CFD for Aerospace Applications

Introduction

Dr. Ziad Boutanios

Concordia University - MIAE

January 6, 2022

Why CFD?

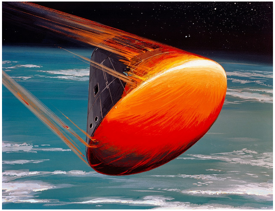
Usual engineering methods

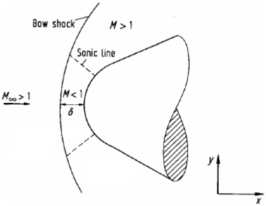
- Ever since the beginnings of engineering two methods were mainly used in design and analysis, usually alternating,
  1. *Theory*
     - Inexpensive and generally successful but limited to simple and ideal cases that rarely occur in real life.
     - Can be good for basic initial configurations though.
  2. *Experiment*
     - Build it and modify it by trial and error until you (hopefully) find the correct configuration.
     - This is error prone and can be pretty expensive, especially for new and original design.

Why CFD?

Untractable cases

- Some cases cannot be tackled with theory or experiment,
  - Atmospheric entry vehicles at $\sim$ 10 km/s (hypersonic) and temperatures of $\sim$ 11000 K.
  - Shock detachment distance δ and bow heating cannot be predicted theoretically (several co-existing flow regimes) and it is impossible to simulate experimentally.
  - Analyzed with CFD in the 60s using simple finite difference algorithms.





Why CFD?

Untractable cases

- Another untractable case,
  - The NASA Highly Maneuverable Aircraft Technology (HiMAT) aircraft
  - Preliminary wind tunnel tests predicted unacceptably high drag at transonic speeds, prohibitive design iteration cost and delays to the project.
  - CFD reduced cost by 8x and met deadlines and design objectives.

What is CFD?

- The physics of fluid flow are governed by the conservation equations of mass (continuity equation), momentum and energy (collectively known as the Navier-Stokes equations) and generally expressed in partial differential form (PDE). *You have seen that in Fluids II*.
- CFD is the practice (art?) of replacing the governing PDE's of fluid flow with numbers, and advancing these numbers in space and/or time, giving us a numerical description of the flow field.

What is CFD?

- This is done by dividing the spatial domain of interest into several cells (a tessellation, or grid, or mesh), and solving for the numbers representing flow variables (pressure, velocity, etc.) in each cell.
- Instead of considering a *continuous* space (like analytical models do) we restrict the problem to a *discrete* number of cells.

What is CFD?

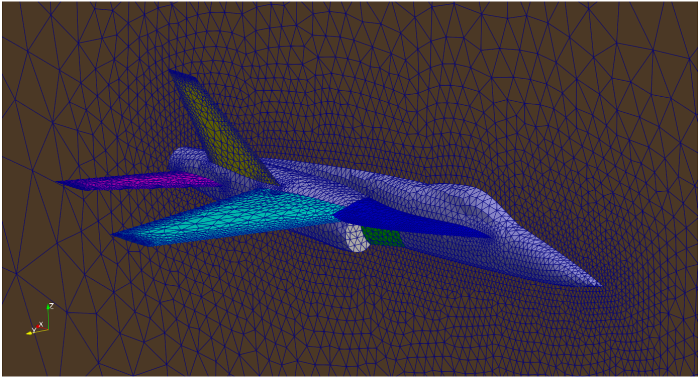


Figure 1: Example of a 3D unstructured mesh on an F-18 fighter jet geometry (tetrahedral cells).

What is CFD?

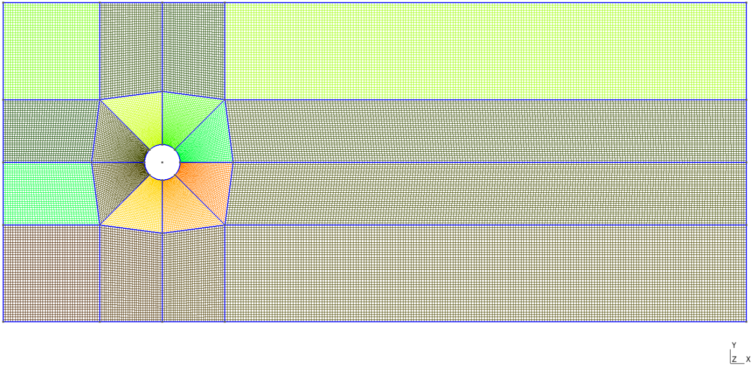


Figure 2: Example of a 2D structured mesh of a cylinder in a rectangular channel (hexahedral cells).

