CFM

COMPUTATIONAL FLUID MECHANICS

cfm@bahria.edu.pk

The CFM group is a well organized leading research cluster that is working on modeling and simulation of heat and fluid flow phenomena through latest computational techniques. Major contribution of this research group can have a significant role at the industrial level, particularly: in combustion, heat exchanger, automobile, aerodynamics, airfoil, laminar and turbulent flow, nanotechnology, thermodynamics etc. For simulation, latest techniques are under consideration for instance: Finite Element Method (FEM), Control Volume finite element method (CVFEM), Finite difference method (FDM), immersed boundary method, Least square Galerkin method, collocation method, Homotopy analysis and perturbation method, domain decomposition method, etc. Apart from the national level collaboration, this group is working with well-known researchers working at Internationally renowned institutions/research centers. The researchers who are collaborating with this group belong to following countries: Australia, China, Turkey, United Kingdom, UAE, Malaysia, Morocco, Saudi Arabia, Iran, Georgia and Canada. The aim of this group is to develop the mathematical models and simulation of complex nature engineering problems based on latest techniques.

OBJECTIVES

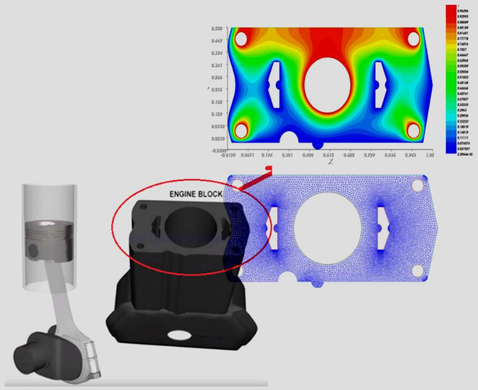
- To become the leading Academic Research Center in the field of Applied Mathematics.
- To construct solutions of complex nature engineering problems related to heat & fluid flow.
- Development of geometries and physical bounds of the problem according to the constitutive equations for both Newtonian and non-Newtonian fluids.
- To Implement new methods in the computational domain for instance: Finite Element method, Finite difference method, Control volume finite element method, boundary element method, homotopy analysis method.
- To produce theoretical work that can be compared and implemented with experimental one at various mechanical and structural engineering branches across the world.
- To construct the scientific research center that involves specifying the fluid flow and heat transfer behavior and properties at all bounding surfaces of the fluid domain.

CONSULTANCY SERVICES

Based on the objectives defined for CFM group, the focus of this group is to provide the modeling and simulation of in the field of heat transfer, thermal management, fluid interaction with solid structure, air foil and many other engineering designs in following manner:

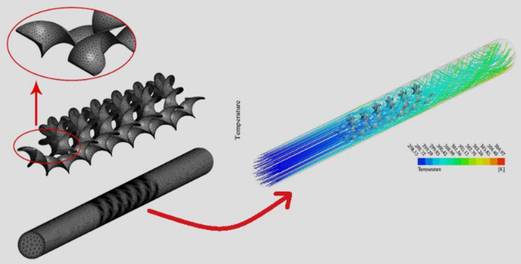
- Mathematical design based on fundamental laws of physics and the derivation of constitutive equations for heat and fluid flow domain.
- Algorithm development and simulation of model that is based upon system of ordinary and partial differential equations.
- Develop the mathematical design of heat transfer model for various shape in the presence of nanoparticles and airfoil modeling and simulation for various nature of wings.

RESEARCH & SERVICES PORTFOLIO



Modeling and Simulation at Surface of Engine Block

With the recent demand of improved efficiency, energy saving, reduction of pollution of cars, research on the high output and the miniaturization of engines has been performed vigorously. According to such trends, the output of engines have been increasing and engine structures have been placed in the thermal loading conditions that are much more excessive compared to the past. When the high-output engine gets into thermal loading condition, the temperature of the wall surface of the combustion chamber increases and the oil viscosity of the engine decreases. Due to the oxidation of the engine oil and others of the like, there is a danger of the ignition plugs and the valves melting down or exploding. Such thermal loading has a bad influences on the engine like, knocking, early ignition, the transformation of the cylinder bore, scuffing of the ring, transformation of the left side of the valve, etc. Therefore, the force exerted on the piston is reduced. Thereby, the reduction of the output of the organ and the lowering of the heat efficiency take place. As a result, the engine must operate within a reasonable temperature range. For this analysis, the clear and definite evaluations of such thermal loading must take place.

