Universiti Malaysia PAHANG
Engineering • Technology • Creativity

Computational Fluid Dynamics

Lecture One

by
Dr. A. Nurye
Faculty of Mechanical Engineering
nurye@ump.edu.my

Introduction to CFD

- Aims
  - The aim of this chapter is to introduce students the need to use CFD to solve engineering problems involving flow and heat transfer.

- Expected Outcomes: At the end of this chapter, students should be able to understand
  - definition and history of CFD
  - why we use CFD instead of analytical or experimental
  - where do we apply CFD

- References

1) J. Tu, G.H. Yeoh, C. Liu, Computational Fluid Dynamics : A Practical Approach, Elsevier, 1st Edition, 2013.
2) C.T. Shaw, Using Computational Fluid Dynamics, Prentice Hall, 1992

Definition and history of CFD

- CFD can be defined as determination of flow characteristics with coupled effects of fluid flow, heat and mass transfer, combustion, solidification, etc.
- In CFD, the abovementioned processes are converted to mathematical equations and solved using numerical process.
- CFD can also be defined as use of computer codes to solve a wide range of problems *in fluid* flow and *heat transfer* [1].

Definition and history of CFD

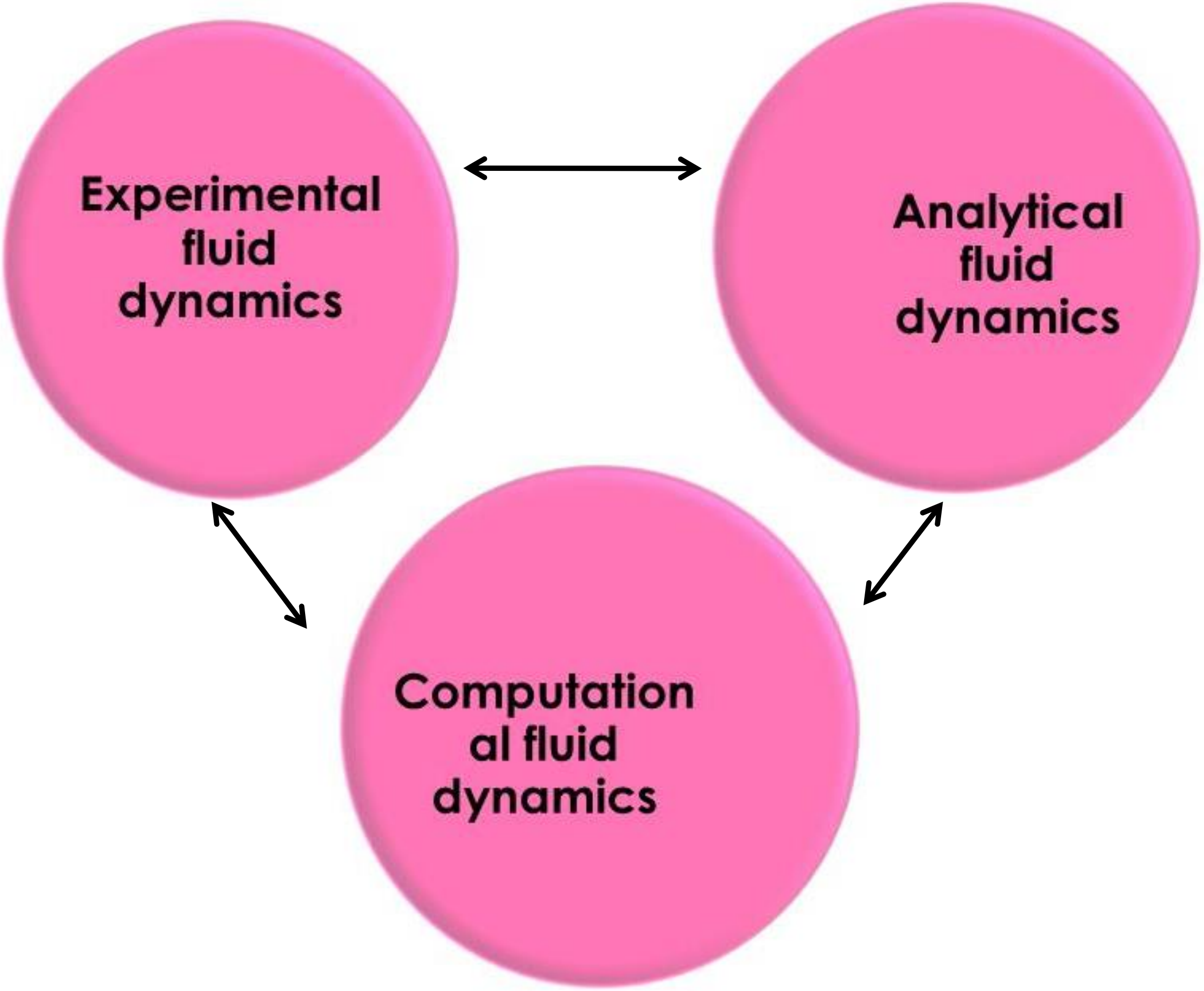
In **CFD,** the flow behavior associated with other processes are predicted both qualitatively and quantitatively. Some of the processes often associate with flow in CFD include:

