute of Technology, Calicut, Postcode: 673601, Kerala, India

This paper investigates the influence of temperature on the performance of external gear pumps operating at low speeds. The study focuses on key performance parameters such as volumetric efficiency, leakage, and minimum speed limits of pump. Through systematic experiments, volumetric efficiency was correlated with speed, temperature, and pressure. Additionally, the influence of mixed lubrication on torque loss under low-speed conditions is also investigated. The lubrication zone is carefully distinguished through Stribeck measurements. The research also delves into the determination of critical speed and zero delivery speed under varying pressure conditions, providing a comprehensive understanding of the pump's performance and minimum rotational speed limits at different operating pressures. A detailed analysis of the temperature's impact on the minimum rotational speed limit forms a significant contribution of this paper understanding the correlation between temperature, rotational speed, and volumetric efficiency. The findings presented in this study contribute to external gear pump Performance under the challenging low speed, high-temperature conditions with practical implications for optimizing the pump performance in real-world applications.

Keywords: External gear pump; Leakage; Low-speed; Volumetric efficiency; Temperature.

A Review on Cushioning Characteristics in High-Speed Hydraulic Actuators

A S Kanaskar[a] , Dr S.N. Sapali[b] , Dr V.K. Haribhakta[c] , Dr SC Sati[d]

[a] Scientist, R&DE(E), DRDO, PUNE 411015, INDIA
[b] Director, Dept of Technology, Shivaji University Kolhapur 416004, INDIA
[c] Associate Prof, COEP Technological University, PUNE 411005, INDIA,
[d] Distinguished Scientist (Retd.) DRDO PUNE 411015, INDIA

High speed actuators are an integral part of the fluid power driven systems and cylinder cushioning requirement in these actuators is one of the important aspects. Cushioning is imperative in any high-speed actuator as it not only increases its life but also gives operating comfort to the operator. Fluid power driven systems are very common in many defense and industrial applications due to their high force/torque and power density. This review paper brings out the available literature in the field of hydraulic cylinder cushioning. Literature, consisting mainly of journal articles, research papers and other media contents is analyzed for the key parameters of research in this field. The classification of cushioning methodology by different researchers is projected. The methodologies of modelling and simulation work carried out by various authors are analyzed. Thereafter the optimization studies done for the cushion piston design in various literature is discussed. Lastly, experimental validation strategies as employed in certain papers are also brought out. Experimental study, and validation of non-conventional cushioning methodology is discussed, which is not much evidenced in the literature.Click here and insert your abstract text.

Keywords: Review Paper; Hydraulic Cylinder Cushioning; Fluid Engineering; Conventional Cushioning, High speed Actuators; Fluid Dynamics

Integrated Furnace Modeling and Circulation Analysis of Natural Circulation Hybrid Boiler

Atul M. Elgandelwar[a]

[a] MIT World Peace University, Pune

Circulation ratio of hybrid boiler with natural circulation are studied under different operating conditions. Heat flux, boiling heat transfer, nucleate boiling, and tube overheating are studied. Examine pressure, heat flow, liquid velocity, dryness fraction, void fraction and circulation ratio. This work creates a model for hybrid boiler natural circulation and tests the boiler design for various failure situations. The circulation ratio determines crucial heat flow and skin temperature in a natural circulation boiler. Boiler pressure, liquid velocity, and maximum heat flow effect circulation ratio. This study aids natural circulation hybrid boiler construction. The study will provide an Excel-based flow distribution analysis tool for natural circulation boilers. Newton Raphson solves node and loop nonlinear equations in the model. Loop calculations for every evaporator tubes include pressure loss and driving force. Typical risers and downcomers use node equations. A unique connection matrix links loop and node equations for each branch flow. The model is verified by a novel indirect technique. Experimental data matches model within 3.57 % absolute error. It builds, analyses and optimizes natural circulation networks.

Effect of septal correction on nasal aerodynamics – An Invitro Experimental and Numerical investigation

Kartika Chandra Tripathy[a], Ajay Bhandari[a*]
[a]Biofluids Research Lab, Department of Mechanical Engineering, Indian Institute of Technology (Indian School of Mines), Dhanbad 826004, India

Septal deviation is a common nasal airway obstruction that significantly affects airflow patterns, leading to breathing difficulties. A surgical procedure called septoplasty is often performed to treat this. The septoplasty's success rate depends on the post-surgery airflow distributions. Due to the complex anatomical features of the nasal cavity, even though septoplasty is a promising treatment, it does not always guarantee symptom improvements. In this regard, it is crucial to investigate the airflow patterns for pre- and post-septal correction before the surgery is planned. This article performs experimental and computational studies on S-shaped septal deviation anatomy and the virtual corrected model. The experiments are performed on the 3D-printed transparent nasal cavity, followed by the numerical simulations. The experimental pressure drops are measured by attaching a differential U-tube manometer through the probes in the 3D-printed models. The results show that there is a significant reduction in pressure drop (up to 2-4 Pa) in between the deviated and corrected models; in addition to total pressure drop, the local pressure drops within various regions are also evaluated, further disturbed airflow becomes relatively uniform, and there is a significant decrease in the average and local shear stresses in the corrected models. The results collectively signify a noticeable improvement in the flow patterns after septoplasty, and the breathing difficulties are mitigated after corrections.

Keywords: Nasal cavity; Numerical simulations; Pressure drop; Shear stress

HEAT TRANSFER THROUGH A HEATED ROD PLACED IN DIFFERENT ARRANGEMENTS

Kunal[a] , Sushil Kumar Dhiman[b]

[a,b] Department of Mechanical Engineering, Birla Institute of Technology, Mesra, Ranchi-835215, India

The present work experimentally analyses the surface heat transfer from a thermally active copper rod of circular cross-section placed in different arrangement of dummy rods. All the rods are arranged in cross-flow. Wind-tunnel experiments for nine different arrangements of heated rod at a Reynold's number of 12600 were investigated. Temperature variation, heat transfer and vertical velocity distribution downstream of the cylinder were studied. Higher heat transfer rate was obtained when the heated rod was placed in downstream position due to the rubbing action of vortex shedding phenomena of upstream rod. Maximum heat transfer rate was reported in tandem arrangement of rod with the thermally active cylinder placed in the downstream row. The wall effect becomes dominant near the wall region and affects the flow and heat transfer in in the vicinity of wall. Pressure and logarithmic value of temperature has been reported at suitable positions to explain the thermofluid physics.

Keywords: vortex shedding, cross-flow arrangement, Reynold's number, heat transfer rate

Parametric Investigation of PIG Travel in Subsea Pipelines – A CFD Approach

Shubham T. Chavan1 , Sarvesh M. Darade2 , Rahul Tarte3 , Samarth Patwardhan4 , Parag Paranjape5 , Spruha Dhavale6 , Anita Nene7

[2 , 3, 7] Department of Mechanical Engineering, Dr. Vishwanath Karad MIT World Peace University

[1, 4] Department of Petroleum Engineering, Dr. Vishwanath Karad MIT World Peace University

[6] Center for Subsea Engineering Research, Dr. Vishwanath Karad MIT World Peace University

[5] Aker Powergas Pvt. Ltd.

Pipeline pigging is an essential and crucial maintenance operation to clear off depositions and precipitates for oil and gas industry. A periodic pigged pipeline is 70% more capable to transport oil and gas as compared to an un-pigged line. Being a critical operation, awareness of operating conditions of the pigging tool is vital before employing the pig under real-time conditions. The present study investigates the critical parameters that define a pig operation in a pipeline, from a CFD perspective. A 3.8-inch pig is developed to satisfy the requirement for pigging operation of a 240.1 ft long sub-sea pipeline network. The objective of present paper is to study the dynamic behavior of pig and therefore the pipeline fluid as water was analyzed considering it as a fluid-structure interaction (FSI) system using ANSYS™ Fluent 2021 R1 adopting finite volume CFD method. The study investigates dynamics of PIG and seeks to scrutinize the critical flow parameters for movement of transport fluid and PIG by incorporating the governing equations for momentum, continuity, and dynamic balancing to approximate the CFD model. The study results were validated against the results approximated by solving the model with analytical calculations through numerical methods. The term error and factor of safety (FOS) which determines the offset maintained by the magnitude of velocity over minimum required, this measurands are defined for validation of computational model. The differential pressure of 54 psi is maintained across the pig at the inlet boundary. The results demonstrate that the minimum velocity acquired by the pig is 4.5 m/s in the vertical section travelling against the gravity, which is higher than the minimum velocity required i.e., 2.6 m/s, maintaining the safety factor above 1.73. Also, the differential pressure across pig rises at bends and the velocity drops. The pressure at inlet section drops form 54 psi to 11 psi as the pig must over overcome the static and fluid friction associated with pig and pipeline interaction and the velocity. The parametric study in this work demonstrates safe pig travel throughout the pipeline as no flow reversal for pig was observed, which was validated through analytical calculations.

Keywords: Pipeline inspection gauge (PIG), CFD, Ansys Fluent, Pigging, Subsea Pipeline, Petroleum operations, Numerical method.

CFD simulation of a Lithium-Ion battery pack with air-cooling and water-cooling types of battery thermal management systems under high discharge rate condition

Devesh Kumar*, Ram Babu Gupta, Tanmay Dutta
Dept. of Mechanical Engineering, IIT (ISM) Dhanbad, Dhanbad – 826004, India
*Corresponding Author Email ID: deveshkumar97@gmail.com, Phone: +91-9873753433

Battery temperature has a direct impact on the performance, safety, and lifespan of electric vehicles. Battery thermal management system plays a vital role in controlling the battery temperature. In the present study, a 3D CFD model is developed for a battery pack consisting of 4 numbers of 26650 cylindrical Li-Ion cells, which are arranged in a 2x2 configuration. A thin aluminum jacket surrounds each cell, around which coolant flow takes place. Heat generation in the cells is modelled by a constant value of the volumetric heat source. CFD analyses are conducted for 5C discharge rate considering water-cooling, air-cooling, and no cooling of the battery. Results show that the battery operating with high discharge rate is highly susceptible to thermal runaway under no cooling and air-cooling conditions. However, water cooling of the battery ensures complete safety and high-performing operation along with long life of the battery even with a very low value of inlet water velocity. Therefore, for high C-rate battery operation, this study finds the air-cooling system completely inappropriate, while the water-cooling type BTMS is the most appropriate.

Keywords: Battery thermal management, Li-ion battery, CFD modelling, Air-cooling, Liquid cooling

Alternative brain cancer treatment using tumor treating fields: A computational approach.

Sourav Choudhury[a] , Prasanth Bokam[b] , Ajay Bhandari[a*]
[a]Biofluids Research Lab, Department of Mechanical Engineering, Indian Institute of Technology (Indian School of Mines), Dhanbad 826004, India
[b]Ansys France

Brain cancers encompass a range of primary and metastatic diseases, with glioblastoma (GBM) the most common and deadly primary brain tumor in adults. Standard-of-care for patients with glioblastoma involves surgery followed by radiation or temozolomide (TMZ) chemotherapy. These traditional treatment protocols are still far from reaching clinical expectations. With the more recent development of tumor treating fields (TTF), which involves delivery of low-intensity (1–3 V/cm), intermediate-frequency (100–300 kHz), alternating electric fields have shown promising results to treat brain tumors. Nevertheless, there is still a lack of understanding of how the mechanisms of TTF will affect chemotherapy drug diffusion. Therefore, this article performs a numerical simulation of TTFields on the sphere model, which is a simplified model of the actual brain. The electric potentials were applied to the transducers and placed at the scalp's outer surface. The results explored the qualitative and quantitative distribution of the electric field and temperatures. It was observed that maximum temperature occurs at the junction of the transducer and scalp due to higher joule heating at those locations. The significant thermal penetration up to the brain's outer surface is observed. Similarly, the penetration of the electric field varies as a maximum at the scalp and decreases towards the inner regions. The results can be considered as the collective effect of the different electrical and thermal tissue properties at the different layers, the strength of the joule heating source, and the blood perfusion. The results will help predict the effectiveness of the drug diffusion inside the tumor and the treatment efficacy.

