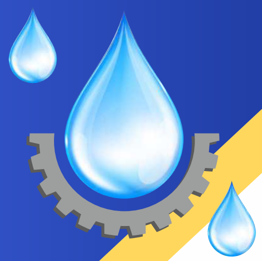
MANIPAL
ACADEMY of HIGHER EDUCATION
(Institution of Eminence Deemed to be University)

Sponsored by



International Conference on
Recent Advances in Fluid Mechanics
ICRAFM-2022

4th to 6th October, 2022

Organized by

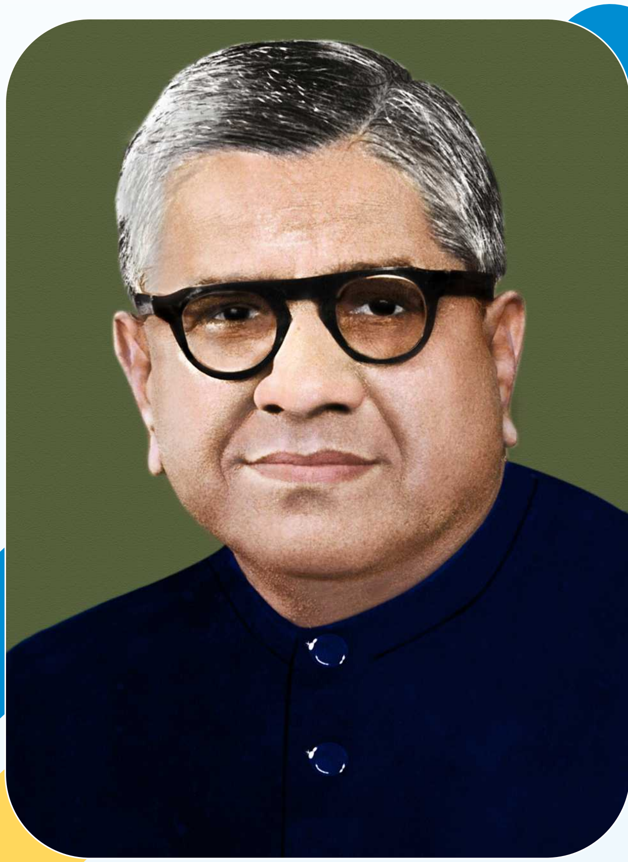
Department of Mathematics
Department of Aeronautical & Automobile Engineering
Department of Mechanical & Industrial Engineering

SOUVENIR

CSIR

Council of Scientific and Industrial Research

FOUNDER



Padmashri Awardee **Dr T M A Pai**

1898 -1979

CONTENTS

ABSID00114

Application of Fluid-Structure Interaction Technique for a Single Pad Externally Adjustable Fluid Film Bearing

Harishkumar Kamat[1], Chandrakant R. Kini[2], Satish B. Shenoy[2*]

[1]Department of Mechanical and Industrial Engineering,
Manipal Institute of Technology, Manipal Academy of Higher Education, Manipal, Karnataka, 576104,India
[2]Department of Aeronautical and Automobile Engineering, Manipal Institute of Technology,
Manipal Academy of Higher Education, Manipal, Karnataka, 576104,India
*Corresponding author [*]E-mail: satish.shenoy@manipal.edu

Abstract: This paper studies single pad externally adjustable fluid film bearing working under high-speed application using fluid-Structure interaction (FSI) technique. The pad can be adjusted in both radial and tilt directions in a controlled manner so that bearing performance can be enhanced. Adjustment in the pad helps in altering the clearance space between the rotor and the bearing. The numerical technique is validated against the already published literature, and the deviation in the results is less than 2%. Initially, computational fluid dynamic(CFD) is used to investigate the pad behaviour under the influence of fluid film pressure. The load carrying capacity of bearing, pad deformation, and stresses in the pad structures are evaluated. In addition, graphs are prepared for both one-way and two-way FSI and results are compared. The numerical findings show that due to large lubricant pressure, the load carrying capacity is superior at a higher eccentricity ratio. But, pad deformation and stresses are on the higher side, which is cause for concern.

Keywords: Externally adjustable pad bearing, Single pad, Fluid-Structure Interaction, Computational Fluid Dynamics, High-speed rotor

ABSID00120

CFD Analysis of Fly Ash Slurry Flow Across Horizontal and 90^0 Pipe Bend

Mukund Kumar[1*] and Gaurav Kumar[2]

[1]Department of Mechanical Engineering, N.I.T, Jamshedpur, Jharkhand-831014,India
[2]Department of Mechanical Engineering, N.I.T, Uttrakhand, Srinagar-831014,India
[*]Corresponding author E-mail: mukundkumar130@gmail.com

Abstract: Fly ash is a significant by-product of coal-fired power stations. Fly ash slurry transportation via long pipelines is a challenge for thermal plants and other industries. In the current investigation, numerical simulation has been performed to analyse the flow pattern of fly ash slurry via horizontal pipe and 90^0 horizontal bend pipe. The study of rheological flow properties of FA slurry is crucial for the development of its transportation system. It is found that pressure loss over the horizontal pipe and 90^0 horizontal bend pipe increase with velocity increase and solid concentration. The pressure loss over a 90^0 horizontal pipe bend is substantially larger than that of straight pipe.

