Course Overview

Due to world population growth, growing industrial development, improvements in living standards, emerging technologies (such as electric vehicles) and the growth in domestic and industrial use of modern equipment, the current growing trend in energy demand and consumption will continue for long into the future. Much of the energy needed by most countries is produced via combustion oriented processors. Pulverised coal power plants, combined cycle and other gas turbine systems, oil powered power generation systems and stationary diesel power generation systems are widely used in this context. Air transport which also relies on combustion has been growing at a steady rate during the last decade. In addition, majority of other transport systems still relies on internal combustion engines. These include rail, shipping and road vehicles used in day-to-day activities. All these systems contribute to atmospheric carbon dioxide levels and other pollutants are a major concern all around the word. The world has recognised the need for the reduction of CO_2 and pollutants. With regard to combustion systems there is still a need to improve efficiency and reduce pollutants such as particulates, NOx and other emissions. Driven by regulatory requirements many industries have taken steps to improve combustion systems and reduce harmful emissions. Such efforts requires research, tools and facilities for the development of improved combustion systems.

Development of efficient and improved combustion processes through experimentation is very time consuming and expensive. Computational Fluid Dynamic (CFD) based combustion modelling techniques are now widely used in the development phase of such improved combustion systems and it has become a vital tool in the research and development cycle. However the use of CFD for the modelling of combustion requires a good understanding of how combustion models are formulated and their limits and applicability for different types of combustion systems. In this lecture series fundamental of combustion modelling using CFD is introduced and recent advances in combustion modelling are presented. Compared to simple combustion models which have been used in early CFD based simulations, modern combustion modelling techniques are much more realistic and accurate. Moreover the developments in Large Eddy Simulation (LES) techniques and other advances have been combined with combustion modelling methodologies to provide very useful simulations for the improvement of combustion system. In this lecture series fundamentals of these advanced modelling techniques are presented with details and their applications are demonstrated in a range of combustion related processes including the modelling of, premixed and non premixed flames, IC engines and safety related explosion situations.

Course Content

- Fundamentals of CFD and governing equations
- Introduction to CFD modelling – RANS techniques, CFD algorithms, discretisation, LES techniques
- Fundamentals of combustion, premixed and non-premixed concepts. Combustion chemistry, stoichiometry, combustion regimes.
- Governing equations used in combustion modelling and coupling with CFD techniques. Additional models required for the modelling of different combustion systems. Non-premixed combustion models and premixed combustion models.
- Advanced combustion modelling techniques. Non-premixed: Pdf models, laminar flamelet based model, FGM methodology, FPV and UFPV methods.
- Advanced combustion modelling techniques. Premixed: Pdf models, laminar flamelet based models, FGM methodology, FSD and DFSD models.
- LES based combustion models: Premixed and Non-premixed combustion models
- Application of LES based non-premixed combustion models for the simulation of flames and other combustion devices.
- Application of LES based premixed combustion models for the simulation of internal combustion engines other safety related situations
- Applications of LES techniques in combustion simulations in non-premixed and partially premixed flames

Objectives

The primary objectives of this short course are as follows:

- Introduce the participants to fundamentals of Computational Fluid Dynamics (CFD).
- Provide exposure to various types of combustion processes.
- Provide an understanding of how combustion models are formulated in CFD methodology.
- Introduce RANS and LES based combustion modelling techniques
- Demonstrate application of RANS and LES based advanced combustion models.

GIAN
Global Initiative of Academic Networks

Ministry of Human Resource Development, **Government of India**

5 Days* course on

Advanced Combustion Modelling with Computational Fluid Dynamics

(*Lectures: 20 hours and Practical Sessions: 10 hours)

April 8-12, 2019

Course Instructor

Prof. Weeratunge Malalasekera

Professor of Computational Fluid Dynamics and Heat Transfer,
School of Mechanical, Electrical and Manufacturing Engineering,
Loughborough University, UK