Keywords: Glioblastoma, Tumor treating fields (TTFields), finite volume methods, electric field, temperature

Design of a Tailsitter VTOL UAV and Investigating the Aerodynamic Performance using CFD Techniques

Kiran R Pande[a]*, Akarsh Choudhary[a], Aditya Sharma[a], Pramendr[a] Kumar Bajpai[a]
[a]Department of Mechanical Engineering, Netaji Subhas University of Technology, Dwarka, New Delhi 110078, India

Due to the increasing demand for versatility, VTOL UAVs, particularly tailsitters, have gained significant interest. These UAVs offer adaptability, operating without runways and adjusting flight modes for diverse environments. Tailsitters, a specific type, combine vertical takeoff and landing with forward flight efficiency, making them ideal for maneuverability in tight spaces and long-range flights. The article discusses a delta wing design with non-vectorized twin rotors, crafted using SolidWorks CAD software and analyzed with ANSYS V2024R1 Fluent CFD software. Aerodynamic features were examined using it including lift and drag capabilities stall characteristics, flight effectiveness, maneuverability and stability, against Angles of Attack (AoA). This comprehensive study under different conditions enables informed design decisions and performance enhancements.

Keywords: CFD Simulation, VTOL, UAV, Tailsitter

Investigating the Thermal Modeling of a Parabolic Trough Solar Collector using Therminol VP-1 oil using Al2O3 nanoparticle as Heat Transfer Fluid

Vijay Kumar Pandey[a], Om Prakash[a] , Lakhbir Singh Brar [a] , Manjeet[a] , Anil Kumar [b]

[1]Department of Mechanical Engineering, Birla Institute of Technology, Mesra, Ranchi, India, 835215
[2]Department of Mechanical Engineering, Delhi Technological University, Delhi, India

This research investigates the effect of incorporating nanoparticles into heat transfer fluids to enhance thermal efficiency in parabolic trough solar collectors (PTSCs), focusing on Therminol VP-1 oil as the base fluid and varying the volume fractions of Al2O3 nanoparticles. A Thermo –mathematical modeling under steady state for PTSC was performed in this study. The results are validated using published data from two different heat transfer fluids, and a maximum relative error of less than 10% was obtained. The model estimates thermal efficiency when heat transfer fluid such as water, Therminol VP-1 oil, and nanofluids with different concentrations of Al2O3 nanoparticles is used. According to the results, the thermal efficiency of the PTSC increases when the nanofluids contain a higher volume fraction of nanoparticles. Thermal oil has the lowest efficiency compared to nano fluids and a maximum efficiency of 76%. With a volume fraction of 0.01, 0.02, 0.03, and 0.04, the thermal efficiency is 76.25%, 77.00%, 78.96, and 80%, respectively. The nanofluid works at low-pressure levels in PTSC compared to pressurized water. This study provides valuable insights into enhancing thermal efficiency in parabolic trough solar collectors, which could advance solar energy technologies. The research may aid global efforts in developing sustainable solar energy systems promoting renewable energy adoption.

Keywords: Nanofluid, Heat transfer in steady state, Therminol VP-1, solar energy, Parabolic trough solar collector

Effect of aspect ratio on laminar flow in an internal shear driven system of 2-D lid-driven cavity

Aman Vardhan Verma[a] , Shravan Kumar Mishra[b] , Subhankar Sen[b]
[a,b] Department of Mechanical Engineering, Indian Institute of Technology (Indian School of Mines), Dhanbad 826004, India

A computational analysis for steady, incompressible and two-dimensional fluid under a shear driven system of Liddriven cavity, has been performed using artificial compressibility methodology for finite volume discretization. A wide range of Reynolds number from 10 to 3200 has been presented for square and rectangular domain with various aspect ratios (i.e. *AR*=1, 2, 4, 0.5, 0.25). Using centerline velocity and shear stress plots, effect of aspect ratio and Reynolds number are realized on boundary layer thickness and fluid momentum diffusion property. Surface vorticity are plotted to prove existence of various small bubbles near cavity wall, which were undetected by streamline contours. By analyzing different streamline contours, three different instances of cat's eye vortices are found and presented.

Keywords: Lid-driven Cavity; Artificial Compressibility; Aspect ratio; cat's eye vortex; surface vorticity; Centerline shear stress.

Optimization of probe positioning in radiofrequency ablation of heterogeneous human brain tumor: A numerical heat transfer study

Dhruvajyoti Roya , Prasanth Bokamb , Ajay Bhandaria*

[a]Biofluids Research Lab, Department of Mechanical Engineering, Indian Institute of Technology (Indian School of Mines), Dhanbad 826004, India
[b]Ansys France

Radiofrequency ablation (RFA) is a cancer treatment technique known for its precision and localized effects. However, its potential in clinical settings has not yet been fully explored. Limited treatment effectiveness is often attributed to inadequate tissue destruction, which can be significantly influenced by blood flow and vascular density. Inspired by this, the present investigation aims to numerically optimize the location of the RF probe in heterogeneous human brain tumor and investigate the effect of heterogeneous vascular volume fraction and blood perfusion on the RF treatment efficacy. Patient-specific information is extracted from Dynamic contrast-enhanced magnetic resonance imaging (DCE-MRI) data, which is incorporated in the bioheat transfer model formulated in OpenFOAM. The numerical results indicate that placing the RFA probe where blood perfusion and vascular volume fractions are low leads to more effective treatment, as tissue temperature and ablation volume increase by an average of 4.1% and 5.4%, respectively, compared to locations where both these parameters are high. These in silico findings may offer valuable insights for clinicians and surgeons to fine-tune RFA probe locations tailored to individual patients, potentially enhancing treatment effectiveness.

Keywords: RFA; Human brain tumors; Heterogeneous blood perfusion; Heterogeneous vascular volume fraction; DCE-MRI;

Fluid Flow CFD Analysis of Orifice Meter

[a] Kajal, [a*] Vishakha Baghel
[a] Department of Mechanical Engineering, Faculty of Engineering, Dayalbagh Educational Institute, Dayalbagh Agra

The research investigates the operational intricacies of orifice meters using Computational Fluid Dynamics (CFD) simulations. The present work compasses in-depth research on orifice meters, examining their workings, advantages, and limitations. The numerical simulation of fluid flow through the orifice meter is also performed using the ANSYS Fluent® inculcating turbulence model [1]. The CFD simulation results are validated with experimental data to ensure that the results are credible. The research aim is to predict flow rates at different points of orifice geometries and different operating conditions [2]. The study also explores how fluid properties, such as density and viscosity, affect orifice meter performance [3]. Firstly, a geometry is taken and is made in SOLIDWORKS then it is imported to ANSYS FLUENT® here a structural meshing with mesh type linear mesh is used. After meshing different plots are produced after simulation using an inlet velocity of 0.755 ms^{-1}. After successful simulation we got the results in the form of different charts and graphs which helped us know better about the flow of liquid in the orifice meter and again on comparing the velocity profile and velocity contour found in our simulation with that of a research paper published earlier we can see the similarity in both and hence the research paper has been validated.

Keywords: Orifice Meter; Fluid Flow Measurement; Computational Fluid Dynamics; Velocity Profile; Turbulence Model.

Advancements in various methods for turbulence prediction in transitional flow

Ria P Dey[a] , Bhavya Saxena[a] , Shubhangi Sundaria[a] , Lipakshi Negi[a] , Anurag Sharma[a] , Sanya Karotiya[a] , Prachi[a] , Ullas AJ[a] , Rajesh Kumar Saluja[a]

[a]Amity Institute of Aerospace Engineering, Amity University, Sector 125, Noida, Uttar Pradesh, India, 201313

Transitional flow is difficult to navigate because of its unpredictability and this makes transitional flow difficult to control. A major cause of this unpredictability is turbulent spots. The turbulent spots are small, dense vortices in which the velocity of the flow increases suddenly. They have high swirling strength, and they originate from hair-pin packets. These spots are generated locally and persist till the fully turbulent environment, and they are persistent with a systematic variation of detection threshold level. A lot of Artificial Intelligence, specifically Machine Learning techniques, have been implemented in analyzing flow characteristics. In the paper, different algorithms like CNN, RNN, SVM and KNN have been studied to design a predictive model for turbulent spots. Their accuracies have been researched, compared and thereafter, the best algorithm has been selected for further testing. This paper provides insight into different algorithms that can be used for predictive modeling of turbulent spots and reviews the use of KNN and RNN algorithms. Physics-informed machine learning (PIML) combines domain knowledge with machine learning (ML) methods, resulting in im-proved data efficiency and more consistent predictions. This opens possibilities for augmenting—and replacing—high-fidelity numerical simulations of complicated turbulent flows, which are frequently expensive due to the need for high temporal and spatial resolution.

Keywords: predictive modelling; KNN Algorithm; Turbulent spots, K-Means, f (frequency)

Design, Fabrication and Aerodynamic Analysis of a RC Aeroplane

Durjoy Chandra Shil[a] , Md Tanvir Ahmed[a] , Md Robin Mia[a`] , Nilotpala Bej[a]

[a]Department of Mechanical Engineering, Kalinga Insitute of Industrial Technology, Bhubaneswar 751024, India

This article presents the design and development process of a radio-controlled (RC) airplane, focusing on the integration of aerodynamics, structural engineering, and control systems. An internal receiver and a hand-held transmitter allow a radio-controlled (model) aircraft, often known as an RC plane or aircraft, to be operated remotely. The design of a 2.4 GHz-operating, lightweight, electronic glider is the basis of this paper. The objective is to create a high-performance, stable, and maneuverable RC airplane suitable for recreational and hobbyist use. Since the used components are freely available in markets and do not require user programming, this paper did not focus on them. In this study, computational fluid dynamic (CFD) analysis was used to discuss wing shape, aspect ratio, and control surface configuration to optimize lift, drag, and stability. N10 airfoil was considered for wing design to get the best possible aerodynamic properties. At top speed, Structural engineering principles are applied to ensure a lightweight yet robust air-frame capable of withstanding flight forces. The control system incorporates precise and responsive control surfaces, along with flight electronics, to enable smooth and accurate flight control. The article also discusses the importance of data analysis in validating and refining the design. Overall, this article provides valuable insights and guidelines for the design and development of RC airplanes, fostering advancements in this exciting field of aviation.

Keywords: RC Airplane; Wingspan; Air foil; Angle of Attack

Visualizing the Bubble Behavior during Pool Boiling on a Rough Vertical tube: A Comprehensive Experimental study

Bibhu Bhusan Sha[a] and Mihir Kumar Das[a]
[a]School of Mechanical Science, IIT Bhubaneswar, Odisha-752050, India

Pool boiling over vertical tube is vastly experienced in nuclear industry. This study shows the bubble dynamics over a vertical tube during saturated pool boiling. The tube is composed of smooth and rough surfaces. This study uses highspeed imaging to analyses the bubble dynamics of a combination of smooth and rough vertical tube which is compared with the purely smooth and rough tube. The presence of combination of smooth and rough qualities improves water boiling stability. Rough areas can serve as nucleation sites, minimalize the creation of vast, unstable vapour zones that could produce a boiling crisis. Smooth parts may help small bubbles cluster and deliver liquid to the boiling region.