What is CFD?

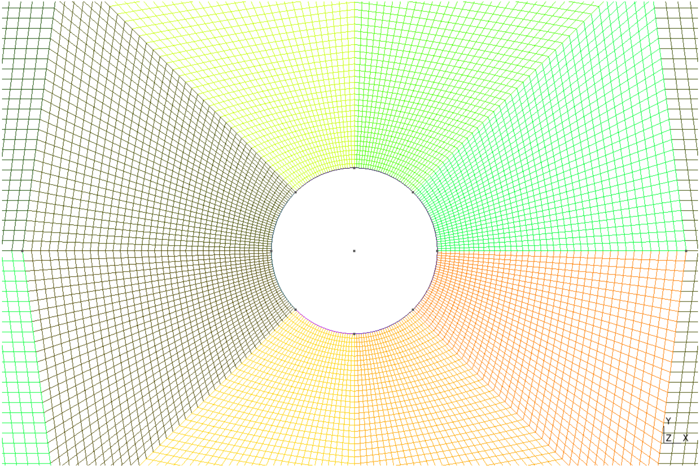


Figure 3: Closeup of a 2D structured mesh of a cylinder in a rectangular channel (hexahedral cells).

What is CFD?

Animation of pressure field of turbulent flow around a cylinder

What is CFD?

Animation of vorticity field of turbulent flow around a cylinder

What is CFD?

Animation of velocity field of turbulent flow around a cylinder

What made CFD possible?

- Even the simplest CFD solutions will require processing thousands of numbers, with billions of numbers possible today for the most demanding cases.
- This is made possible by the use of high-speed digital computers, and advances in computer hardware have been intimately linked to advances in CFD in the past 50 years.
- The strongest driving force for development of supercomputers has come in fact from the CFD community as recognized since 1982 [1].

The role of CFD in modern fluid dynamics

- The 1st generation of CFD solutions appeared in the 1950s spurred by the advent of efficient high-speed computers and the need to solve high velocity and temperature re-entry vehicle problems (high temperature gas dynamics).
- High temperatures requiring vibrational energies, equilibrium and non-equilibrium chemical reactions made analytical solutions impossible, even for the simplest flow configurations.
- These limitations were removed by CFD, which led to successful design.

The role of CFD in modern fluid dynamics

- The 2nd generation of CFD solutions consisted of applying the full Navier-Stokes equations to problems so complicated that boundary layer analytical approximations were not possible.
- These problems involved 2D Mixed subsonic-supersonic flow, separating viscous boundary layers with recirculation, etc.
- Again, analytical limitations and prohibitive experimental cost and delays were overcome by using CFD.

The role of CFD in modern fluid dynamics

- Nowadays CFD has become an essential part of fluid dynamics and a legitimate third approach, complementing and often superceding the initial theory and experiment.
- This was made possible by the steady development of new computers with faster run times and the appearance of massively parallel supercomputers.
- Today a \$2K PC is 10x faster than a \$1M server with 20 processors from 1995!

The role of CFD in modern fluid dynamics

- Other factors promoting CFD adoption are the decreasing cost of computations and increasing cost of wind tunnel testing.
- A typical design workflow in the aerospace industry today would consist of several CFD design iterations, with wind tunnel or experimental testing for final validation.
- This is especially true for gas turbine engines where it is very difficult, if not impossible, to install and operate sensors inside the engine.

The Objective of this course

- The objective of this course is to provide an introduction to CFD practice in the aerospace industry.
- The course is aimed at students with no prior CFD experience in order to prepare them for professional practice and/or future specialization.

Curriculum

- We will cover the following,
  - The Navier-Stokes equations and their different forms in CFD for incompressible and compressible flow, from subsonic to supersonic regimes.
  - Direct and iterative solvers.
  - Finite-differencing of operators and order of accuracy.
  - An introduction to the finite volume method.
  - Popular solution techniques.

Curriculum

- We will also cover,
  - Meshing techniques (structured and unstructured).
  - Example workflows from industry using both opensource software (OpenFOAM) and commercial software (ANSYS Fluent or Star-CCM+).
- We will apply what we learned in group projects.

References

[1] R.A. Graves. "Computational Fluid Dynamics: The Coming Revolution". In: *Astronautics and Aeronautics* 20.3 (1982), pp. 20–28.