Keywords: Slurry transportation, CFD, Rheology, Flow behaviour, Pressure loss

ABSID00128

Computational Fluid Dynamic Investigation of non-Newtonian Turbulent Duct-flow Subsequent to in-plane Double Bends

Arka Banerjee[1,2], Sayantan Sengupta[1,*] and Shantanu Pramanik[1]

[1]Department of Mechanical Engineering, National Institute of Technology Durgapur, Durgapur-713209(W.B.), India.
[2]Department of Mechanical Engineering, Dr.B.C.Roy Engineering College, Durgapur-713206,(W.B.),India.
[*]Corresponding author, E-mail: sayantan.sengupta@me.nitdgp.ac.in

Abstract: In engineering applications, fluids are often subjected to flow through bend or bend combinations. Different applications of the Newtonian or non-Newtonian fluid flow in curved ducts can be seen in chemical plants, hydro power stations, automobiles, paper industries, and oil and gas transportation. Besides industrial applications, the study of curved-duct flow is equally essential to understanding the flow behaviour in veins, arteries and other channels in mammals. So, it finds its applications in medical sciences, where in many cases, the fluid is non-Newtonian. Flow in different kinds of single bends has been extensively studied both experimentally and numerically for the last few decades; in contrast, studies on flow through bend combinations specifically for rheological fluids are limited in numbers. A fluid is subjected to Prandtl's secondary flow of the second kind when the main flow is skewed due to a bend or bend-combinations. In the present study, an attempt has been made to analyse this particular secondary flow behaviour of non-Newtonian fluids (both shear thickening and shear thinning)passing through an in-plane double bend. In our computational fluid dynamics(CFD)simulations, the turbulent flow is predicted by the standard k-ε turbulence model with the scalable wall function. The numerical results are validated against the benchmark experimental results. For parametric study in a pipe with a circular cross-section, the curvature ratio (R_c), Reynolds number(Re)and flow behaviour index(n)are varied systematically. It is demonstrated here that the secondary flow is greatly influenced by the inlet Reynolds number and the curvature ratio of the double-bend. Change in average velocity due to the bend is measured by a newly defined dimensionless parameter called enhancement ratio. Major integrated output parameters, viz. non-dimensional pressure loss and enhancement ratio, have been determined at various Re, R_c and n. The enhancement ratio is found to be decreasing within creasing Re, R_c and n. It has also been observed that the non-dimensional pressure loss inside bend decreases with increasing Re,R_c and n. It is realised that the reduction of enhancement ratio is related to the corresponding fall in non-dimensional pressure loss. Contours of normalised primary velocity as well as normalised secondary velocity are presented at different sections inside the double-bend. Non-dimensional pressure distribution at bend down stream has been determined for different n. Our CFD model makes it possible to capture the spatial changes of the local vortical patterns in the three-dimensional flow field. Our CFD solutions nicely capture the decay of the in-plane flow(or secondary flow)and concurrent re-establishment of primary flow down stream of the bend.

Keywords: Double bend, Non-Newtonian, CFD, Secondary flow, Turbulence, Enhancement ratio

ABSID00129

Improved Design of Solar Desalination System

BJM Rao[1]*, K.V.N.S.Rao[2], Mihir Barman[3]

[1]Associate Professor, Department of Mechanical Engineering,
Vignan's Foundation for Science, Technology & Research (Deemed to be University) Andhra Pradesh State, India,
[2] Professor, Department of Mechanical Engineering, B.V. Raju Institute of Technology, Narasapur, Telengana State, India.
[3]Assistant Professor, Department of Mechanical Engineering, Vignan's Foundation for Science, Technology & Research (Deemed to be University)Andhra Pradesh State, India.
*Corresponding author Email: bjaganmohanr@gmail.com

Abstract: As 97% of the world's water is salt water in the seas, it is doable to address the lack of water issue with seawater desalination as it were. There is a need to utilize earth environmentally-friendly energy sources so as to desalinate seawater. Right now new structure is proposed for a local sunlight based desalination procedure to change over salt water into fresh water. In the majority of existing little scope sunlight based desalination frameworks, static water is presented to sun rays for desalination process. Be that as it may, in the current framework, water is made to move through the channels during desalination process. The maximum fresh water production rate is enhanced to 8.16L/ (m^2 day) against 4-6 L / (m^2 day) with conventional solar stills. Water investigation after and before the desalination procedure are thought about and results are above worthy level.

Keywords: desalination; solar energy; water flow channels; conversion of salt water; solar still

ABSID00134

Fixed Volume Computational Study of Positive Displacement Blower

Sourabh S.Varne[1], Rajsekhara Reddy Mutra[2],T Gopalakrishnan[3*]

[1,2]School of Mechanical Engineering, Vellore institute of technology, Vellore
[3]Department of Information Technology, Manipal Institute of Technology, Bengaluru,
Manipal Academy of Higher Education, Manipal, India
*Corresponding author Email: varnesaurabh@gmail.com, Rajasekhara.reddy@vit.ac.in, gopalakrishnan.ct@gmail.com

Abstract: Positive displacement blower like twin lobe and tri lobe blower are under this category. They work very well in moderate pressure measurements and work very well. Alternatively, the flow inside the rotary blower/pump also known as roots blower can be obtained through computational fluid dynamics techniques. Accordingly, the assembly of this blower has been studied and a solid model was created. Flow passage of suction, discharge, casing comprises lobes are extracted. They are used for volume flow rate in various industries. These blowers having rate of capacities10m^3/hr to 200m^3/hr for pressures limit 800mbar in single stage construction. Due to presence of small clearances of the order~2-3mm between lobe to wall and lobe to lobe, domain discretization with hexahedral elements is made to capture flow in the clearance regions. The volume mesh has imported to flow solvers to simulate compressible flow in steady state for studying slip ages which is leakage flow and pressure fluctuations. With the flow/thermal conditions such inlet stagnation pressure, exit static pressure, lobe rotations, and etc. Leakage mass flow rate and pressure fluctuation was obtained through three-dimensional flow simulations.

Keywords: Computational fluid dynamics, Mesh generation, Lobe, mass flow rate, blower, Contour, Rotational speed.