Keywords: Vertical tube, pool boiling, high-speed videography, smooth and rough surface, pro-analyst software

Analysis of ignition delay period by using Arrhenius Equation for a dual fuel diesel engine using hydrogen as a secondary fuel

Harsh Vardhan [1*], VNL Durga [1] , Chandra Bhushan Kumar [2]

[1]Department of Mechanical Engineering, Sarala Birla University, Ranchi 835205, India
[2]Department of Mechanical Engineering, University College of Engineering and Technology (UCET), Hazaribag, India

Real-time prediction of in-cylinder combustion parameters is very important for robust combustion control in any internal combustion engine. The ignition delay period in a compression ignition (CI) engine is a very essential combustion parameter that influences subsequent combustion processes and engine performance. The ignition delay in diesel engine working on dual fuel mode is examined, while employing the hydrogen as gaseous fuel. It is shown that the changes due to gaseous fuel admission in the oxygen concentration, volumetric efficiency, in-cylinder pressure, temperature and ignition delay period. In this regards the experiments were conducted on single cylinder four stroke direct ignition diesel engine, power 3.50 KW at constant speed 1500 rpm Kirloskar model TV1 with base fuel as diesel and hydrogen as secondary fuel. Experiments were conducted to measure the ignition delay of the diesel engine working with dual fuel mode at low (2%), medium (36%) and high (69%) load conditions.

The experimental values of ignition delay were compared with proposed correlation which was predicted on the basis of Arrhenius equation for ignition delay by considering oxygen concentration, in-cylinder pressure and cylinder temperature, activation energy and variations are found in between 0 % to 18.12 %. Furthermore, with substitution of pilot fuel diesel with hydrogen fuel, oxygen concentration and volumetric efficiency decrease with increase in hydrogen fuel percentage at low, medium and high load conditions. The hydrogen substitution also alters the in-cylinder pressure of the engine. At low and medium load condition, peak value of in- cylinder pressure is decreases, while it increases with higher substitution of hydrogen (more than 26%) at high load conditions. This might be due to the variation of heat release rate, ignition delay, temperature and pre-ignition reaction rate of hydrogen fuel.

Keywords: Hydrogen fuel, Duel-Fuel diesel engine, Ignition delay, oxygen concentration

Design for a Horizontal Ground Heat Exchanger Using a Centrifugal Pump

Sital Roy, Arijit kundu
Jalpaiguri Government Engineering College, Jalpaiguri-735102,West Bengal

By the year 2040, India intends to meet its increasing energy needs. The nation must maximize the environmentally beneficial use of all available energy sources, both conventional and renewable, in order to achieve sustainable growth. Increasing the proportion of green and renewable energy sources like geothermal energy, solar power, wind energy, etc. in the energy portfolio is crucial because India is the third-largest emitter of greenhouse gases in the world. Geothermal energy is still untapped, despite a notable rise in the generation of solar and wind energy. Because of the high concentration of heat-producing granites that are found all over the country, India has many geothermal regions with the potential to produce large amounts of heat. The current study suggests building a straightforward horizontal heat exchanger. The pipes for this heat exchanger model are supposed to be buried. Furthermore, heat is transmitted from the soil to the pipes and ultimately to the working fluid, which is water. The characteristics of the soil and working fluid (water), the type of pipe, the overall length of the pipe, the pump's power, and the working depth all affect how much heat is transferred. The ground heat exchanger's pumping power needs are also taken into consideration by the model. The findings show that, if the number of parallel loops is increased under the same conditions and with the same total pipe length, the variation in ground temperature has a considerable impact on both the overall pipe length and the pump power.

Keywords: Design; Heat Exchanger; Centrifugal Pump; soil conductivity.

Comparison of Latent Heat Storage Unit (LHSU) Designs Utilized with Low-Temperature Solar Air and Water Heating Systems

Ashutosh Sharma[a] , Tej Singh[b] , Ranchan Chauhan[c]
[a]College of Science and Engineering, James Cook University, Townsville, Queensland, Australia
[b]Savaria Institute of Technology, Faculty of Informatics, ELTE E ̈otv ̈os Lor ́and University, Szombathely 9700, Hungary
[c]Department of Mechanical Engineering, Dr. B. R. Ambedkar National Institute of Technology, Jalandhar, Punjab, India

Low-temperature solar thermal utilization systems (STUS) are extensively utilized solar energy applications. Utilising thermal storage, particularly latent heat storage (LHS), may greatly lessen the issue of solar energy's intermittent availability. Either combining the collector-storage or attaching a separate latent heat storage unit (LHSU) with a solar collector can both be employed to integrate LHS with STUS. The present article compares the various designs of LHSU studied with STUS. The packed bed, shell, and tube designs have undergone significant research with solar water heating systems (SWHS) as well as solar air heating systems (SAHS). A comparison between shell and tube and packed bed designs in application with SAHS and SWHS has also been carried out to comparatively evaluate these two designs' design, operation, and performance. The thermal efficiency of both designs has been found to be approximately similar. However, the average thermal efficiency of LHSU in application with SWHS has been found to be 10-20% better than the SAHS. Thermal storage capacity has been determined to be much higher for the shell and tube design than the packed bed type. The thermal storage capacity per unit collector area for most designs lies in the range of 1000-10,000 kJ.m-2. The charging/discharging rate for the packed bed design has been found to be significantly better. Thereby, the shell and tube and packed bed designs can be used for applications requiring large storage capacity and quick charging/discharging, respectively.

Keywords: solar air heater; solar water heater; thermal storage; latent heat storage unit; PCM.

Thermohydraulic Performance Analysis of a Rectangular Fluid Flow Channel with a Variety of Turbulence Geometries

Ashutosh Sharma[a] , Ranchan Chauhan[b]
[a]College of Science and Engineering, James Cook University, Townsville, Queensland, Australia
[b]Department of Mechanical Engineering, Dr. B. R. Ambedkar National Institute of Technology, Jalandhar, Punjab, India

Most efficient utilization of energy is of prime concern for every engineering application. Solar air heater which is used for many applications and generally of rectangular cross section has very least effectiveness. Many different turbulent geometries have been explored in the literature, each one optimized to maximize the transfer of heat. Optimizing heat transmission by turbulence enhancement also increases the need for pumping power, which should be maintained to a minimum. In this investigation, three recently investigated turbulent geometries – S-shaped ribs, circular arcs with gaps, and protruded roughness – have been taken into account for the maximum rate of heat tranfer and the lowest friction factor at various Reynolds number levels. Heat transfer enhancement (HTE) is found maximum of 5.16 for S-shape ribs and friction factor enhancement (FFE) of 4.11 for circular arc with gaps from smooth rectangular duct. Thermo-hydraulic performance parameter (THPP) at different values of Reynolds number for all selected geometries also has been evaluated. For 3000-9000 range of Reynolds number protruded roughness provides maximum value of THPP from 1.83 to 2.56 and for higher values of Reynolds number (9000-21000) S-shape ribs gives maximum value of THPP from 2.86 to 3.75. Results reveal that protruded roughness is suitable for applications involving low flow rate and S-shape ribs for higher flow rates. The best suitable roughness geometry for different Reynolds number ranges could be selected with the help of this research.

Keywords: Rectangular air flow channel; turbulence geometries; Thermo-hydraulic Performance Parameter

Numerical Studies on Simultaneous Heating and Cooling of Phase Change Materials

Shyam Kumar Rajak[a] and Debasree Ghosh*

[a,*]Department of Chemical Engineering, Birla Institute of Technology Mesra, India

Phase Change Material (PCM) symbolizes a substance capable of releasing and absorbing significant energy at its phase transition temperature, thereby offering useful heating and cooling effects, respectively. The utilization of PCMs holds immense promise in storing thermal energy from low-temperature sources and waste heat as latent heat. The latent heat content within PCM surpasses that of sensible heat by a considerable margin, thus providing a substantial supply of latent heat that can meet a portion of the energy requirements for specific applications.This study presents a numerical simulation of the concurrent melting and solidification processes of various PCMs (RT50, RT27, RT35) and evaluates their suitability for continuous application. The proposed model comprises three concentric hollow cylinders, with ice occupying the outermost cylinder, PCM positioned in the middle, and the inner cylinder left vacant. The temperature of the isothermal inner cylinder wall is maintained higher to charge the PCM, ceasing the charging process once the liquid fraction approaches unity. To prevent this, ice in the outer annular space acts as a coolant, ensuring a sufficient liquid fraction of PCM for continuous charging. The simulation is conducted using Ansys-Fluent software, employing the enthalpy porosity model to simulate transport equations for both solid and liquid phases. The study shows that with the appropriate selection of PCM, a constant melt fraction of PCM can be maintained. These findings demonstrate the potential for PCM to support continuous operations and partially meet domestic power demands.

Keywords: Thermal energy storage, Phase Change Materials, Enthalpy porosity method, Charging and discharging

The effect of different subgrid-scale models on the flow field in cyclone separator using large-eddy simulations: A benchmark study

Dinesh Kumar[a] , Vikash Kumarb , Kailash Jha[a] , Lakhbir Singh Brar[c]
[a]Department of Mechanical Engineering, Indian Institute of Technology (Indian School of Mines), Dhanbad, India
[b]Department of Mechanical Engineering, Indian Institute of Information Technology, Design and Manufacturing, Kancheepuram, India
[c]Department of Mechanical Engineering, Birla Institute of Technology, Mesra, Ranchi, India

The cyclone separator is extensively utilized across industries to extract solid particles from gas streams. This study is focused on the steady and unsteady simulations of the Stairmand cyclone using large-eddy simulations aiming to assess the performance using different subgrid-scale models viz. standard Smagorinsky (Cs=1), dynamic Smagorinsky, Wall-Adapting Local Eddy-Viscosity (WALE), Wall-Modeled LES (WMLES), and dynamic kinetic energy models. The velocity profiles within the cyclone separator were analyzed under both steady and unsteady conditions, with reference to the Hoekstra experiment for validation and comparison. Velocity profiles inside the cyclone separator were inadequately predicted by the steady-state simulation, whereas the unsteady-state simulation yielded results more aligned with experimental values. The current study suggests that employing subgrid-scale LES models standard Smagorinsky and WMLES presents a better option for analyzing flow patterns.

Keywords: Cyclone separators; Large eddy simulation (LES); Unsteady simulation; Velocity fluctuations.

Performance analysis of Plate type Steam Condenser Using Nano-fluid

Vijay Kumar Pandey[a] , Om Prakash[a] , Lakhbir Singh Brar[a] , Ankur Kumar[a] ,Sandeep Shriwastava[a] , Anil Kumar [b]
[a]Department of Mechanical Engineering, Birla Institute of Technology, Mesra, Ranchi, India, 835215
[b]Department of Mechanical Engineering, Delhi Technological University, Delhi, India

This paper investigates the application of nanofluid coolant, specifically a mixture of water and aluminium dioxide (Al2O3) nanoparticles, to augment heat transfer efficiency within flat plate heat exchanger. Beginning with an introduction highlighting the pivotal role of heat exchanger in industrial processes, the study emphasizes the imperative for enhanced efficiency and reduced energy consumption. The system description delineates the components and functionality of flat plate heat exchanger, elucidating operational and experimental conditions including initial temperatures and subsequent temperature transitions during cooling. Key parameters such as effectiveness and overall heat transfer coefficient are identified, with an exploration of their variation when comparing water alone to nanofluid coolant. Comparative data showcases experimentally obtained effectiveness and heat transfer coefficient values for both coolant types, revealing a substantial enhancement in efficiency with nanofluid coolant. Like effectiveness for water was 18% while that for nanofluid was 35%. The future potential and broader applications of nanofluid coolant in heat transfer technologies, advocating for continued research to unlock further advancements and efficiencies in industrial processes. By this technology we can reduce water losses as well as reduce power consumption in plants.

