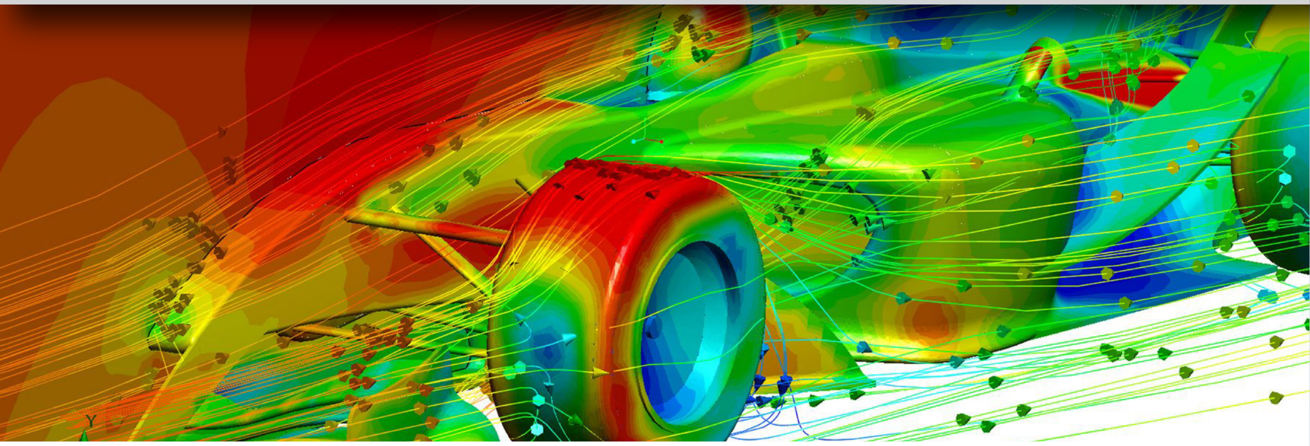
WORKSHOP 2

MERCon 2021
Moratuwa Engineering Research Conference

A Hands-on Introduction for Computational Fluid Dynamics (CFD) Promoting Multidisciplinary Research



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 underling 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 researches of various disciplines. The workshop is designed to be largely with hands on sessions solving real-life engineering problems.

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 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

Aim

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

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

>>

- 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 form 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 researches of different disciplines and pave a path for identifying potential future collaborations

Participation is **FREE** of charge !

Register @: http://mercon.mrt.ac.lk/workshops.html

WORKSHOP 2

7th International Multidisciplinary Engineering Research Conference

29th July 2021

1.30 pm - 5.30 pm

Online WORKSHOP