- **Heat** and **mass** transfer,
- **phase change** such as solidification and boiling,
- chemical **reaction** in engines such as combustion,
- Coupling of solid and fluid stresses (eg. in flow induced fiber orientation)

Definition and history of CFD

- In CFD, fluids that are in motion are simulated and analyzed. In most cases, the influence of the flowing fluid behavior on other processes need to be taken into consideration.
- in CFD, the motion of the fluid is represented in a form differential equations, which govern a process of interest and are often called governing equations.
- The governing equations are converted to systems of algebraic equations which can easily be converted into computer programs.

TO SOLVE ANY FLUID DYNAMICS AND HEAT TRANSFER PROBLEM, THE FOLLOWING THREE BASIC APPROACHES BAN BE EMPLOYED [1]



HISTORY OF CFD

- Since 1940s analytical solution to most fluid dynamics problems was available for idealized solutions. Methods for solution of ODEs or PDEs were conceived only on paper due to absence of personal computer.
- Daimler Chrysler was the first company to use CFD in Automotive sector.
- Speedo was the first swimwear company to use CFD.
- There are number of companies and software's in CFD field in the world. Some software's by American companies are FLUENT, TIDAL, C-MOLD, GASP, FLOTRAN, SPLASH, Tetrex, ViGPLOT, VGRID, etc.

WHY WE USE CFD?

- Analysis and Design

  1. Simulation-based design instead of “build & test”
     - ✓ More cost effective and more rapid than EFD
     - ✓ CFD provides high-fidelity database for diagnosing flow field
  2. Simulation of physical fluid phenomena that are difficult for experiments
     - ✓ Full scale simulations (e.g., ships and airplanes)
     - ✓ Environmental effects (wind, weather, etc.)
     - ✓ Hazards (e.g., explosions, radiation, pollution)
     - ✓ Physics (e.g., planetary boundary layer, stellar evolution)

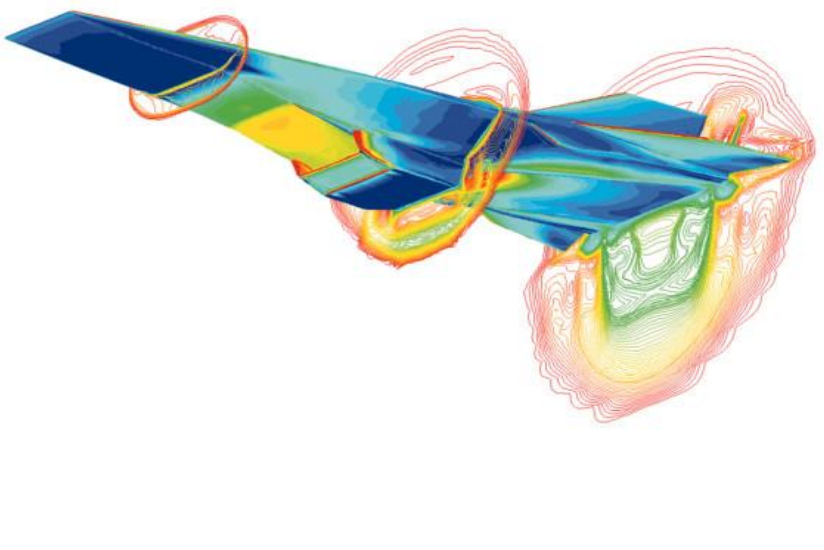
- Knowledge and exploration of flow physics