Multi-objective optimization of fin and tube heat exchanger with diagonally aligned straight winglets for performance enhancement

Rishikesh Sharma[a] , Lakhbir Singh Brar[a] , Saurav Chakraborty[a] , Om Prakash[a] , D.P. Mishra[a]

[a]Department of Mechanical Engineering, Birla Institute of Technology, Mesra, Ranchi 835215, India

The present study is undertaken to enhance the performance of fin and tube heat exchangers equipped with vortex generators. A group of diagonally aligned three straight winglets (to act as a single entity) are placed over the alternate tubes. Three geometrical entities, viz., the height of winglets, the relative gap between the trailing edge of the three winglets and the radial distance of the trailing edge of the winglet closest to the tube – identified as the most influential ones – are subjected to optimization to enhance the thermal performance of the heat exchanger. The study utilizes the genetic algorithm approach for optimization with objectives to enhance the heat transfer (expressed by the Colburn factor) and minimize the frictional losses (expressed by the friction factor). The experiments have been designed using the Latin hypercube sampling technique consisting of 180 design points, and computational fluid dynamics has been used to generate the response against each set of design variables. The fluid domain is discretized using hexahedra mesh, while the solid domain is discretized using polyhedral mesh and thin mesher. The resulting data serves as the training data for the artificial neural network that serves as a surrogate model for the genetic algorithm (GA). With GA, a set of optimal data sets (or Pareto front points) have been generated over a wide range of values for the two dependent variables. The results indicate that the optimized models significantly increase the performance of the heat exchangers and largely outperform the ones reported previously. Also, these optimized models deliver an exceptionally good performance at the off-design conditions.

Keywords: Heat exchangers; Rectangular winglet; Artificial neural network (ANN); Latin hypercube sampling (LHS) space; Genetic algorithms (GA)

Study of Flow Regime and Flow Patterns in Inverted U Bend

Mushtaque Momin[a] , Mukesh Sharma[a]
[a]Birla Institute of Technology Mesra, Ranchi, India

The experimental investigation on liquid-liquid two-phase flow in an inverted U bend configuration revealed several significant findings. Inverted U bend predominantly exhibited droplet flow at lower superficial velocities of the continuous phase (V_{sk}) and higher superficial velocities of the dispersed phase (V_{sw}). Slug flow decreased as water velocity decreased, with wavy-stratified flow dominating at higher kerosene velocities. Under specific conditions, water slug flow was also observed in an inverted U bend. A novel flow pattern, sinuous core annular flow, was identified. wavy-stratified flow was more common in the inverted U bend. The minimum pressure drop observed in the U bend was 1228.8 Pa/m at V_{sk} = 0.13 m/s and V_{sw} = 0.13 m/s. On the other hand, for the inverted U bend, the minimum pressure drop observed was 1044.5 Pa/m for the same sets of superficial velocities of kerosene and water. It is evident that the U bend leads to a higher pressure drop during the kerosene water flows in the bend section.

Keywords: Inverted U bend, Flow regime, flow patterns, Pressure drop

DESIGN, DEVELOPMENT & CFD MODELLING OF THERMAL OIL BASED PARABOLIC TROUGH COLLECTOR

Vijay Kumar Pandey [1] , Lalan Kumar [1,2] , Om Prakash [1] , Lakhbir Singh Brar [1] , Saurav Chakraborty [1] , Asim Ahmad [3] , Anil Kumar [4]

[1]Department of Mechanical Engineering, Birla Institute of Technology, Mesra, Ranchi, India, 835215
[2]Faculty of Engineering and Applied Sciences, Usha Martin University, Ranchi, India, 835103
[3]Department of Mechanical Engineering, National Institute of Technology, Silchar, India,788010
[4]Department of Mechanical Engineering, Delhi Technological University, Delhi, India,110042

This paper describes the comprehensive analysis of parabolic trough solar collector (PTSC) which was completely performed in this study. Based on the simulation on COMSOL 6.0, an actual system parameters solar and thermal models were developed by using differential and non-linear algebraic correlations. The numerical solution of a combined set of heat and mass transfer partial differential equation was achieved using the Finite Element (FE) method. The model employed Lagrange triangle Finite Elements of extremely small size and fourth-order geometry shape for meshing the model's geometry. Through this developed model, the temperature distribution across the collector tube was analyzed at varying solar radiation conditions, revealing both maximum and minimum temperature points. The Total Embodied Energy was found to be approx. 2061.36 kWh. The predicted versus the experimental results observed that the heat generated is suitable enough for the heat exchanger to work efficiently as well as sustaining the environmental balance.

Keywords: Solar energy, Parabolic solar trough collector, Heat transfer in fluid, COMSOL Multiphysics, Thermal modelling, Energy analysis.

STUDY OF PRESSURE DISTRIBUTION AROUND CIRCULAR CYLINDERS

Kunal[a], Sushil Kumar Dhiman[a]
[a]Birla Institute of Technology, Mesra, Ranchi-835215,India

The present work deals with study of pressure distribution around circular cylinders in cross-flow. Experiments for single and two tubes were performed in sub-sonic wind tunnel at a sub-critical Reynolds number of 35000. The experimental results were also compared with the previous literature and it was found to be in good agreement. Pressure distribution around cylinder was found to be function of pitch ratio and arrangement of tubes. Pressure distribution for the single and front tube of two-tube arrangement was found to be similar in nature with varying separation points. In the single tube the rear part has flatter distribution of pressure, however, in the front part there is continuous decrease in pressure upto the flow separation point. Effect of vortex shedding was present in the downstream tube, hence suction pressure dominated in the downstream tube.

Keywords: Reynolds number, pitch ratio, Cross-flow, tandem arrangement, flow separation.

REVIEW OF POLLUTION CONTROL MEASURES FOR METALLURGICAL COMPANIES IN SUB-SAHARAN AFRICA

Kabiru Bashir
Department of Mechanical Engineering, Kano State polytechnic, Nigeria

Steel production and other raw materials like iron ore and scrap and steel products are traded worldwide. Over the last decades' improvements have been achieved in the Metallurgical industry regarding several major polluting substances, and gradually the environmental impact has shifted towards so called diffuse sources of pollution. Nevertheless, Steel production processes still account for a considerable share of the overall pollution in Sub-Saharan Africa. This paper analyzed the various types of industrial pollutants, examines their implication to human life and the environment. It also prescribes alternative Steelmaking processes, remediation technologies and generally highlights some pollution prevention, mitigation and control measures for Metallurgical industries.

Keywords: pollution, industrial pollutant, prevention, industry, environment.

Enhancement of specific heat capacity of silica nanofluids synthesized in nitrite-nitrate ternary salt for concentrated solar power applications

Anupama V Joshi[a], Vinod Kallur[b]
[a]Anupama V Joshi, Department of Chemical Engineering, RV College of Engineering, Bengaluru-560059, India
[b]Vinod Kallur, Department of Chemical Engineering, RV College of Engineering, Bengaluru-560059, India

The application of molten salts in Concentrated Solar Power (CSP) systems is limited by their high melting point and low specific heat capacity. The presence of nanoparticles in molten salts is known to enhance specific heat capacity of molten salts. We report in this work synthesis of a molten salt-based heat transfer fluid (HTF) with low melting point and significantly enhanced specific heat capacity on dispersion of silica nanoparticles in it. A novel ternary molten salt HTF composed of $NaNO2 - KNO3 - CaNO3$ was synthesized by fusing the individual salts at high temperature. 1 wt.% silica nanoparticles of size $(30 - 50nm)$ were dispersed in HTF using a two-step process – dispersion using probe sonication with distilled water as the medium and evaporation of water from molten salt-based heat transfer nanofluid (HTnF). The melting point of the synthesized nanofluid is between $61°C - 63°C$ which is much lower than the melting points of other known molten salt HTFs ($91°C$-$225°C$). The specific heat capacity (Cp) of the ternary molten salt and ternary molten salt nanofluid were measured using differential scanning calorimetry. The dispersion of nanoparticles caused 49% enhancement in specific heat value which is substantial (Figure 1). The highest reported enhancement in the literature for 1% loading is 27%. The enhancement is notably nearly uniform over a wide range of liquid state temperature (65°C-300°C). The SEM images indicate presence of nanoparticles and nanoclusters (Figure 2). FTIR and XRD studies were also performed to understand the structure and composition of HTF and HTnF. This work has not only resulted in synthesis of a novel low melting ternary molten salt HTF but has also opened the possibility of exploring a mechanism for specific heat enhancement through the formation of nanoclusters.

Keywords: molten salts; specific heat enhancement; heat transfer fluid; differential scanning calorimetry

Performance of silicon and nitrile rubbers in the constrained layer damping using fractional order derivative constitutive model

Rahul Kumar, Satyajit Panda and Vishwanil V. Sarnaik
Department of Mechanical Engineering, Indian Institute of Technology Guwahati, Guwahati 781039, Assam, India.
Corresponding author address: Rahul Kumar, Department of Mechanical Engineering, Indian Institute of Technology Guwahati, Guwahati 781039, India.

In this work, the performance of silicon and nitrile rubbers in the constrained layer damping (CLD) of vibration of a beam element is investigated. The beam element is taken in the sandwich configuration with the constrained rubber core between two stiff layers. The sandwich beam is considered to operate under a transverse triangular impulse excitation, and the corresponding nonlinear transient responses are analyzed to address the usefulness of silicon/nitrile rubber-based CLD in the structural vibration attenuation. First, the viscoelastic properties of silicon and nitrile rubbers are determined experimentally through the dynamic mechanical analysis (DMA). Subsequently, the experimentally obtained dynamic properties are modelled through the four parameter fractional Zener constitutive relation. On the basis of this constitutive model, a nonlinear finite element (FE) model of the sandwich beam is derived in the time domain for the evaluation of nonlinear transient responses under the transverse triangular impulse excitation. The results reveal a good damping capability of both the silicon and nitrile rubbers for vibration attenuation of beams through the CLD technique. However, nitrile rubber provides more effective CLD than that for the silicon rubber.

Keywords: Viscoelastic damping, sandwich beam, constrained layer damping, fractional order derivative model, finite element method.

Optimization Driven Fault Detection: A Comprehensive Study on the Heath of Bearings

Subhranil Das[a], Rashmi Kumari[b], Raghwendra Kishore Singh[c]
[a]School of Buisness, MIT World Peace University, Pune, Maharastra
[b]School of Computer Science Engineering and Technology, Bennett University, Greater Noida, Uttar Pradesh
[a]Department of Electronics and Communication Engineering, NIT Jamshedpur, Jharkhand

The last decade has seen Prediction and Health Management (PHM) technologies become increasingly significant in military, civilian, and aerospace applications. One core aspect of PHM is the ability to monitor health conditions and predict potential faults in equipment. With equipment complexity escalating, the ability to foresee failures has become more critical. A failure in any component, such as a bearing, could lead to complete operational failure, resulting in substantial economic losses. This paper explores the application of new optimization technique, Multi-objective Moth Swarm Algorithm (MOMSA) for finding the faults in the diagnosis inn the bearings where the real timed data has been acquired through FEMTO dataset. Moreover, the simulation results have been executed in order to find the faults in the bearings by using the values of specific entropy.

Keywords: Health Management; Fault Prediction; Bearing Analysis; Machine Learning; Deep Learning; Health Indicators; Optimization Algorithm.

