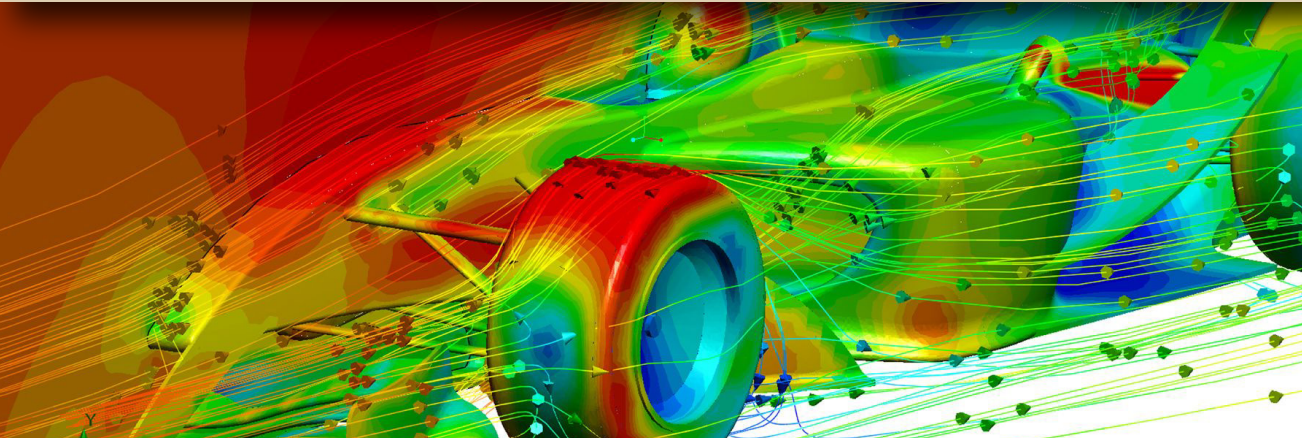




A Hands on introduction to Computational Fluid Dynamics (CFD)



Computational Fluid Dynamics (CFD) has now become a common tool for modelling heat and fluid flow applications in many engineering disciplines. Obtaining accurate results through proper use of CFD requires a solid understanding of underlying modelling principles with technical know-how gained through experience. The proposed workshop aims to disseminate the required basic theoretical understanding and technical know-how among researchers of various disciplines. The workshop is designed to be largely with hands on sessions running through a full day with solving several real life engineering problems. As the main outcome of the workshop, it is expected to promote the interdisciplinary collaborations through research on multidisciplinary, multi-physics CFD modelling exercises.

Speakers

Dr. RACP Ranasinghe

Department of Mechanical Engineering
University of Moratuwa, Sri Lanka

Dr. NAID Nissanka

Department of Mechanical Engineering
University of Moratuwa, Sri Lanka

Target audience

- Research students, early career researchers, practicing engineers in different engineering fields

Workshop Programme

Session	Activities
Lecture 1 30 min	Potential of CFD: Applications and Basics
Hands on Session 1 1 hour	Performing a CFD Simulation: Work Flow
Hands on Session 2 1 hour	Techniques and Best Practices of Meshing
Hands on Session 3 1 hour	Modelling Turbulence Flow and Heat Transfer
Lecture 2 30 min	Benchmarking, Verification and Validation

Aim

- To provide essential knowledge and skills required by beginners of various engineering/research disciplines to use CFD as a modelling tool

Workshop Objectives

- To provide an insight into CFD and its applications in diverse engineering disciplines
- To provide the basic knowledge on underlying theoretical principles of CFD modelling
- To provide essential hands-on skills required for effective use of a CFD software packages

>>

- Hands on Session will be conducted as guided tutorials with interactive discussions on theoretical aspects of each modelling step.
- Activities will cover setting up problems, solving and visualization of results.
- Case studies cover a range of real-life problems from many engineering disciplines.
- Soft versions of workshop materials will be provided
- Ansys Fluent is used as the software package for modeling for the convenience of beginners. Student version of this software is freely available.
- However, the content covered is equally applicable for any CFD package, including popular open-source codes such as Open FOAM and Code Saturn.

Expected outcomes

- Promote multidisciplinary and multi-physics research through the use of CFD as a modeling tool.
- Unveil the potential of CFD among researchers of different disciplines and pave a path for identifying potential future collaborations

20 Participants Only

Participation is FREE

WORKSHOP 1

8th International Multidisciplinary Engineering Research Conference

29th
July 2022

1.30 pm -
5.30 pm

