EURO^2

Introduction to Computational Fluid Dynamics

Taygun Recep Güngör, PhD
İstanbul Teknik Üniversitesi

ncc@ulakbim.gov.tr

What is CFD?

- There are basically three approaches or methods that can be used to solve a problem in **fluid mechanics** and **heat transfer**:

(1) Theoretical

(2) Experimental

(3) Computational

Governing Equations

Navier-Stokes equations (3D in Cartesian coordinates)

$$\rho\frac{\partial u}{\partial t}+\rho u\frac{\partial u}{\partial x}+\rho v\frac{\partial u}{\partial y}+\rho w\frac{\partial u}{\partial z}=-\frac{\partial \hat{p}}{\partial x}+\mu\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right]$$

$$\rho\frac{\partial v}{\partial t}+\rho u\frac{\partial v}{\partial x}+\rho v\frac{\partial v}{\partial y}+\rho w\frac{\partial v}{\partial z}=-\frac{\partial \hat{p}}{\partial y}+\mu\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right]$$

$$\rho\frac{\partial w}{\partial t}+\rho u\frac{\partial w}{\partial x}+\rho v\frac{\partial w}{\partial y}+\rho w\frac{\partial w}{\partial z}=-\frac{\partial \hat{p}}{\partial z}+\mu\left[\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right]$$

Local acceleration

Convection

Pressure gradient

Viscous terms

Continuity equation

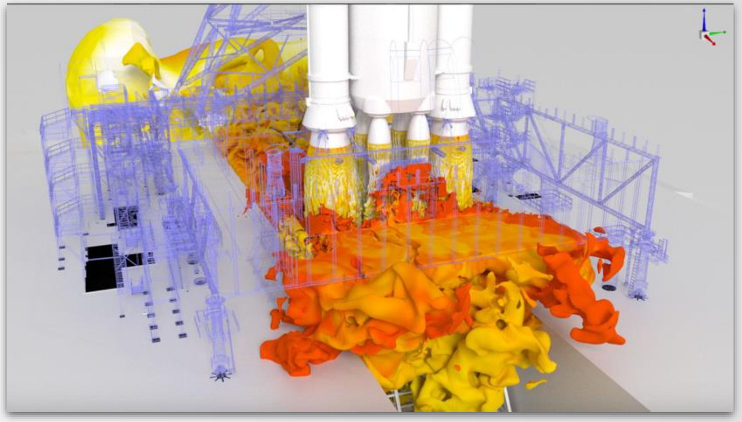
$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0$$

Equation of state

$$p=\rho RT$$

What is CFD?

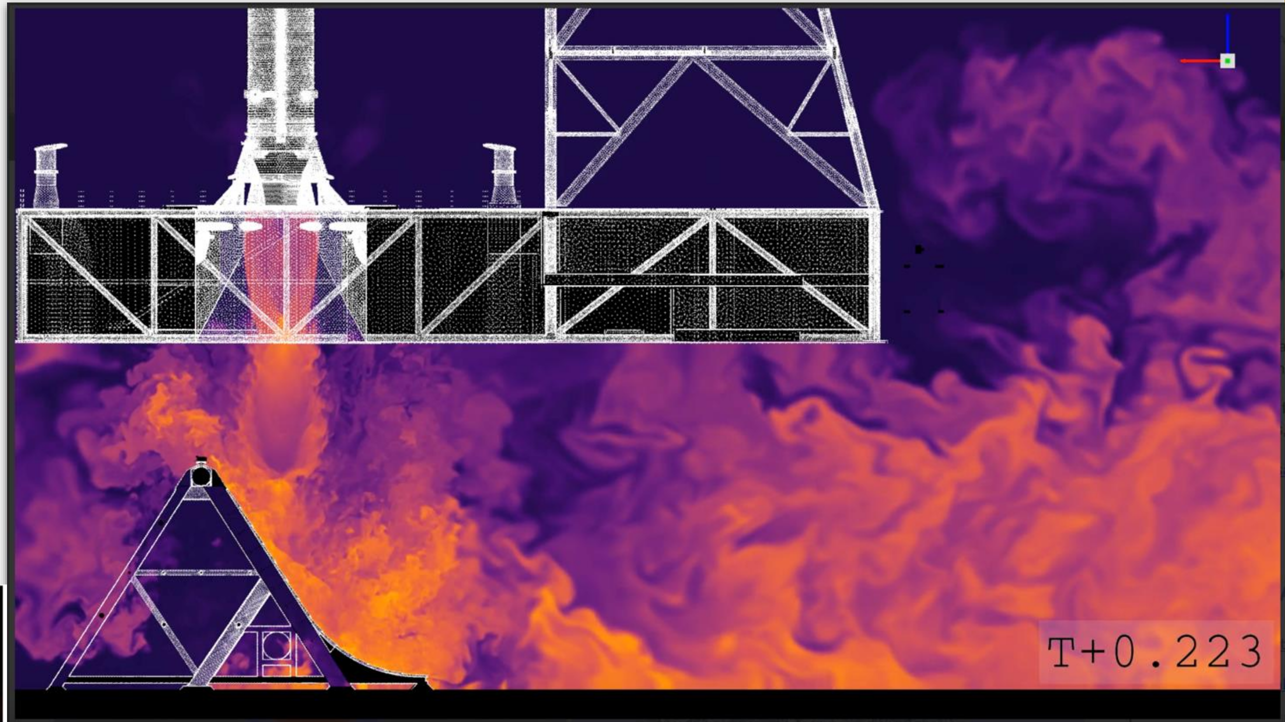
- **Computational fluid dynamics** (CFD) is the science of predicting
  - fluid flow,
  - heat transfer,
  - mass transfer,
  - chemical reactions,
  - and related phenomena
  - by solving the **mathematical equations** which govern these processes using a **numerical process**