An Investigation of Prototype Development using 3D Printer: A Digital Fabrication Approach

Amit Kumar Rana[a], Sanjib Kundu[b], Suchibrata Datta[b]
[a]Department of Mechanical Engineering, ICFAI University Tripura, Agartala-799210, India
[b]Department of Mechanical Engineering, Academy of Technology, Hooghly-712121, India

Additive manufacturing (AM) is a widely used technique in several industries for rapid prototyping and layer-by-layer fabrication of a new product. The influence of printer settings and print quality is essential for the performance of products in a variety of applications. The popularity of manufacturing using 3D printing is growing rapidly. In view of this, an investigation is carried out on the 3-D printing techniques, their applications, and prototype materials for complex mechanical parts such as an impeller. The present study thoroughly considers the fundamental techniques in 3D printing technology, materials, and recent advancements applied to the current application and accepted trends. The 3D printing technology is advantageous as it accepts a wide range of designs, allows customized manufacturing, minimizes waste, builds intricate structures, and creates prototypes rapidly. Layer height of 0.2 mm, speed of 90 mm/s, and infill density of 80% were determined as the optimal print settings considering time, material cost, and properties. Considering the flexural qualities, the optimized values of layer height, print speed, and infill density were 0.15 mm, 70 mm/s, and 50%, respectively. It was observed that the uniform elongation (UEL) and ultimate stress (UTS) of a component increase as the layer thickness increases for the considered layers.

Keywords: Additive Manufacturing; 3D Printing; 3D printing technologies; Digital Fabrication; prototypes

Bond graph modelling and dynamic analysis of an Accumulator Based Hybrid Power Transmission for wind turbine

Prabhat R. Mahto[a,b] and Anil C. Mahato[b]
[a]Department of Mechanical Engineering, University of Polytechnique, Birla Institute of Technology, Mesra, Ranch, Jharkhand-835215, India
[b]Department of Mechanical Engineering, Birla Institute of Technology, Mesra, Ranch, Jharkhand-835215, India

This work discussed the detail dynamic model of an Accumulator Based Hybrid Power Transmission (ABHPT) technology. It is an unite operation of two individual power transmission systems such as gear train and a state-of-art hydraulic power transmission (HPT) with an accumulator. The output shaft of the gear train is couple to the hydraulic pump that supplies flow to the hydraulic motor of the HPT. An accumulator and two control valves are incorporated between hydraulic pump and the motor. An ON/OFF controller controls the control valves for charging and discharging operation of the accumulator. The dynamic bond graph model of the ABHPT is developed and simulated using SYMBOLS SHAKTI software. The simulation responses of the model indicate that the output speed of the hydro-motor is stable to 8.2 rad/s although the input speed to the gear train is variable in nature. Moreover, the pressure and energy storage variation into the accumulator is reported.

Keywords: Hybrid power transmission; Bond graph; Dynamic modelling; Accumulator.

Constitutive modeling for understanding stress-stretch behavior of Lennard-Jones non-crystalline molecular Solid

Raj Kumar and Paritosh Mahata

Department of Mechanical Engineering, Birla Institute of Technology, Mesra, Ranchi- 835215, India

Non-crystalline molecular solid materials have many scienti c and engineering applications. This study develops a constitutive equation for understanding stress-stretch behavior of non-crystalline molecular solid using Lennard-Jones (LJ) intermolecular interaction. The strain energy derived from Lennard-Jones interactions between molecules. Based on the excluded volume (spherical volume occupied by the molecules maintaining center to center distance with a reference molecule) and density of the molecules, strain energy density is developed. In order to relate the molecular approach with continuum approximation, the excluded volume and density are expressed in terms of strain invariants of right Cauchy Green deformation tensor. Finally, the constitutive equation is expressed in terms of Cauchy stress tensor using the present strain energy density function. The present constitutive model is used to study nite deformations of the molecular solid like uniaxial extension. We compare our theoretical results with the experimental data of exible polyurethane foams and obtain very good agreements. The current constitutive model can predict the deformation of micro/nano engineering system components.

Keywords: Constitutive Model, Stress-Stretch, Lennard-Jones Potential, Finite Deformations

Properties and characterization of Nanocellulose

Upasana Mishra[1], Dr. Neeraj[2]
[1]Research Scholar, Department of Chemistry, Ranchi University Ranchi, Jharkhand, India
[2]Asst. professor, University Department of Chemistry, Ranchi University Ranchi, Jharkhand, India

In the field of nanotechnology, nanocellulose plays crucial role. Some features of nanocellulose like high surface area, crystallinity, porosity and tensile stress makes remarkable changes in medical and Industrial area. It is known for its biocompatibility, renewability, high aspect ratio, infinitesimal toxicity and sustainability. Nanocellulose can be prepared by microcrystalline cellulose. For the synthesis of nanocellulose, Acid hydrolysis method was used. Various method of characterizations was used for the prepared nanocellulose such as Fourier transforms infrared spectroscopy (FT-IR), Scanning electron microscopy (SEM) and X-ray diffraction. Nanocellulose drew the attention of scientists for their biochemical properties due to its physical and chemical structure. Various applications of nanocellulose in different field such as biochemical, waste water management, electronics, packaging, automotive etc

Keywords – Sustainability, Acid hydrolysis

Strengthening 5th Series Aluminum Alloy with Silicon Carbide: A Stir Casting Approach

Spruha A. Dhavale[1], Shivprakash B. Barve[2]
[1]Department of Mechanical Engineering, Dr. Vishwanath Karad MIT World Peace University, Pune, India.
[2]Department of Mechanical Engineering, Dr. Vishwanath Karad MIT World Peace University, Pune, India.

This paper presents a comprehensive experimental exploration into metal alloy composites by reinforcing Silicon Carbide with 5th series Aluminum Alloy for maritime applications, employing the stir casting method with varying Silicon Carbide weight fractions (0-6%). The study evaluates crucial properties like tensile strength, microhardness, and ductility by incorporating micron-sized Silicon Carbide particles into AA 5086 aluminum alloy matrix. Utilizing scanning electron microscopy, morphological characteristics and particle distribution were examined, while X-ray diffraction analysis identified alloying elements. Results indicate a noteworthy enhancement in tensile strength, peaking at 196.23 MPa in the 6% SiC composite, marking a 20.49% increase over the base alloy. Microhardness values improved, reaching 120.97 VHN for the 6% Silicon Carbide-reinforced Aluminum Alloy Composites. Despite these improvements, a marginal reduction in ductility was observed in the Silicon Carbide reinforced composites. This study offers valuable insights into advancing materials for the maritime industry, underscoring the potential for future research in this domain.

Keywords: Silicon Carbide; Aluminum Alloy; Metal Matrix Composites; Stir Casting

Tribological Properties of Nano lubricants Utilizing Graphene Oxide Nanosheets Prepared by Ball Milling

Anand Gaurava, Gayatri Paula
[a]Department of Production and Industrial Engineering, Birla Institute of Technology Mesra, Ranchi, Jharkhand - 835215

Graphene oxide (GO) nanosheets are prepared by mechanical exfoliation and fragmentation of graphite under wet ball milling conditions. The successful exfoliation of graphite into few-layered graphene is examined through X-Ray diffraction showing amorphous peaks of GO and field emission scanning electron microscopy exhibiting fragmented layered GO nanosheets. The synthesized GO nanosheets are utilized to prepare synthetic oil based nanolubricants by two-step process. dispersion stability of the nanolubricants tested through UV-Vis spectrophotometry exhibit marginal decrease in relative absorbance suggesting minimal settlement of the nano-additives over a period of one month. Sliding reciprocation wear testing of mating stainless steel tribo-pair exhibits lower coefficient of friction (COF) in presence of nanolubricants in comparison to base oil. Increase in the concentration of the nano-additive leads to a higher reduction in the average COF. A maximum reduction of 22% and 35% is observed for 0.1 wt.% and 0.2 wt.% dispersed nanolubricants, respectively. An increased applied load increases the average COF significantly. Wear scar analysis shows reduced width of the wear scars with deposition of GO nanosheets on the wear tracks leading to the reduction in COF of the mating tribo-pair.

Keywords: Graphene oxide nanosheets; Ball milling; Nanolubricants; Sliding wear; Tribology; Friction

An experimental investigation on the effect of filler form and temperature on compressive strength of epoxy composites

Saurav Ranjan Sahay[a], Siddharth Kumar[a]*, Sarthak S. Singh[a]

[a]Polymer Mechanics Research Lab, Department of Mechanical Engineering, Indian Institute of Technology (Indian School of Mines) Dhanbad, Dhanbad - 826004, India
[a*]Corresponding Author, Email: siddharthkr.20dr0138@mech.iitism.ac.in, Polymer Mechanics Research Lab, Department of Mechanical Engineering, Indian Institute of Technology (Indian School of Mines) Dhanbad, Dhanbad - 826004, India

This study aims to improve the compressive strength of epoxy at operating temperatures below the glass transition temperature by introducing micron-sized spherical and milled glass fillers at varying volume fractions. Test specimens were prepared by mixing spherical and milled fillers up to 15% and 20% volume fractions, respectively at 27°C, 45°C, and 60°C. Results showed that 15% and 10% volume fractions of spherical and milled filler-reinforced epoxy composites exhibited the highest stress-bearing ability among their respective filler volume fractions. However, the 10% volume fraction of milled filler composites provided higher yield and post-yield strength compared to a 15% volume fraction of spherical filler composites at all temperatures considered.

Keywords: Polymer-matrix composites, Non-linear behavior, Stress/Strain curve, Scanning Electron Microscopy (SEM)

Analysis of Sandwich Composite Subjected to Impact Loading for Enhanced Energy Absorption

Valluru Revanth, Namala Kiran Kumar, Chintamaneni Rajeswari, Ch NaveenReddy, Sai Hari SNS
Department of Mechanical Engineering VNR VJIET, Vignana Jyothi Nagar, Pragathi Nagar, Nizampet (S.O), Hyderabad, Telangana, India - 500 090

Composite materials are engineered to achieve the balance of properties for a wide spectrum of applications. In the current work, PU foam is used as a core, and Epoxy E-Glass (UD) is used as a face sheet. Impact of sandwich composite with single core and double core are considered. The effect of energy absorption, contact force, residual velocity, and deformation on impact velocity are discussed. The phenomenon of perforation, penetration and rebounding was observed in 150 m/s, 130 m/s and 100 m/s respectively. It was found double core sandwich composite absorbs more energy and even stiffer than single core sandwich composite.

Keywords: Contact force, Residual velocity, Energy absorption, Single core sandwich composite, Double core sandwich composite, Contact time.

Statistical Analysis of Input Parameters and their Interaction on Surface Roughness in Incremental Sheet Metal Forming

Ranjeet Prasad[1], Manish Oraon[1], Vinay Sharma[1*]
Department of Production and Industrial Engineering, Birla Institute of Technology, Mesra

A cost-effective solution for rapid prototyping, customized production, or scenarios with low batch sizes is developed through Incremental Sheet Metal Forming (ISMF), utilizing a hemispherical tool moving along predetermined paths to deform the sheet metal. The primary challenge involves controlling surface roughness via layer-by-layer technique on the worksheet. To optimize results, input variables and their interactions are studied, considering some variables with contradictory natures not previously analyzed. This experimental study aims to analyze the significance of input parameters and their interaction on surface roughness. Statistical tools are employed to analyze effects, revealing that both the type of metal being deformed and its interaction with lubricant significantly influence controlling surface roughness of the formed product.

Keywords: ISMF; Surface Roughness; Input Parameters; ANOVA; Interaction.

