Track 3: CFD Applications

Sponsors: Fluids Engineering Division

Topic 3-1: Computational Modeling in Hydro- and Aero- flow dynamics (CFDTC)

Organizers:

Haibo Dong: hd6q@virginia.edu
Chengyu Li: chengyu.li@villanova.edu
Shanti Bhushan: bhushan@me.msstate.edu

Descriptions:

The goal of the hydro- and aero flow dynamics topic is to provide a platform for presenting results and disseminating recent research and developments in marine hydrodynamics and or aerodynamics. Areas of interest for this symposium include, but are not limited to, the following topics: (1) computational and experimental marine hydrodynamics; (2) resistance and propulsion; (4) seakeeping and maneuvering; (5) waves; (6) under water explosion; (7) hull form optimization; (8) fluid-structure interaction; and (9) hydro-acoustics. Authors and presenters are invited to participate in this event to expand international cooperation, and to explore outstanding and frontier problems in marine hydrodynamics for further research and applications.

Topic 3-2: Computational Turbulent Combustion (CFDTC)

Organizers:

Chaitanya Ghodke: chaitanyaghodke1@gmail.com
Leitao Chen: lchen1@tnstate.edu

Descriptions:

The interest in the field of numerical simulation of reactive flows has grown considerably in the last decade or so. Computational modeling has now become an important design tool to study complex combustion physics in many engineering applications including internal combustion engines, gas turbines, power plants, boilers and small-scale burners. However, owing to the complex nature of the problem, simulating multi-scale physics in turbulent reactive flows in detail is still a challenge. This topic solicits research work covering numerical modeling of combustion especially in the field of unsteady turbulent combustion should be presented. The authors are encouraged to submit any fundamental and/or applied work covering recent progress in the area of computational turbulent combustion.

Topic 3-3: Open Source CFD (CFDTC)

Organizers:

Shanti Bhushan: bhushan@me.msstate.edu
Chengyu Li: chengyu.li@villanova.edu

Descriptions:

Computational Fluid Dynamics (CFD) has seen a surge in its usage from research-only utilization to industrial-scale research and optimization. It is here that the algorithms refined during research come to fruition. However, a large number of industries are small or medium-sized enterprises and can not afford to pay the high licensing fee some softwares demand. Open Source CFD softwares, therefore, are starting to become popular. This topic invites researchers to submit original work- research or industrial oriented, where any Open Source CFD package has been used to provide solution to a CFD related problem.

Topic 3-4: CFD for Nuclear Thermal Hydraulics (CFDTC)

Organizers:

Victor Coppo Leite: victor.coppoleite@inl.gov
Elia Merzari: ebm5351@psu.edu

Descriptions:

Authors are invited to submit advancements in code development or analyses utilizing the full fidelity spectrum of CFD targeting nuclear thermal hydraulics. Works involving CFD and related analysis methods in reactor cores are also welcome, including those requiring coupling with neutron transport and/or system-level simulations.

Topic 3-5: Coupled Multiphysics Simulation (CFDTC)

Organizers:

Haoxiang Luo: haoxiang.luo@vanderbilt.edu
Lakshman Anumolu: anumolu@uwalumni.com
Mahbobur Rahman: mahbobur@vt.edu

Descriptions:

In the real world, a fluidic system often involves multiple physical process which interacts in complex and interesting ways. The fluid fields can significantly affect the chemical reactions, biological transportation, heat exchange, electromagnetic system, structural mechanics and so on. Therefore, it is important to develop coupled multi-physics computational methods to simulate and study these complicated phenomena, not separately. This topic encourages researchers to simplify the real systems, construct multi-physics models and conduct numerical simulations to investigate the flow behavior and how it influences the coupled process such as in chemical engineering, energy industry, bio- and micro-fluidic applications, etc.

Topic 3-6: Exascale Fluids Simulation (MFTC, CFDTC)

Organizers:

Yupeng Xu: yupeng.xu@netl.doe.gov
Jingsen Ma: jingsen.asme@gmail.com

Descriptions:

Exascale computing represents a significant leap in computational power, capable of performing a quintillion (10^18) calculations per second. With exascalecomputing, it becomes possible to run high-fidelity simulations that can capture a broader range of scales, from microscopic turbulence to large-scale atmospheric dynamics. This capability is crucial for advancing our understanding of fluid behavior, optimizing designs in engineering, and making more accurate predictions in various industries. Furthermore, exascale computing accelerates the pace of innovation by reducing the time needed for simulations, enabling faster iteration and development cycles. For example, MFIX-Exa, developed at National Energy Technology Laboratory, is a high-performance, massively parallel, machine agnostic multiphase flow code for the exascale computing era.This topic is seeking papers in the area of exascale simulation of multiphase flow and other complex multi-scale fluids simulations.