https://www.nas.nasa.gov/pubs/gallery.html

Applications of CFD

Photo: Blue Origin

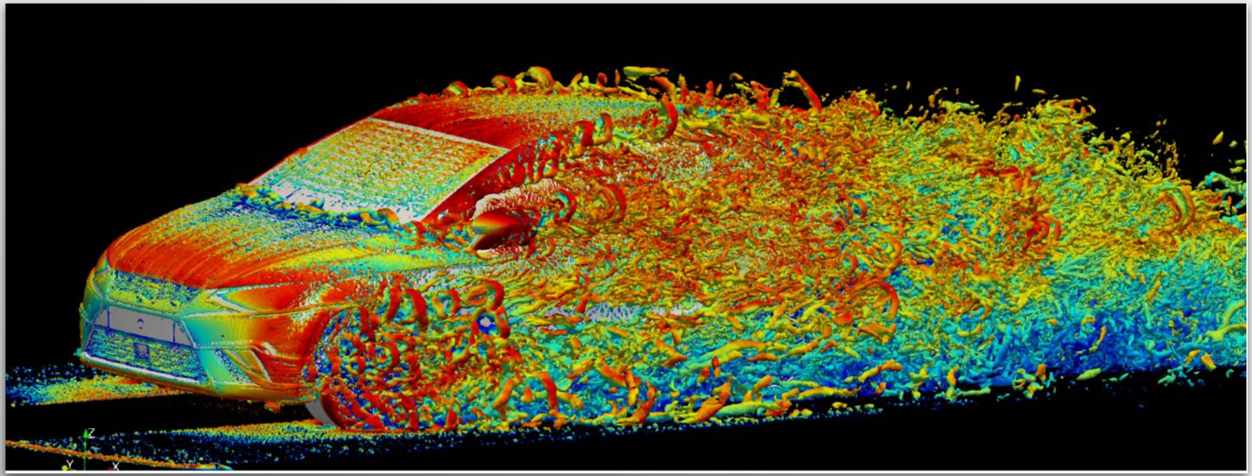


Simulation of launch ignition for NASA's next-generation Space Launch System at NASA Kennedy Space Center's Launch Complex 39B, generated using the NAS-developed Launch Ascent and Vehicle Aerodynamics flow solver. Particle colors indicate temperature of exhaust, where white is hotter and black is cooler. The image plane slices through the centerline of one of the two solid rocket boosters. *Michael Barad, Tim Sandstrom, NASA/Ames*

https://www.nas.nasa.gov/pubs/gallery.html

Applications of CFD

Aerodynamics of ground vehicles



https://www.bsc.es/discover-bsc/organisation/research-departments/large-scale-computational-fluid-dynamics

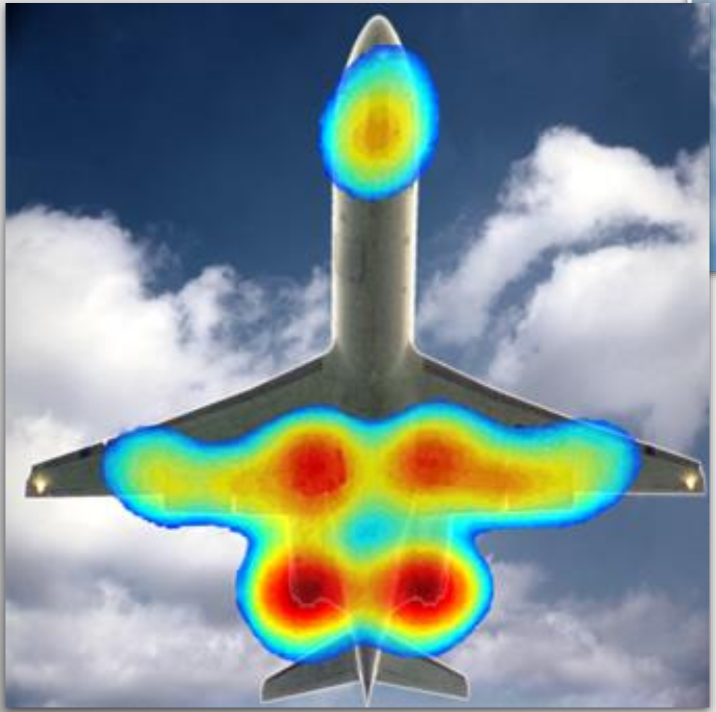
https://www.youtube.com/watch?v=TXMPE5mtXcw