Comparative study and characterization of the products formed by the interaction of D-Ribose with TBC in DMF & DMSO as a solvent

Kriti kumari[1, a)] and Hari Om Pandey[2, b)]

[1]Research scholar, Deptt. of chemistry, Ranchi University, Morabadi Ranchi – 834008- Jharkhand India
[2]Associate prof. Deptt. of chemistry, Ranchi University, Morabadi Ranchi – 834008- Jharkhand India

A comparative study was conducted to characterize the products resulting from the interaction of D-Ribose with (TBC) in two different solvents, namely dimethylformamide (DMF) and dimethyl sulfoxide (DMSO). The investigation aimed to understand the solvent-dependent variations in the formation of products and their characteristics. The synthesized compounds were subjected to thorough characterization, with a focus on their structural composition and properties. The results of this comparative study contribute to a comprehensive understanding of the solvent-dependent reactivity of D-Ribose with TBC, shedding light on potential applications and variations in product formation based on the choice of solvent. The products resulting from the interaction of D-Ribose with (TBC) in two different were investigated within the context of microwave-assisted organic synthesis (MAOS). The research aimed to discern solvent-dependent variations in product formation and to characterize the synthesized compounds comprehensively. The resulting products being analysed which depends on the nature and characteristics of the reaction using various techniques such as FTIR, TGD-DTA, Elemental analysis etc Elemental analysis facilitated the determination of the percentage composition of carbon, hydrogen, and oxygen in the products, offering insights into their molecular structures. Additionally, the chromium content in the compounds was estimated through volumetric titration. This comparative study contributes valuable insights into the reactivity of D-Ribose with TBC under microwave-assisted conditions, elucidating potential solvent effects and providing groundwork for understanding the mechanisms underlying microwave-assisted organic reactions.

Keywords: - Ditertiary butyl chromate (TBC), Carbohydrates (Ribose), CrO3 , FTIR, TGA-DTA, DMF (Dimethyl formamide), DMSO (Dimethyl sulphoxide) , TBA (tert. butyl alcohol).

Assessing Formability Behavior of High-Strength Thermal Resistant (HSTR) Alloys

Ranjeet Prasad[a], Manish Oraon[b], Vinay Sharma[c]

[A,b,c]Department of Production and Industrial Engineering, Birla Institute of Technology, Mesra, Ranchi 835215

The demand for sheet metal forming is significant globally, with various methods employed to achieve desired shapes based on specific applications. High-strength, thermally resistant metal alloys play a vital role in this market, finding wide use in industries such as automotive, aerospace, energy production, petrochemical refining, defense, electronics, and industrial applications. This paper examines the formability of such alloys, particularly focusing on their behavior in incremental sheet metal forming processes. The comparison of formability tests investigates the economic feasibility and ease of forming these alloys. Notably, incremental sheet metal forming is emphasized due to its cost effectiveness in producing customized products. Alloys like nickel-based superalloys, cobalt-based superalloys, and specific stainless steels are favored for high-temperature environments due to their superior strength, thermal resilience, and resistance to oxidation and corrosion. Through this research, insights into optimizing the formability of high-strength, thermally resistant metal alloys for various applications can be gained.

Keywords: Formability; HSTR; ISMF

A performance analysis of wind turbine blade with changing pitch angle by using FSI

Raj Abhishek, Swaroop Kumar Mandal, Rahul Kumar, Deepak Kumar*, Shubham Pattanayak, Tathagata Gautham

Department of Mechanical Engineering, National Institute of Technology Jamshedpur, Pin - 831014, India

Wind energy technology advancement is a major priority for many countries worldwide. Almost every aerodynamic study of wind turbine blades and simulations to check aero elasticity is currently done with lo-fi models for easy implementation and simulation. The modeling of fluid-structure interaction (FSI) with a one-way FSI coupling ANSYS application. The blade, in which significant structural changes have been done in the recent past, is the main power-generating element in the turbine. The pitching moment in the turbine blade plays an ultimate role in power production. For a particular wind speed, the pitch angle that will give the best result is having the optimum angle of attack. Here we have taken composites such as Glass-S, Kevlar and CFRP (Carbon Fiber Reinforced Polymer). The impact of pitch angles on the turbine's performance is discussed. The SST k-ω turbulence model was considered in this study. The work showed that in the case of CFRP, the Von-Mises stress and blade deflection are comparatively less for the same range of wind velocity and pitch angle variation. Pitch angle 40 is found to be the best-suited angle for all the considered velocity ranges, for a rated wind speed of 10m/s, for all the materials.

Keywords: - Wind Turbine, FSI, Blade Pitch Angle, Composites, Turbine Blade Aerodynamics.

Influence of Natural fiber on drop weight Impact test of Hybrid fiber Reinforced Concrete

Sagar Sarangi[a], Birendra Kumar Singh[b], Abhay Rajak[a]

[a]Sarala Birla University, Ranchi, Jharkhand, India
[b]Birla Institute of Technology, Mesra, Ranchi, Jharkhand, India.

Concrete is a heterogeneous brittle material massively used in construction industries. To make it a ductile one discrete and randomly oriented reinforcing agent were incorporated into concrete matrix. Most of the time only compressive strength, split tensile strength and flexural strength test were considered as parameter of mechanical properties of the concrete while neglecting on the impact test. In this paper we have investigated on drop weight impact test to obtain the influence of discrete and randomly oriented natural fiber in a cement based composite material. From the observation obtained that incorporation of reinforcing agent not only enhanced density but also enhanced mechanical properties like compressive strength, tensile strength and impact resistance compared to conventional concrete. Due to higher degree of impact energy absorption rebound hammer test is not acceptable for the fiber reinforced concrete.

Keywords: Compressive strength; Composite Material, Impact Test; Density.

An investigation of the deformation caused by balls in a flexible race ball bearing using a 2D numerical method

Apoorva Verma[1], Vineet Sahoo[1]

[1]Department of Mechanical Engineering, National Institute of Technology Jamshedpur-831014.

Flexible Race Ball Bearing is commonly used in power transmission systems, particularly in precision drives. The flexion of the bearing structure can lead to deformation even no external loads are applied. The present investigation shows a comparative analysis of the deformation and stress distribution between the flexible race ball bearing and single row deep groove ball bearing. A 2D Finite Element Method model is used for the analysis which shows the effect of race thickness and load applied. The race thickness of the single row deep groove ball bearing gradually decreases until it precisely matches the race thickness of the flexible race ball bearing. The result shows the total maximum deformation occurs at the inner race of the deep groove ball bearing, which is 0.01852mm. The radial deformation at different ball positions indicates that the maximum radial deformation occurs at the position of 1800 for both deep groove ball bearing and flexible race ball bearing, occurring at a thickness of 1.65mm corresponding to the radial deformations of 0.0928mm and 0.069mm, respectively. Flexible race ball bearing gives better results due to presence of more number of balls so that it gives less total deformation, and radial stress which is 0.01409mm and 141.54MPa without failure.

Keywords: Flexible race bearing, Deep groove ball bearing, Race thickness, FEM analysis.

Design and development of a hybrid finite element analysis-based model for impact analysis of blasts on underwater

Pankaj Meena[a], R. Sharma[b]

[a]Reserch scholar, Design and Simulation Laboratory, Department of Ocean Engineering, Indian Institute of Technology Madras, Chennai (TN) - 600 036, India
[b]Professor, Design and Simulation Laboratory, Department of Ocean Engineering, Indian Institute of Technology Madras, Chennai (TN) - 600 036, India

Herein, we present the design and development of a hybrid model for impact analysis on underwater vehicles. Our primary idea is the incorporation of explosion studies in the design and development of new age shield engineering designs, especially focused towards the application to the underwater vehicles (e.g. autonomous underwater vehicles, submarines, submersibles, and torpedoes, etc.). We present extensive results that are key towards bench marking and show and analyze the effect of explosion on a square steel plate and distribution of stresses, strains and displacements after explosion/impact, etc., are examined in-detail. Our proposed hybrid model is governed by the basic numerical analysis, finite element analysis and thermal/material science. We implement our model in ABAQUS*TM and Matlab**TM software solution systems and develop subroutines to ensure seamless integration. In our model the focus is on large deformation finite element analysis, the Jones-Wilkins-Lee (JWL) equation of state, and mild steel Johnson Cook parameter. Presented results examine the role of explosion at various heights in z axis direction on a mild steel plate and the TNT is used as a representative explosive material. Finally, based upon the results and their analyses suitable design guidelines are derived.

Keywords: Hybrid finite element analysis, explosion, impact analysis, large deformation analysis, and material properties.

Finite Element Analysis of a Beam Resting on a Layered Soil

Utkarsh Deep Kamal[a], Kumar Venkatesh[b]

[a]M.Tech. Student, Department of Civil Engineering, MNNIT Allahabad, Prayagraj-211004, India
[b]Associate Professor, Department of Civil Engineering, MNNIT Allahabad, Prayagraj-211004, India

In general, the structure interacts with the surrounding soil. It is thus not permissible to analyze only the structure. The analysis of interaction behavior between soil media and the structure is important. Soil structure Interaction problems may evolve complex geometry, material properties, loadings, and boundary conditions, so finding exact solutions by an analytical approach is a cumbersome job. To improve computational effort, various numerical methods are used. Finite element method (FEM) is the most suitable numerical technique to solve the soil-structure interaction problem. In this paper, a beam resting on layered soil is analyzed analytically and numerically (by the Finite element method). The soil media is modelled as a series of Winkler's springs and beam as a structural element. Deflection of the beam has been calculated for uniform and varying flexural rigidity of the beam and subgrade modulus of soil. The formulation of Stiffness matrix and its assembly and imposition of boundary conditions are explained in the Finite element method. Analytical results are compared to those from Finite Element software PLAXIS 3D.

Keywords: Soil-Structure Interaction, Finite Element Method, Winkler's Model, Layered Soil.

Design and development of collapse model for ring-stiffened cylinders under hydrostatic pressure

Rishabh Jambhulkar[a], R. Sharma[b], R. Vijayakumar[c]

[a]Research scholar, Design and Simulation Laboratory, Department of Ocean Engineering, Indian Institute of Technology Madras, Chennai (TN) - 600 036, India
[b]Professor, Design and Simulation Laboratory, Department of Ocean Engineering, Indian Institute of Technology Madras, Chennai (TN) - 600 036, India
[c]Professor, Department of Ocean Engineering, Indian Institute of Technology Madras, Chennai (TN) - 600 036, India

Herein, we present the design and development of collapse modes for ring-stiffened cylinders under hydrostatic pressure and the models are based upon analytical/numerical formulations (i.e. the Arc-Length method). Our analysis is based on cylindrical stiffened shell with key parameters such as operating depth, thickness, diameter, length between bulkheads, size, and number of ring stiffeners, etc. In the model, four sub-models with varying bulkhead lengths are examined and compared against both analytical and the 'Finite Element Analysis (FEA)' based solutions. Presented results show that the Arc-Length method effectively identifies variations in yield pressure as bulkhead length increases, and this capability is not observed in the analytical solutions. Finally, we believe that this study contributes to a deeper understanding of submarine design, particularly in accommodating different bulkhead lengths.

Keywords: Collapse model, ring-stiffened cylinders, hydrostatic pressure, finite element analysis, arc-length method, submarine, and stiffened shell.

Strategic Selection of Metal-Cutting Processes for Thick Steel Plates: A Hybrid MOORA-PSI Approach

Anita Kumari*, Bappa Acherjee

Department of Production and Industrial Engineering, Birla Institute of Technology: Mesra, Ranchi – 835215, India

Metal-cutting is an indispensable process in manufacturing, enabling precise fabrication of components with specified dimensions and desired surface quality, playing a crucial role across industries. Industries, including construction, shipbuilding, automotive, aerospace, and heavy machinery, necessitate cutting thick steel plates for structural components and machinery fabrication, emphasizing the importance of accurate and efficient cutting processes. Five prominent metal-cutting processes, including oxygen flame, plasma arc, laser, wire EDM, and abrasive water jet cutting, are widely used for cutting thick steel plates, each offering distinct process capabilities, advantages, and limitations. This paper introduces a novel case study to strategically determine the optimal metal-cutting process for thick steel plates utilizing a hybrid MOORA-PSI approach, addressing the literature gap in this area. The use of the hybrid MOORA-PSI method simplifies decision-making by integrating weight assignment and alternative ranking. Wire EDM ranks as the optimal choice for cutting thick steel plates based on defined evaluation criteria, with laser cutting closely trailing, followed by oxygen flame, abrasive water jet, and plasma cutting successively. The results are validated by comparing them with those of other MCDM approaches and by conducting a Spearman's rank correlation coefficient test, yielding consistent results. Additionally, sensitivity analysis, employing criteria weight exchange and dynamic variations in the decision-making matrix, further confirms the accuracy and reliability of the findings.