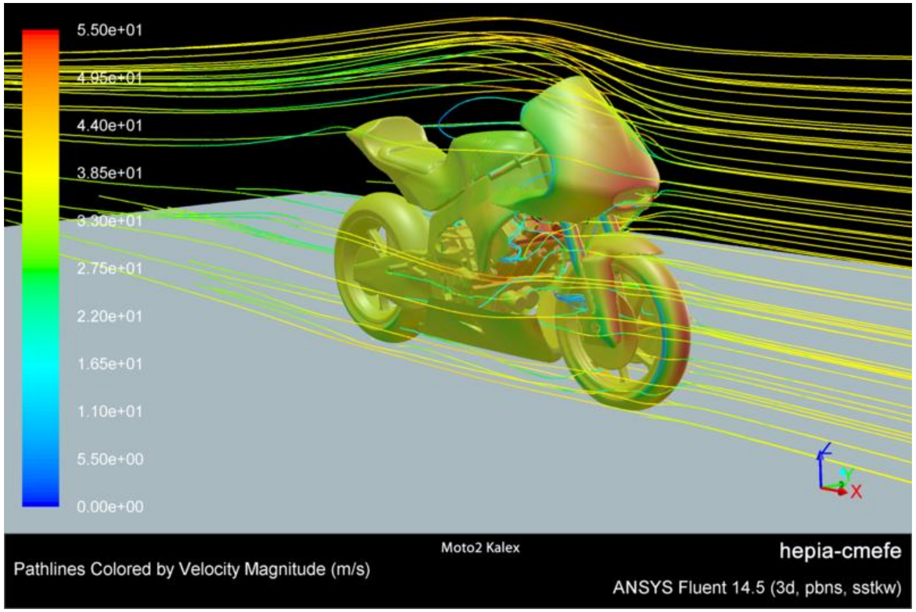
APPLICATION OF CFD (WHERE?)

CFD can be used in many applications involving flow and heat transfer. Some of the applications include:

- ***Engineering***
- ***Environment***
- ***Architecture and building science***
- ***Other phenomena or features***

APPLICATION OF CFD (WHERE?)