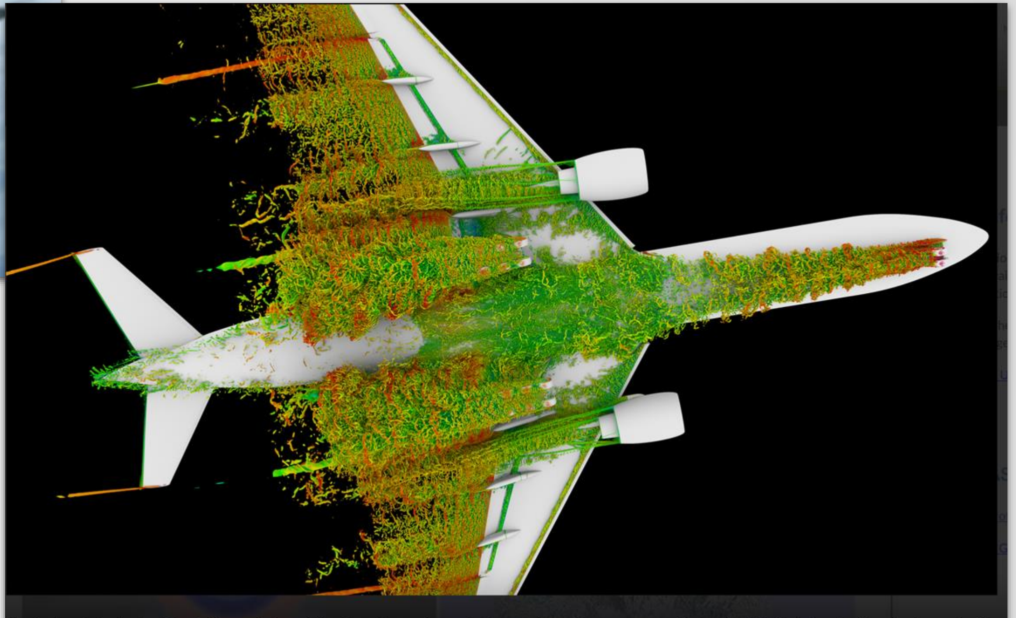
https://www.nas.nasa.gov/pubs/gallery.html

Applications of CFD

EURO²

Noise Prediction





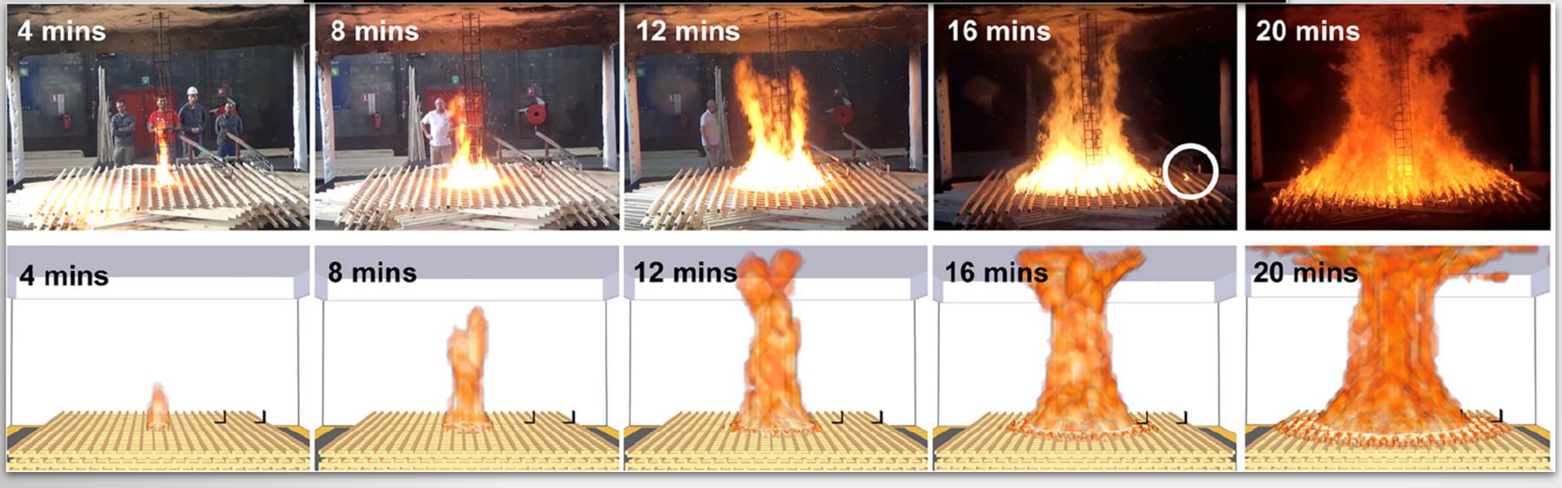
Visualization of the simulated flow field around a large airliner. Note the complex, vortical, unsteady flow features generated by the wing high-lift devices (leading edge slats and trailing edge flaps) and aircraft landing gear. *Benedikt Koenig, Dassault Systemes; Patrick Moran, NASA/Ames*

