



COMPUTATIONAL FLUID DYNAMICS USING FINITE VOLUME METHOD

PROF. KAMESWARARAO ANUPINDI

Department of Mechanical Engineering
IIT Madras

PRE-REQUISITES : Courses on linear algebra, numerical methods, incompressible fluid mechanics, and heat transfer

INTENDED AUDIENCE : Bachelors, masters and doctoral students, practicing scientists and engineers from industry.

INDUSTRIES APPLICABLE TO : General Electric, General Motors, ANSYS, ISRO, DRDO

COURSE OUTLINE :

In this course, the discretization and solution of diffusion, convection-diffusion, and incompressible fluid flow equations are discussed. Finite volume method is used to discretize each of the equations. The discretization and solution methods are formulated on structured as well as unstructured meshes. This course involves hand-calculations on simple meshes as well as numerical programming of the algorithms discussed.

ABOUT INSTRUCTOR :

Prof. Kameswararao Anupindi is an Assistant Professor in the Department of Mechanical Engineering at IIT Madras (IITM), Chennai. He obtained his Ph.D. from Purdue University in 2013. Prior to joining IITM, he worked as a post-doctoral researcher at Oak Ridge National Laboratory and at University of Southampton. His research interests include computational fluid dynamics, turbulence modeling and lattice Boltzmann methods.

COURSE PLAN :

- Week 1:** Review of governing equations
- Week 2:** Classification of governing equations
- Week 3:** Overview of numerical solution methods
- Week 4:** Steady diffusion equation on structured meshes
- Week 5:** Unsteady diffusion equation on structured meshes
- Week 6:** Diffusion in unstructured meshes
- Week 7:** Convection and diffusion
- Week 8:** Higher-order schemes
- Week 9:** Convection and diffusion on unstructured meshes
- Week 10:** Linear system solvers
- Week 11:** Incompressible flow field calculation
- Week 12:** Staggered and co-located formulation