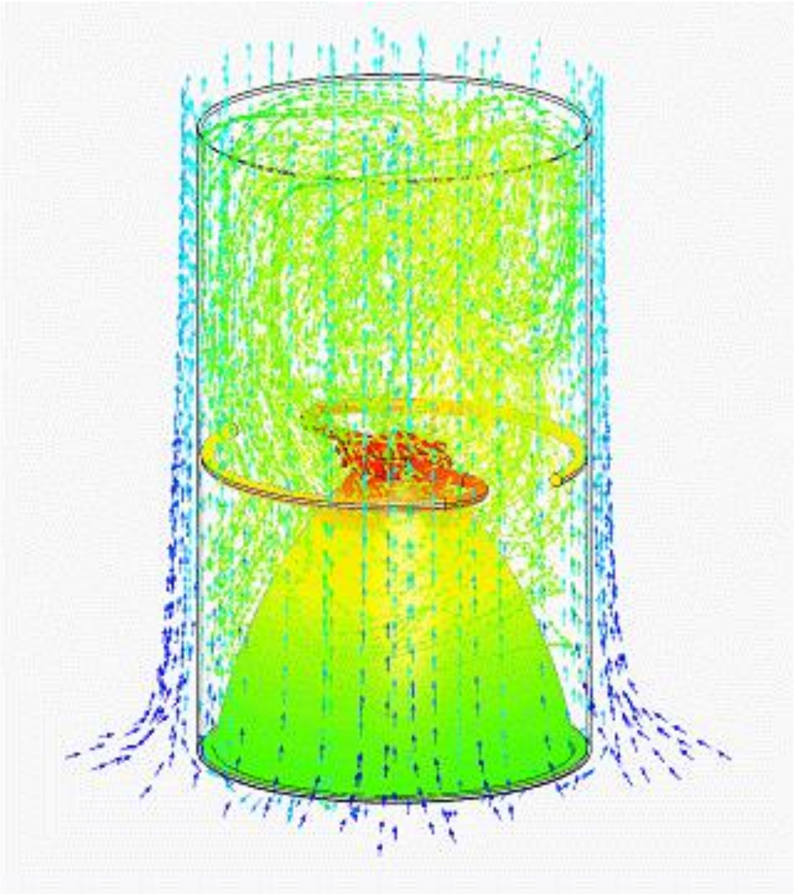


Computational fluid dynamic (CFD) image of the Hyper - X at the Mach 7 test condition with the engine operating.

http://www.dfrc.nasa.gov/Gallery/Photo/X-43A/HTML/ED97-43968-1.html

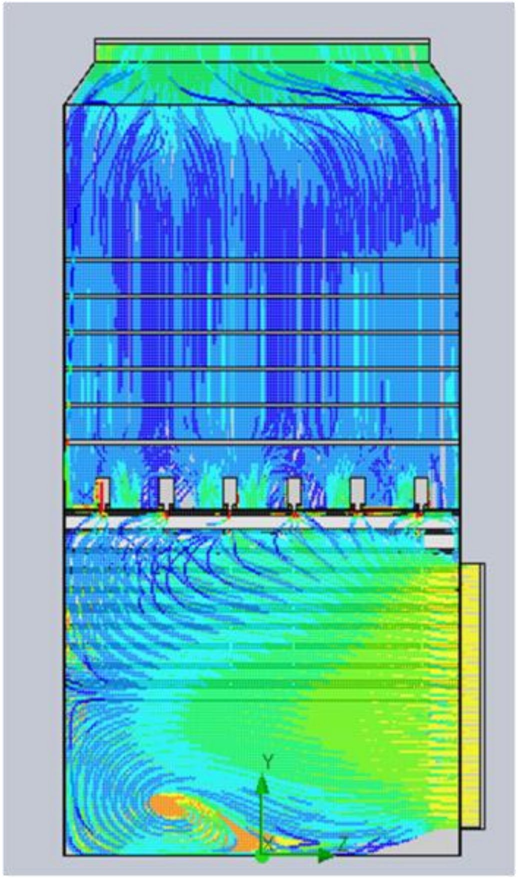
CFD simulation of the flow around a racing motorcycle Moto2 Kalex