Course Coordinator

Dr. Shanmugam Dhinakaran

Department of Mechanical Engineering, IIT Indore, INDIA

Course Website: http://people.iiti.ac.in/~sdhina

The Centre for Fluid Dynamics

Department of Mechanical Engineering

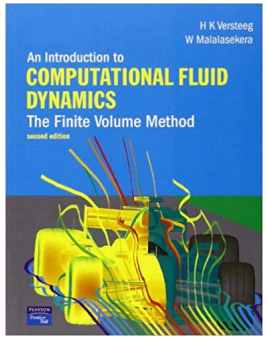
INDIAN INSTITUTE OF TECHNOLOGY INDORE

Simrol, Indore, INDIA

Teaching Faculty

Prof. Weeratunge Malalasekera is a Professor of Computational Fluid Dynamics & Heat Transfer at the Loughborough University, United Kingdom. Prof. Malalasekera maintains good collaborative links with a number of internationally leading research groups. His research interests include modelling and simulations of IC engines; Investigations into the performance of CCGT power plants; Development and investigations into energy storage solutions; Investigation into hydrogen applications, combustion, deflagration and flame propagation in hydrogen mixtures and Large eddy simulations of premixed and non-premixed flames.

Prof. Malalasekera contributes to all levels in undergraduate teaching and runs and contributes to the M.Sc Programme in Mechanical Engineering. Computational Fluid Dynamics and Heat Transfer are the main subjects taught. He is the co-author of the textbook entitled 'An introduction to Computational Fluid Dynamics: The Finite Volume Method'. First published by Longman Higher Education in 1995, this book as become a widely used popular CFD course text at many universities worldwide. An enhanced second edition of this book covering advanced topics and recent development in CFD has been published in February 2007, by Pearson Higher Education. By popular demand, a number of international editions (Chinese, Asian, Japanese, Greek and Korean) have also been published.



The widely adopted text book of Prof. Weeratunge Malalasekera on Computational Fluid Dynamics

Course Coordinator

Dr. Shanmugam Dhinakaran is an Associate Professor at the Department of Mechanical Engineering, Indian Institute of Technology Indore, India. He received his PhD in the area of Computational Fluid Dynamics and Heat Transfer from IIT Kharagpur, India in 2008. Before joining IIT Indore as an Assistant Professor in 2012, he has worked as a post doctoral researcher at the Université de Pau et des Pays de L'Adour, France; Universidade do Minho, Portugal; Faculdade de Engenharia da Universidade do Porto, Portugal and Université de Valenciennes et du Hainaut-Cambrésis, France.

Dr. Dhinakaran is also an adjunct faculty in the Department of Biosciences and Biomedical Engineering, IIT Indore. He is the coordinator of The Centre for Fluid Dynamics, IIT Indore. Bluff body flows; Non-Newtonian fluid flows; Heat transfer in Porous media; Nanofluids and Biofluid Mechanics are his research interests.

Who should attend?

- **Prerequisite**: A basic understanding of Computational fluid dynamics, combustion, and thermodynamics is expected.
- Executive engineers and researchers from academia, industry and government organizations including R&D laboratories with a background in Aerospace, Automobile, Mechanical, and Chemical Engineering.
- Students at advanced levels (B.Tech/M.Sc/M.Tech/ Ph.D) or faculty members from reputed academic institutions and technical institutions.

Examination & Certificate

An examination will be conducted at the end of the course and grade sheet as well as participation certificate will be give to all the participants.

Travel Information

IIT Indore is located at central part of India in Indore City . For more information, please visit the course website.

Registration Fee

UG & PG Students	Rs. 5,000
Research Scholars	Rs. 10,000
Faculty members	Rs. 15,000
Industry, R&D Organizations	Rs. 30,000

The above fee include all instructional materials; computer use for tutorials and assignments; laboratory equipment usage charges; 24 hr free internet facility. Fee concession may be considered for individual having limited financial support

How to Register?

1. Send an e-mail to the coordinator (sdhina@iiti.ac.in) expressing your interest and wait for acceptance.
2. If accepted, pay the relevant fee online and send the details to the course coordinator.

Important dates and venue

Last date for Registration	April 5, 2019
Course schedule	April 8 - 12, 2019
Venue	IIT Indore, Indore, India

Accommodation

Limited paid accommodation may be provided to participants on first-come-first-serve basis depending on availability. Participants are advised to make their own arrangement if accommodation is not available.

Contact Details

Almost all the information regarding eligibility, fee payment, travel information, accommodation, etc., are available in the course website. If you have any other queries, you may write to or call the course coordinator.

Dr. Shanmugam Dhinakaran
Associate Professor
The Centre for Fluid Dynamics,
Department of Mechanical Engineering
Indian Institute of Technology Indore,
Khandwa Road, Simrol, Indore, - 453 552
Email: sdhina@iiti.ac.in, **Mobile**: +91-9111-74-9191
Course Website: http://people.iiti.ac.in/~sdhina