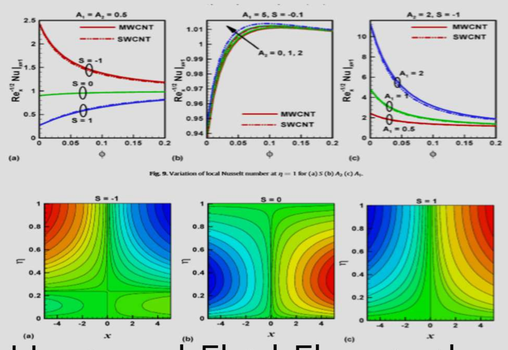


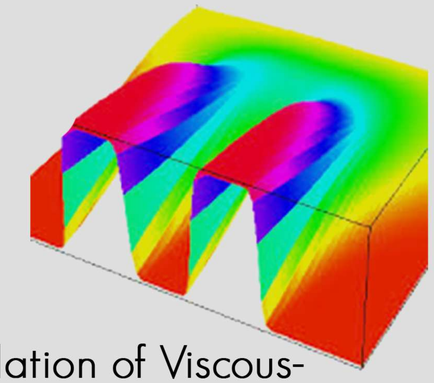
Heat Transfer in Innovative Turbulator with Twisted Tape

Heat exchangers are used in many areas of the industries. Hence, the use of high performance heat exchangers is very important for saving energy. Twisted tape swirl turbulator is one of the commonly used passive types for heat transfer augmentation due to their advantages of steady performance, simple configurations and ease of installation. It generates swirling flow and causes improved fluid mixing between central region and the near wall region, therefore, the heat transfer in tubes can be enhanced by fluid mixing.

Analysis of Fluid-Particle Interaction in the Nanofluid

Fluid flows can be found in nature and are broadly observed by those individuals who monitor nature. In nature, almost every fluid contains small particles which is why fluids in which solid spherical particles are distributed are named as dusty fluids. In many industrial processes nanofluids are required since they have the ability to enhance the thermal conductivity of the fluid. Accordingly, in this project we focus on the flow of dusty Nano fluid in a channel or on a continuously moving surface with linear and non-linear velocity of wall.





Heat and Fluid Flow in the Presence of NanoParticles

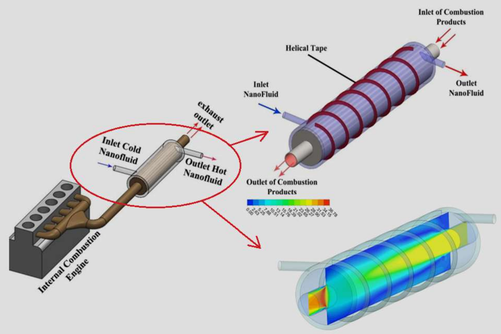
This project focuses on heat transfer due to addition of nanoparticles in the working fluid. Heat transfer has played a vital role in numerous industrial applications including in-vehicles and avionics, electronic equipment dissipates heat which needs cooling, heating and cooling system in buildings, industrial processes, Petrochemical industries, textile, food and other plants. Fluids are best source to remove, add or transfer of heat. Suspension of the nanoparticles in a working fluid is one of the best ways to enhance the heat transfer. Nanofluids are studied due to their heat transfer properties. Heat transfer augmentation is an essential issue in energy saving aspects. Conventional fluids like (e.g. water, ethylene and engine oil) have low thermal conductivities than solids. For example, thermal conductivity of Cu is 700 times greater than water and 3000 times greater than engine oil. Since nanoparticles are small in size and have large specific area, nanofluids have superior properties like thermal conductivity, minimum clogging in flow passage, long-term stability and homogeneity.