Keywords: Metal cutting, thick steel plates, process selection, MCDM, hybrid MOORA-PSI method, sensitivity analysis

Post Processing of Waste Plastic after Shredding from a Shredder for Efficient Use in Construction Materials

Phuleshwar Baitha[a], Richa Pandey[a]

[a]Department of Mechanical Engineering, BIT Mesra, Ranchi 835215, India

In the present global scenario, the usage of plastic and papers is very high and this in return is creating a huge environmental impact. Plastics and papers are two of the most commonly used materials in the world today. In this project, we are majorly concerned with plastic waste. This plastic waste is causing severe effects on both land as well as marine life. Hence the need of the hour is to design and manufacture a 'Plastic Shredder machine' which is not only highly effective but cost efficient. The shredding machine will play a considerable role in the plastic waste recycling process and up to some extent in solving the problems associated with these wastes. The shredder machine is designed to shred various types of plastic waste, such as LDPE, PET, HDPE, and PVC, into small pieces that can be used as raw materials for making new products. The machine is cost-effective, easy to operate, and can shred large volumes of plastic waste in a short time. The machine has five main subassemblies that make up the machine. The driving mechanism, which combines a V-belt drive and gear mesh drives, the prime mover, and the machine frame, which includes the machine stand, feeder unit with the hopper and the cutting unit housing, cutting unit with the cutting blades and shaft, and drive mechanism. The machine's primary mover provides the necessary power for it to function. Our desired thickness of output pellet is 10mm, so the thickness of the cutter blade designed is 10 mm as thickness of the output pellet is directly proportional to thickness of cutter blade. On Performance evaluation at a motor speed of 1440rpm it was found that machine efficiency for HDPE, PET and PVC waste plastic was around 91-94% and for LDPE was around 83 %. From performance evaluation it was found that PET type plastic waste has a maximum efficiency of 93.63%. The shredded plastic waste was then used in fabrication of bricks. The bricks made from waste plastic have comparable properties to traditional clay bricks, which means they are strong, durable, and suitable for use in construction. Moreover, the use of waste plastic as a raw material for construction provides a sustainable solution to the plastic waste problem. Since most of the plastic bottles are made from PET polymer, therefore the developed machine can be used as a waste plastic bottle crusher and can be installed at various public areas like railway stations, airports, etc. Future research can lead the developed machine to provide higher efficiency for LDPE plastic.

Keywords: Waste management, Plastic Shredder, Post processing, Construction material, Bricks;

Deformation and volume change of axisymmetric non-linear dielectric elastomer membranes: a theoretical study

S.O.V Jaswanth Sriharsha, Rakesh Bollu, Vancha Bhaavan Reddy, Dr. Nirmal Kumar

Birla Institute of Technology Mesra, Ranchi, 835215, INDIA

This work investigates the change in volume of an axisymmetric non-linear hyper-elastic membrane subjected to dielectric actuation. The study involves solving the equilibrium equations for the deformed state of the membrane under pressure loading to determine the required pressure for a given deformation. Subsequently, an electric field is applied to the membrane, which, due to the dielectric nature of the material, induces a change in the material properties and deformation of the membrane, leading to a change in its overall volume. A non-linear material model is employed for the constitutive equations, and all variables are non-dimensionalized. The equilibrium equations in cylindrical coordinates, coupled with the non-linear constitutive relations, are solved numerically to obtain the deformed membrane geometry under pressure loading. The effect of dielectric actuation is incorporated by considering the change in material properties and deformation induced by the applied electric field. The deformed geometry under dielectric actuation is then determined, and the change in volume is calculated by comparing the deformed states with and without the applied electric field. The proposed approach enables the quantification of the volumetric change in non-linear hyper-elastic membranes due to dielectric actuation, which has potential applications in areas such as soft robotics, adaptive optics, and microfluidic devices. The non-dimensionalization of variables facilitates the identification of key dimensionless parameters governing the problem and allows for a broader applicability of the results.

Enhancing Dynamic Control Of Left Ventricle Pressure Through High-Capacity Pump Integration

Aditya Kumar Singh, Deepak Raj, Nirmal Kumar

Birla Institute of Technology Mesra, Ranchi 835215, Jharkhand, India

The dynamic control of pressure within the left ventricle is critical in various medical applications, particularly in simulating physiological conditions for research and training purposes. In this study, we investigate the enhancement of pressure control mechanisms within an artificial left ventricle apparatus by integrating a pump of higher capacity. The apparatus comprises an air compressor providing controlled pressurized air and a suction pump facilitating efficient operation. By replacing the suction pump with a higher-capacity alternative, we aim to evaluate the resultant changes in various control parameters governing the system's functionality. The theoretical foundation underpinning this endeavour involves principles of fluid dynamics, control theory, and biomedical engineering, emphasizing the importance of precise pressure regulation for realistic simulation and effective training in medical contexts. Through rigorous experimentation and analysis, this study seeks to elucidate the efficacy of utilizing advanced pump technology to optimize dynamic pressure control within the left ventricle, thereby advancing the state-of-the-art in medical simulation and training methodologies

Entropy-Driven Microplasticity Effects on Fatigue Fracture Behaviour: Insights and Implications"

Prakyat Reddy B, Lankapalli Jaswanth, Dr. Nirmal Kumar.

Birla Institute of Technology, Mesra, Ranchi, 835215, INDIA

This study investigates the effects caused by entropy in the formation of fracture in a testing specimen upon being subjected to axial cyclic loading. The objective is to find the point where microplasticity is observed throughout the stress cycle. Various tests with different loads are performed till breakdown using a fatigue testing machine.

Graphs are used to determine the point where microplasticity begins and where it turns into plasticity. Points where the graph takes a drastic turn due to the emergence of plasticity is observed and noted down. The load is applied on the specimen till it forms a crack. During this, various parameters such as frequency, stress amplitude and tensile strain are noted.

The study examines how the speed of loading impacts the onset of tiny changes in materials. Additionally, it tracks the applied force during testing. The investigation aims to determine if cracks require reaching a specific size before exhibiting these small alterations. This research provides insights into material fracture mechanisms and their implications for product durability and safety.

This approach provides the life cycle of a product under similar situations and it could be used to find the limit to which it could be utilised. A hypothesis is also observed where microplasticity is only observed when the crack crosses a certain size, in this case being grain size. Otherwise, the crack is assumed to regenerate and retain its elasticity.

Numerical study of viscoelastic drop impact on a non-wetting heated surface

Abhishek Raj[a]

[a]Department of Mechanical Engineering, IIT Guwahati, Assam-781039, India

Study of viscoelastic drop impact on a heated solid surface is carried out in an axisymmetric configuration using numerical simulations. An initial spherical viscoelastic drop placed in a surrounding Newtonian fluid is vertically impacted on a heated non-wetting solid surface. The viscosity and relaxation time is considered to be varying with temperature. Effects of surface temperature and the Reynolds number on drop impact dynamics is studied. At high surface temperatures, the drop rebounces above the surface after the impact. With increase in the surface temperature the rebouncing becomes stronger and reaches higher heights after the rebouncing. The maximum height of drop increases by about 65% with an increase in dimensionless surface temperature from 0 to 0.4. With increase in Reynolds number drop rebounces to higher heights. As Reynolds number increases from 5 to 75, the maximum height of drop increases by about 142%. For high Reynolds numbers, multiple surrounding fluid bubbles are entrapped in the drop during the impact.

Keywords: Viscoelastic; Drop impact; Heated surface; Rebounce; Bubble entrapment; Relaxation time

Newtonian bubble rise in a viscoelastic surrounding fluid in a channel with imposed wall temperature variation

Onkar V. Bichkar[a]

[a]Department of Mechanical Engineering, IIT Guwahati, Guwahati-781039, India

A study of Newtonian bubble dynamics in a viscoelastic surrounding fluid in a channel is carried out. Viscosity of both bubble and surrounding fluid are temperature dependant. Linearly varying temperature is imposed on channel walls. The effects of temperature gradient and solvent to total viscosity ratio on bubble dynamics are studied. Without temperature gradient, bubble attains terminal shape, but with imposing temperature gradient, bubble shape changes with elevation of domain. For larger temperature gradients, the flow becomes asymmetric about mid vertical plane. In comparison with Newtonian surrounding fluid velocity of bubble is higher in viscoelastic surrounding fluid. The velocity increases with increase in viscoelastic property of surrounding fluid. With increase in viscoelastic property of surrounding fluid, the bubble undergoes stronger shape variations at the beginning of bubble rise.

Keywords: Bubble rise; Linear temperature variation; Temperature gradient; Viscoelastic fluid

CFD Investigation on the Efficacy of Horizontal and Vertical Air Curtain Systems in Refrigerated Room

Kedar Vaibhav Kulkarni, Joel Varghese Mammen, R. Harish*

School of Mechanical Engineering, Vellore Institute of Technology, Chennai, Tamil Nadu 600127, India.

Air curtain systems have been used extensively since the 1940s and 1950s for separating two different environments in terms of heat and mass transfer while still allowing the flow of traffic between the environments, thereby providing an invisible barrier. The applications of air curtain systems spreads over hundreds of industries such as preserving temperature sensitive goods in refrigerated trucks, warehouses and also using different configurations of air curtains (vertical, horizontal, multiple jets, etc.) at different values for parameters such as temperature, mass flow rate, velocity and discharge angles of the air curtains. In this paper, analysis is done by (i) varying the discharge angles in a single room (ii) comparing the efficiency of different configurations of the air curtain (vertical and horizontal) using a two-room configuration coupled with an air conditioner in both cases.

Keywords: Configurations; Temperature; Discharge angle; Vertical; Horizontal

Development of an Energy Regenerative Technology for the Swing Mechanism of a Hydraulic Excavator Machine and its Performance Analysis using Simulation Technique

Shubhyanshu Singh[a] , Ajit Kumar[b] , Shatrughan Thakur[c] , Naveen Kumar[d] , Jayant Alfred Joseph Ekkae

[a, b, c, d, e] Department of Mechanical Engineering, Indian Institute of Technology (Indian School of Mines) Dhanbad, Dhanbad 826004, India

The present study deals with the development of an energy regenerative technology by using a hydraulic accumulator in the conventional swing drive system of a hydraulic excavator machine. The swing motion of the hydraulic excavator machine consists of acceleration, constant speed rotation and then deceleration of its upper deck. The high- pressure fluid returns to the hydraulic tank through the pressure relief valve (PRV) as waste energy during acceleration and deceleration. The accumulator incorporated in the proposed energy regenerative swing system stores the waste energy released during the turntable (superstructure) swing motion of the excavator. The energy stored in the accumulator is used to drive the swing hydraulic motor along with the main hydraulic pump in the next cycle of the excavator's swing motion. For the analysis of the proposed system, the detailed simulation model is made in the MATLAB / Simscape environment. The performance analysis of the proposed swing system is compared with the conventional swing system through simulation. It is observed that the energy saved by the proposed system incorporating an accumulator is 24% of the energy consumed by the hydraulic pump of a conventional swing system. The energy regeneration efficiency of proposed system is 83%.