https://www.nas.nasa.gov/pubs/gallery.html

Applications of CFD

Forest Fires

The Glass Fire burns a hillside above Silverado Trail in St. Helena, Calif., on Sunday. (AP Photo/Noah Berger)(AP/Ethan Swope)



[An engineering CFD model for fire spread on wood cribs for travelling fires]

Dai et al, Advances in Engineering Software, Vol 173, 2022

Why do we use CFD?

Relatively low cost

Using physical experiments and tests to get essential engineering data for design can be expensive.

CFD simulations are relatively inexpensive, and costs are likely to decrease as computers become more powerful.

Speed

CFD simulations can be executed in a short period of time.

Quick turnaround means engineering data can be introduced early in the design process.

Ability to simulate real conditions

Many flow and heat transfer processes can not be (easily) tested, e.g. hypersonic flow.

CFD provides the ability to theoretically simulate any physical condition.

Why do we use CFD?

Ability to simulate ideal conditions

CFD allows great control over the physical process, and provides the ability to isolate specific phenomena for study.

Example: a heat transfer process can be idealized with adiabatic, constant heat flux, or constant temperature boundaries.

Comprehensive information

Experiments only permit data to be extracted at a limited number of locations in the system (e.g. pressure and temperature probes, heat flux gauges, LDV, etc.).

CFD allows the analyst to examine a large number of locations in the region of interest, and yields a comprehensive set of flow parameters for examination.

Limitations of CFD

Physical models

CFD solutions rely upon physical models of real world processes (e.g. turbulence, compressibility, chemistry, multiphase flow, etc.).

The CFD solutions can only be as accurate as the physical models on which they are based.

Numerical errors

Solving equations on a computer invariably introduces numerical errors.

Round-off error: due to finite word size available on the computer. Round-off errors will always exist (though they can be small in most cases).

Truncation error: due to approximations in the numerical models. Truncation errors will go to zero as the grid is refined. Mesh refinement is one way to deal with truncation error.

Boundary conditions

As with physical models, the accuracy of the CFD solution is only as good as the initial/boundary conditions provided to the numerical model.

Numerical Methods

Many CFD techniques exist

- Finite volume
- Finite difference
- Finite element
- Spectral methods
- Boundary element
- Vorticity based methods
- Lattice gas/lattice Boltzmann
- And more!

- The most common in commercially available CFD programs are:
  - Finite volume method - OpenFOAM
  - Finite element

What kind of flow physics?

Based on the physics of the fluids phenomena, CFD can be distinguished into different categories using different criteria

- Viscous vs. inviscid (Reynolds number)
- External flow or internal flow (wall bounded or not)
- Turbulent vs. laminar (Reynolds number)
- Incompressible vs. compressible (Mach number)
- Single- vs. multi-phase (Capillary number)
- Thermal/density effects (Prandlt number, g, Eckert number)
- Free-surface flow (Froude number) and surface tension (Weber number)
- Chemical reactions and combustion (Peclet number, Damköhler number)
- etc…

Thanks

This project has received funding from the European High-Performance Computing Joint Undertaking (JU) under grant agreement No 101101903. The JU receives support from the Digital Europe Programme and Germany, Bulgaria, Austria, Croatia, Cyprus, Czech Republic, Denmark, Estonia, Finland, Greece, Hungary, Ireland, Italy, Lithuania, Latvia, Poland, Portugal, Romania, Slovenia, Spain, Sweden, France, Netherlands, Belgium, Luxembourg, Slovakia, Norway, Türkiye, Republic of North Macedonia, Iceland, Montenegro, Serbia