https://commons.wikimedia.org/wiki/File:Hepia-cmefe_Kalex_CFD.png



LED downlight heat sink design thermal animation using CFD simulation

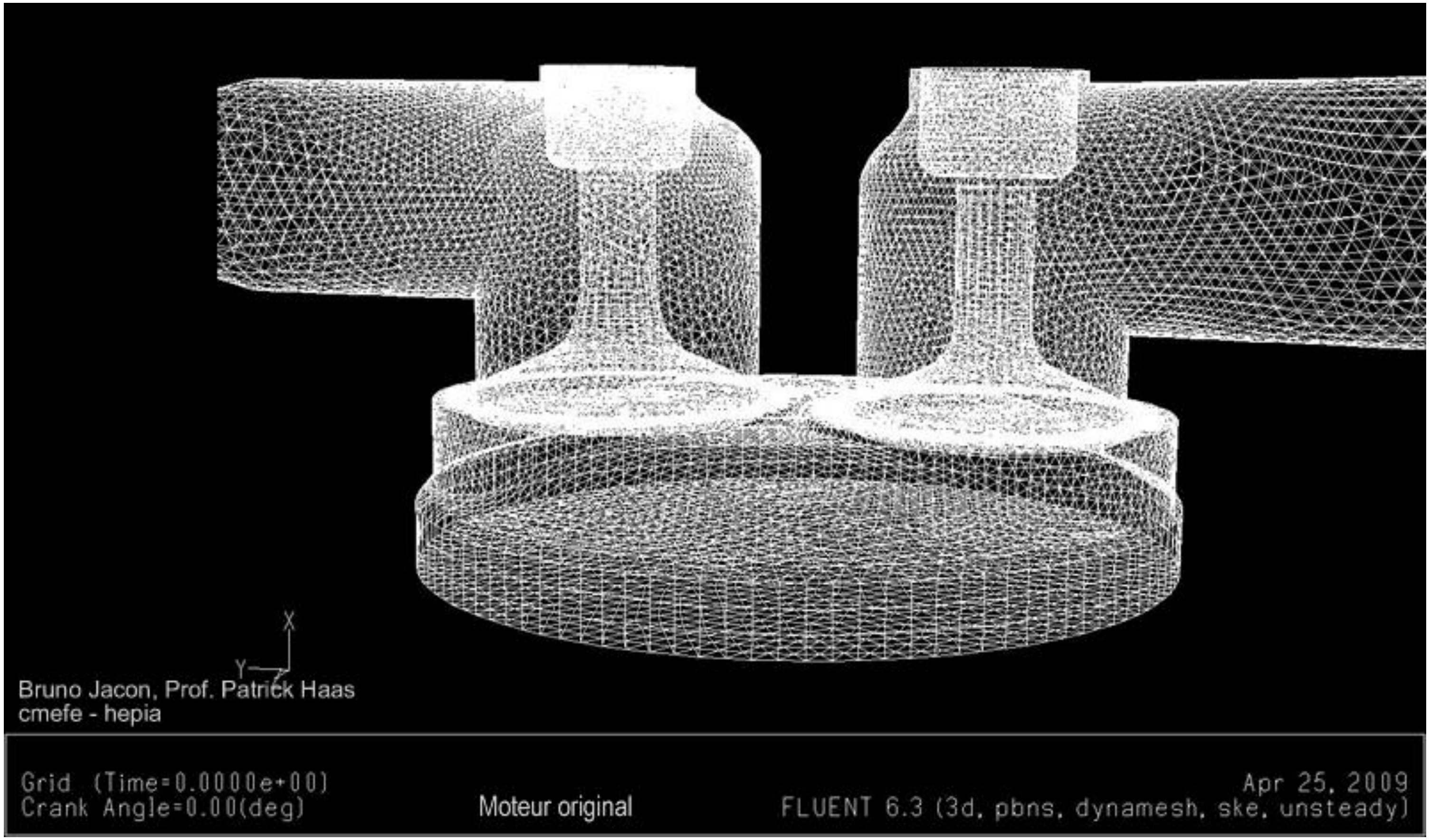
https://commons.wikimedia.org/wiki/File:CFD_LED_Free_Convection_Heat_Sink_Design.gif



A CFD model for airflow inside a FBC Adsorber tower. The air passes through a diffuser and six layers of perforated stainless steel trays.

https://commons.wikimedia.org/wiki/File:FBC-CFD.png

Universiti Malaysia PAHANG
Engineering • Technology • Creativity



CFD simulation of an internal combustion engine.

https://commons.wikimedia.org/wiki/File:Cmefe_CFD_moteur.gif

Dr. A. Nurye

Research interest:

- Computational Fluid Dynamics,
- Thermo-fluids,
- Multidisciplinary Numerical Modelling and Simulation

Contact:

Tel: +094246259
email: nurye@ump.edu.my