Simulation of Viscous-Quantum-HydroDynamical Models in Semiconductor

Modelling and numerical simulation of semiconductors is necessary for the development of new industrial devices to increase their efficiency and reduce their manufacturing cost and time. In this project,the viscous-quantum-hydrodynamic model of the semiconductor devices is analyzed. In this model, the governing equations for electron and current densities are the Euler equations together with a viscous and quantum correction terms and the electrostatic potential is given by the Poisson equation. The model equations are highly nonlinear in nature which require efficient and accurate numerical methods to get physically realistic solutions. Several state-of-the-art numerical methods such as kinetic approaches, finite volume schemes (FVS) and finite element methods (FEMs) have been used to investigate the model. For validation of the obtained numerical results, different test problems have been performed.

Heat Exchanger for Heat Recovery System

The effect of different swirler geometries on the flow dynamics inside a micro gas turbine combustion chamber model are widely used in automobiles. Designing air swirler to produce stable and efficient combustion with low-pressure losses is a challenge. A majority of heat exchangers used in thermal power plants, chemical processing plants, air conditioning equipment, and refrigerators, petrochemical, biomedical and food processing plants serve in heating and cooling of different types of fluids. Both the mass and overall dimensions of heat exchangers employed are continuously increasing with the unit power and the volume of production. This involves huge investments annually for both operation and capital costs. Hence, it is an urgent problem to reduce the overall dimensional characteristics of heat exchangers; and the need to optimize and conserve these expenditures has promoted the development of efficient heat exchangers.