Keywords: Efficient Swing System, Mobile Hydraulic, Accumulator, Energy Saving and Regeneration, Modelling & Simulation

Multiphysics Modeling of Thermoelectric Unicouple and Comparison of Electrical Performance with Different Thermoelectric Materials

Shruti Verma[a], Shobhana Singh[b]

[a]Department of Civil and Infrastructure Engineering, IIT Jodhpur, Jodhpur-342037, India
[b]Department of Mechanical Engineering, IIT Jodhpur, Jodhpur-342037, India

This article makes use of COMSOL Multiphysics software to model and simulate thermoelectric generator (TEG). The purpose of modelling thermoelectric generator is to study the physics behind the working of TEG under steadystate condition and to investigate its thermal and electrical performance parameters using different thermoelectric materials, such as NaCuTe, NaCuSe, and Bi2Te3. This study highlights the potential of novel thermoelectric materials like NaCuTe and NaCuSe by comparing their thermoelectric properties like Seebeck coefficient, electrical conductivity, thermal conductivity and figure of merit, with the conventional thermoelectric material bismuth telluride (Bi2Te3). Various performance parameters, such as, power, current, voltage and maximum conversion efficiency, is investigated by varying the hot side temperature from 300K to 700K and keeping the cold side temperature at 300K. It is discovered that the novel thermoelectric materials NaCuTe and NaCuSe showed a higher maximum conversion efficiency of 28.50% and 8.14%, respectively, when compared to Bi2Te3 which is 4.19%. The power output of NaCuTe and NaCuSe is found to be 16.68 mW and 4.24 mW respectively, and that of Bi2Te3 is 12.23 mW. The figure of merit of NaCuTe is 4.86 and that of NaCuSe and Bi2Te3 is 0.52 and 0.24 respectively. The results show that having a high figure of merit does not imply better performance of a thermoelectric device; there are several other factors that need to be considered.

Keywords: Thermoelectric generator; COMSOL Multiphysics software; Seebeck effect and Heat transfer.

Optimizing Thermal Management in Hydraulically Operated Fan Drive Systems: A Comparative Study

Naveen Kumar[a] , Jayanta Das[b] , Shubhyanshu Singh[c] , Jayant Alfred Joseph Ekka[d] , Amit Kumar[e]

[a, b, c, d, e] Department of Mechanical Engineering, Indian Institute of Technology (Indian School of Mines) Dhanbad, Dhanbad 826004, India

This article investigates the efficient cooling method of working hydraulic fluid of a hydraulic system the performance of the hydraulic system efficiently depends on the temperature of the cooling flood working flood under the optimal temperature elevated temperature can lead to various detrimental effects including fluid degradation and increased city and component failure this article investigates the mechanism of cooling the working flood the robustness in the hydraulically operated fan cooling drive it is efficient for the calling and energy saving technique as compared to the other method of mechanical technical it was noted that the flow through the radiator affect the cooling significantly as compared to the fan speed and airflow through the radiator. Keeping the fluid flow constant through the radiator and wearing the speed of the fan that is airflow through the radiator decreases all temperatures by 8.03% and keeping the airflow constant and decreasing the flow through the radiator of working flood decreases the temperature by 11.59%

Keywords: Fan cooling Drive, Working fluid, Variable hydraulic pump, Energy Saving and Modelling & Simulation.

Design and development of collapse model for ring-stiffened cylinders under hydrostatic pressure

Rishabh Jambhulkar[a], R. Sharma[b] , R. Vijayakumar[c]

[a]Reserch scholar, Design and Simulation Laboratory, Department of Ocean Engineering, Indian Institute of Technology Madras, Chennai (TN) - 600 036, India
[b]Professor, Design and Simulation Laboratory, Department of Ocean Engineering, Indian Institute of Technology Madras, Chennai (TN) - 600 036, India
cProfessor, Department of Ocean Engineering, Indian Institute of Technology Madras, Chennai (TN) - 600 036, India

Herein, we present the design and development of collapse modes for ring-stiffened cylinders under hydrostatic pressure and the models are based upon analytical/numerical formulations (i.e. the Arc-Length method). Our analysis is based on cylindrical stiffened shell with key parameters such as operating depth, thickness, diameter, length between bulkheads, size, and number of ring stiffeners, etc. In the model, four sub-models with varying bulkhead lengths are examined and compared against both analytical and the 'Finite Element Analysis (FEA)' based solutions. Presented results show that the Arc-Length method effectively identifies variations in yield pressure as bulkhead length increases, and this capability is not observed in the analytical solutions. Finally, we believe that this study contributes to a deeper understanding of submarine design, particularly in accommodating different bulkhead lengths.

Keywords: Collapse model, ring-stiffened cylinders, hydrostatic pressure, finite element analysis, arc-length method, submarine, and stiffened shell

Performance Analysis of Two Flow Control Strategies for Hydraulic Position Control

Jayant Alfred Joseph Ekka[a], Rakesh Kumar Nayak[b] , Jayanta Das[c] , Naveen Kumar[d] , Shubhyanshu Singh[e]
a, b, c, d, e Department of Mechanical Engineering, Indian Institute of Technology (Indian School of Mines) Dhanbad, Dhanbad 826004, India

Different techniques for enhancing the position control and stability of a power hydraulic system are now extremely necessary. Different control algorithms are being developed, and efforts are being made to address the issue of position control for stability and energy-saving strategies in the design of effective hydraulic control systems. This study compares the performance of two types of actuator position control techniques, namely, swashplate control of the primary prime mover and signal control of the Direction control valve. The approach of Swashplate control involves implementing a servo mechanism to regulate the angle of the primary pump's swashplate, using a proportional-integral-derivative (PID) controller, in contrast to the Direction control valve strategy's use of the error signal to control the opening and closing positions of the DCV through PID. Two MATLAB-Simulink models were created for this aim and tested through experiments.

Keywords: Position Control, Energy-Saving Strategies, Swash Plate control strategy, Hydraulic Control Systems, Tracking Control

Optimizing Closed-Circuit Hydrostatic Transmission Systems through Thermal Modeling

Amit Kumar[a] , Navneet Gupta[b] , Naveen Kumar[c] , Shubhyanshu Singh[d] , Niranjan Kumar[e]
[a, b, c, d, e] Department of Mechanical Engineering, Indian Institute of Technology (Indian School of Mines) Dhanbad, Dhanbad 826004, India

This Article focuses on thermal Analysis of closed-circuit hydrostatic transmission (HST) systems by thermal modeling. Two models, thermal and hydraulic, are developed in MATLAB Simulink to interface flow and pressure variations with temperature predictions. This enables pre-emptive identification and resolution of thermal issues, enhancing HST system design efficiency. Model accuracy is validated against test data, showing successful temperature predictions within a mean error of ±4.67%. Results highlight the significant impact of actuator speed on hydraulic oil temperature. Although limited to constant fluid properties, the methodology can be extended to analyse varied operating conditions, aiding in fluid selection and system optimization.

Keywords: Thermal Fluid, PPRV, Hydrostatic Transmission system (HST), Temperature Prediction, Fluid Properties

Static analysis at active contact surface of orbit motor for contact stress and deformation

Vivek Kumar Singh[1] and Abhijit Nag[2]

[1]Research Scholar, Department of Mechanical Engineering, Birla Institute of Technology, Mesra, Ranchi
[2]Department of Mechanical Engineering, Birla Institute of Technology, Mesra, Ranchi

Estimation of contact stress developing at all the ideally form closed contacts is done. These form closed contacts separates the chambers in stator and rotor gears of orbit motors. The tooth profile of rotor and stator of orbit motor are having epitrochoidal nature. The contact position of stator and rotor are inaccessible for physical measurement through installation of any measuring device or probe. Also, the theoretical calculation for the estimation of physical property like deformation and stress at contact position leads to the problem of statically indeterminate condition due to multiple contacts. Also the point of contact changes frequently with the rotation of rotor. Thus to estimate the deformation and contact stress at these contact positions we have made a model with realistic boundary condition and loading condition. To find the deformation and stress at contact position analysis is done using FEM software.

Keywords: Orbit motor, contact stress, deformation and gap

Optimizing Cyclone Separator Design for Selective Particle Size Removal

Manoj Kumar[a] , Satyanand Pandey[a] , Lakhbir Singh Brar[a] , Om Prakash[a] , Vijay Kumar Pandey[a]

[a]Department of Mechanical Engineering, Birla Institute of Technology, Mesra, Ranchi, India, 835215

Cyclone separators are crucial in various industrial processes for separating particles based on size and weight characteristics. This research investigates the optimization of cyclone separator design parameters to remove particles above a specified size/weight proportion efficiently. The critical factors affecting cyclone operation, including gas stream velocity, particle agglomeration tendencies, and cyclone geometry, are analyzed and optimized to enhance particle separation performance. Experimental studies involve varying the dimensions of the cyclone sections, specifically the cyclone diameter-to-length ratios of the cylinder and cone sections, to determine their impact on particle separation efficiency. By manipulating these ratios, cyclones can be tailored to remove particles larger than a selected size, catering to specific industrial requirements. The research employs computational fluid dynamics (CFD) simulations and experimental validation to analyze the flow patterns within the cyclone separator and evaluate its performance under different operating conditions. Additionally, particle trajectory simulations are conducted to assess the efficiency of particle separation based on varying cyclone design parameters. The findings of this study provide valuable insights into the design and optimization of cyclone separators for selective particle removal. The proposed methodology offers a systematic approach for engineers and researchers to develop cyclone systems tailored to specific particle size separation requirements, contributing to enhanced efficiency and productivity in industrial processes.

Keywords: Cyclone separators, Particle agglomeration, Computational fluid dynamics (CFD), Gas-solid separation, Pressure drop

Harnessing Hydrogen: A Comprehensive Examination of Fuel Cell Vehicles

Sounav Banerjee[a], Om Prakash[a] , LS Brar[a] , Vijay Kumar Pandey[a]

[a]Department of Mechanical Engineering, Birla Institute of Technology, Mesra, Ranchi, India, 835215

This book chapter offers a comprehensive analysis of the emergence and potential of hydrogen fuel cell vehicles (HFCVs) within the landscape of sustainable transportation. The ongoing discourse regarding automotive sustainability provides an insightful analysis and presents HFCVs as a promising alternative to electric vehicles (EVs) and internal combustion engines (ICEs). Highlighting the distinctive features of HFCVs emphasizes their utilization of hydrogen fuel to power electric motors. This inherent characteristic positions HFCVs as potentially carbon-neutral alternatives, capable of significantly mitigating greenhouse gas emissions compared to traditional ICEs and battery-powered EVs.Central to the discussion are the technological advancements driving the development of HFCVs, including advancements in hydrogen production, storage, and distribution infrastructure. Evaluates the efficiency, reliability, and environmental implications of HFCVs compared to other propulsion systems, shedding light on their viability as sustainable mobility solutions. Moreover, the chapter delves into the socio-economic factors shaping the adoption of HFCVs, such as regulatory frameworks, investment incentives, and consumer perceptions. It underscores the necessity of collaborative efforts among policymakers, industry stakeholders, and consumers to foster the widespread acceptance and integration of HFCVs into existing transportation networks. This chapter asserts that HFCVs offer a promising pathway toward achieving sustainability objectives in the automotive sector. However, it underscores that realizing their potential hinges on effectively addressing technological, infrastructural, and regulatory hurdles through collaborative efforts and innovative solutions. By embracing HFCVs, the automotive industry can spearhead the transition toward a cleaner, more sustainable future for global transportation systems.

Key words: Hydrogen Fuel cell, Sustainable Transportation, Automotive, Collaborative Solutions, Green Transportation