

Introduction to Computational Fluid Dynamics (CFD) – Part II

Hands-on Experiments/Lab Demonstrations

Prepared by:

Dr Milinkumar Shah

Senior Lecturer, Chemical Engineering Discipline, Curtin University,
Western Australia

ELECTRONIC WARNING NOTICE FOR COPYRIGHT STATUTORY LICENCES

WARNING

This material has been reproduced and communicated to you by or on behalf of **Curtin University** in accordance with section 113P of the *Copyright Act 1968* (**the Act**)

The material in this communication may be subject to copyright under the Act. Any further reproduction or communication of this material by you may be the subject of copyright protection under the Act.

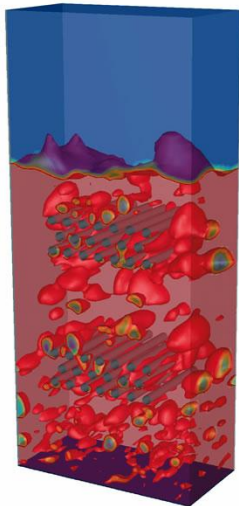
Do not remove this notice.

Outline

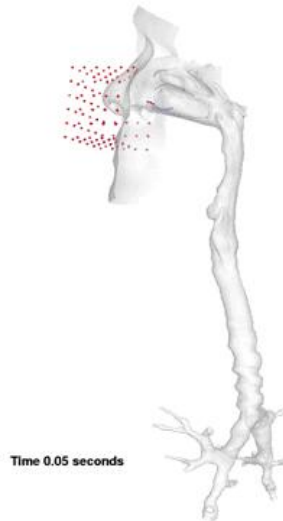
- CFD simulation tools
 - ANSYS-Fluent
 - OpenFOAM
 - ANSYS-CFX
- Hand-on practice using ANSYS-Fluent
 - Lab-1, Lab-2 → Simple pipe flow
 - Lab-3, Lab-4 → Flow in mixing elbow
- Discussion on best practices
- Homework → Lab-5 and Lab-6

What is CFD?

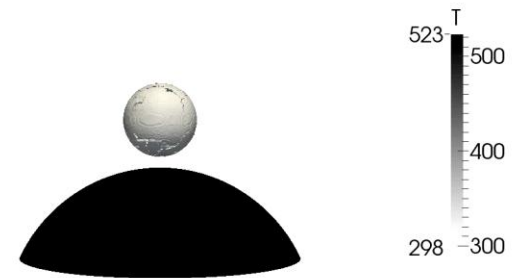
- *CFD is the analysis of systems involving fluid flow, heat transfer, mass transfer and associated phenomena such as chemical reactions by means of computer-based simulation.*
- *It is prediction tool based on the solution of governing equations of mass, momentum and energy conservation.*
- *Few examples:*



Bubbles in a
Fluidized beds



Particle migrations
through patients' airways
during breathing cycle



Interaction of liquid
droplets on spherical
hot surface