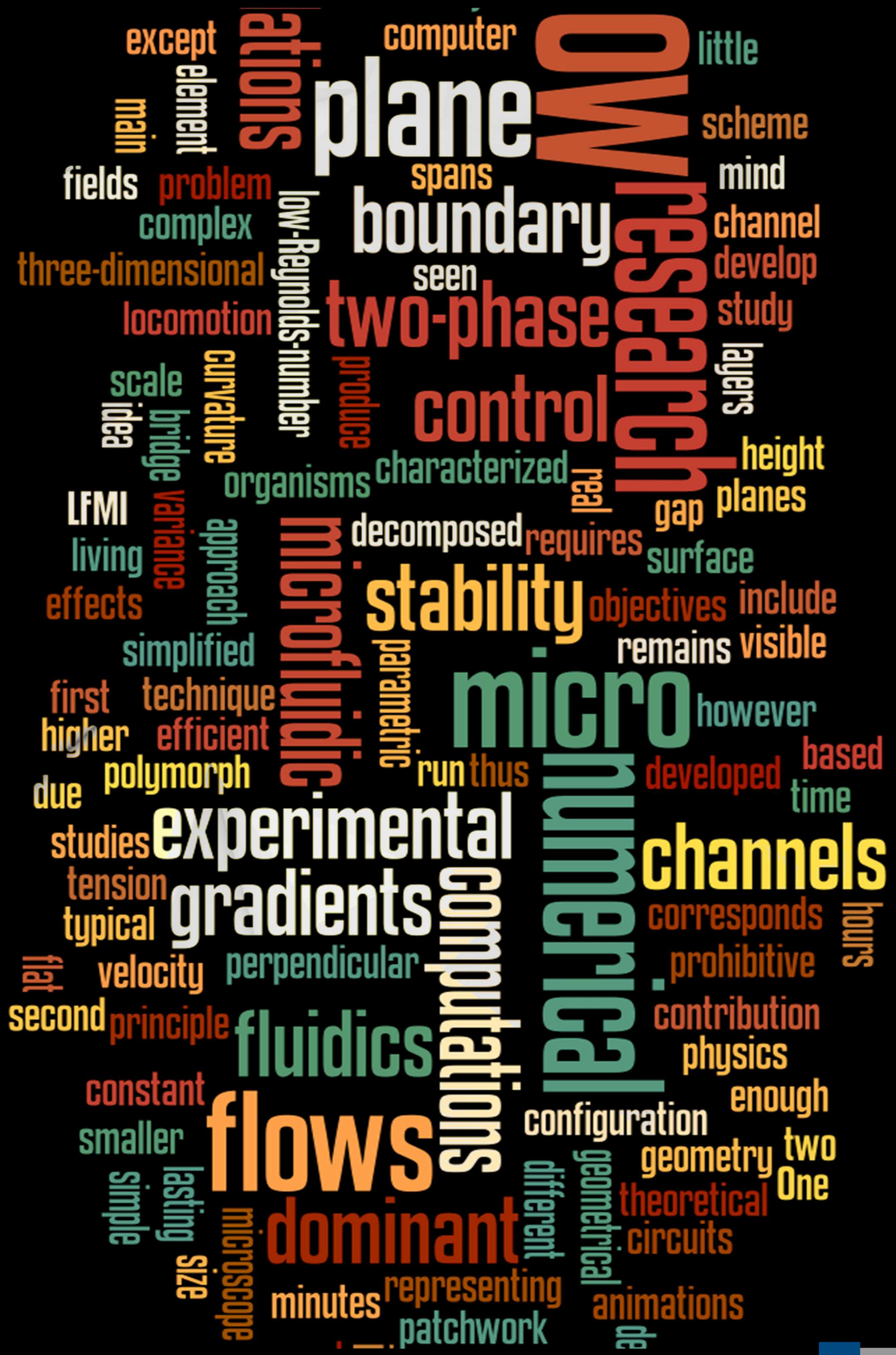
KEY RESEARCHERS

Prof. Dr. Muhammad Ramzan

Prof. Dr. Muhammad Ramzan is a Senior Professor in the Department of CS. He is also the head of the research group "Computational Fluid Mechanics". His research area includes Wave motion, Wiener Hopf technique, Fluid Dynamics, nanofluid, heat transfer, numerical analysis, Boundary layer flow, channel flow Homotopy analysis method, Optimal Homotopy analysis method, shooting technique. He has an experience of 24 years in teaching and research. He has also published 63 ISI Indexed journal papers with an accumulative impact factor 150. His paper received "Best Paper Award" at 7th ICAT, Antalya. He was also the declared "Best Presenter" at 4th ICONTES, Kuala Lumpur. Further, he is the recipient of HEC Indigenous Scholarship for Ph.D. studies. He also won certain HEC research projects worth more than 1 Million. He is also a Visiting Professor at Jiangsu University, China; and has strong research collaboration with Sejong University, South Korea, Jiangsu University, China, and Majmaah University, Saudi Arabia.

Dr. Rizwan Ul Haq

Dr. Rizwan Ul Haq is a Sr. Asst. Professor in the Department of EE. His majors are: Fluid Dynamics, nanofluid, heat transfer, numerical analysis, Boundary layer flow, Cavity flow models, channel flow, Peristaltic flow, Homotopy analysis method. Adomian Decomposition method, Finite difference method, shooting technique, Finite element method. He has worked in Spencer Engineering Building, Western University, Canada where he developed a test for numerical algorithm for fluid over a corrugated surface. He also worked in Moulay Ismail University, Errachedia, Morocco. So far, he has published more than 84 impact factors ISI publications. The total impact factor is more than 155; and total citations of his entire publications are 2765. The h-index is 29 and i-10-index is 61. He has also achieved Research Productivity award and Productive scientist of Pakistan He was also the recepient of Best Researcher Award of BUIC in 2018. Furthermore, he has received 7 awards in his career.

Dr. Jafar Husnain

Dr. Jafar Hasnain is working as a Sr. Assistant Professor in the Department of Computer Sciences, BUIC. His majors are: Fluid Dynamics, multiphase flow, nanofluid, chemically reactive fluids, heat transfer, numerical analysis, Boundary layer flow, channel flow, Homotopy analysis method, perturbation method, shooting technique. He has gained research experience from the Department of Engineering Science, University of Auckland, New Zealand and the Department of Mathematics and Statistics, Macquarie University, Australia. So far, he has published more than 12 impact factor ISI publications. The total impact factor is more than 15. He is also the recipient of HEC Indigenous Scholarship for Ph.D. studies. He further won HEC research projects of worth 0.5 million. He also has collaborations with developed countries like: Romania, Saudia Arabia. Currently, 6 MS mathematics students are under his supervision in BUIC and also has 2 Ph. D students under his co-supervision at The Islamia University of Bahawalpur.