Modern Aircraft Design and CFD

Hrvoje Jasak

hrvoje.jasak@fsb.hr

FSB, University of Zagreb, Croatia

What is CFD?

Computational Fluid Dynamics

- Definition of CFD: Versteeg and Malalasekera: “An Introduction to Computational Fluid Dynamics”

  > “Computational Fluid Dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer-based simulation.”

- CFD is also a subset of Computational Continuum Mechanics, which also includes
  - Numerical stress analysis
  - Electromagnetics
  - Weather prediction and global oceanic circulation models
  - Large scale systems: galactic dynamics and star formation
  - Complex heat and mass transfer systems
  - Fluid-structure interaction
- In all cases, the common factor is the **continuum representation**
- Equations describing the system are in all cases identical or very similar
- Thus, similar numerical solution practices can be applied

CFD in Aerospace Industry

Introduction

- Aerospace industry is the first and most prevalent in the use of numerical techniques, including Computational Fluid Dynamics (CFD)
- Early beginning of CFD in early 1960’s
- First successes came to prominence in the 1970’s
- Creation of the CFD-service industry started in the 1980’s
- The CFD industry expanded significantly in the 1990’s
- Wide acceptance of computer-based design for external aerodynamics design in a commercial aircraft in the 2000’s. Use expended beyond initial interest in external aerodynamics
- In most phases of the process, it was the aerospace industry driving the CFD development to answer to its needs

CFD in Aerospace Industry

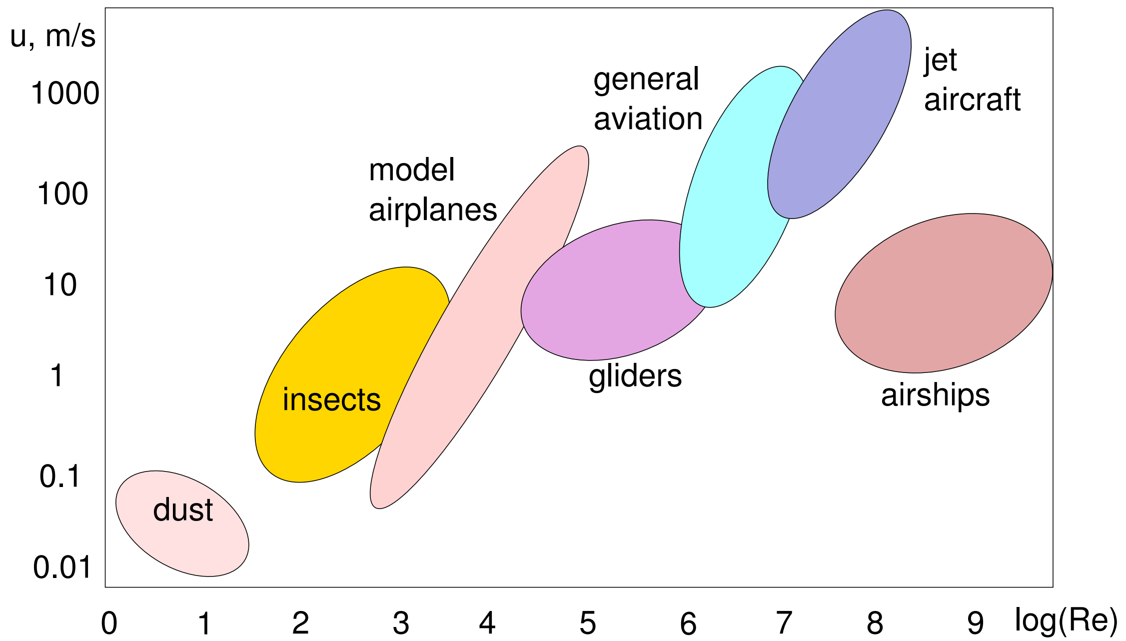
First fully computer-based design process for external aerodynamics design in a commercial aircraft: Airbus 380 in the 2000's

CFD in Aerospace Industry

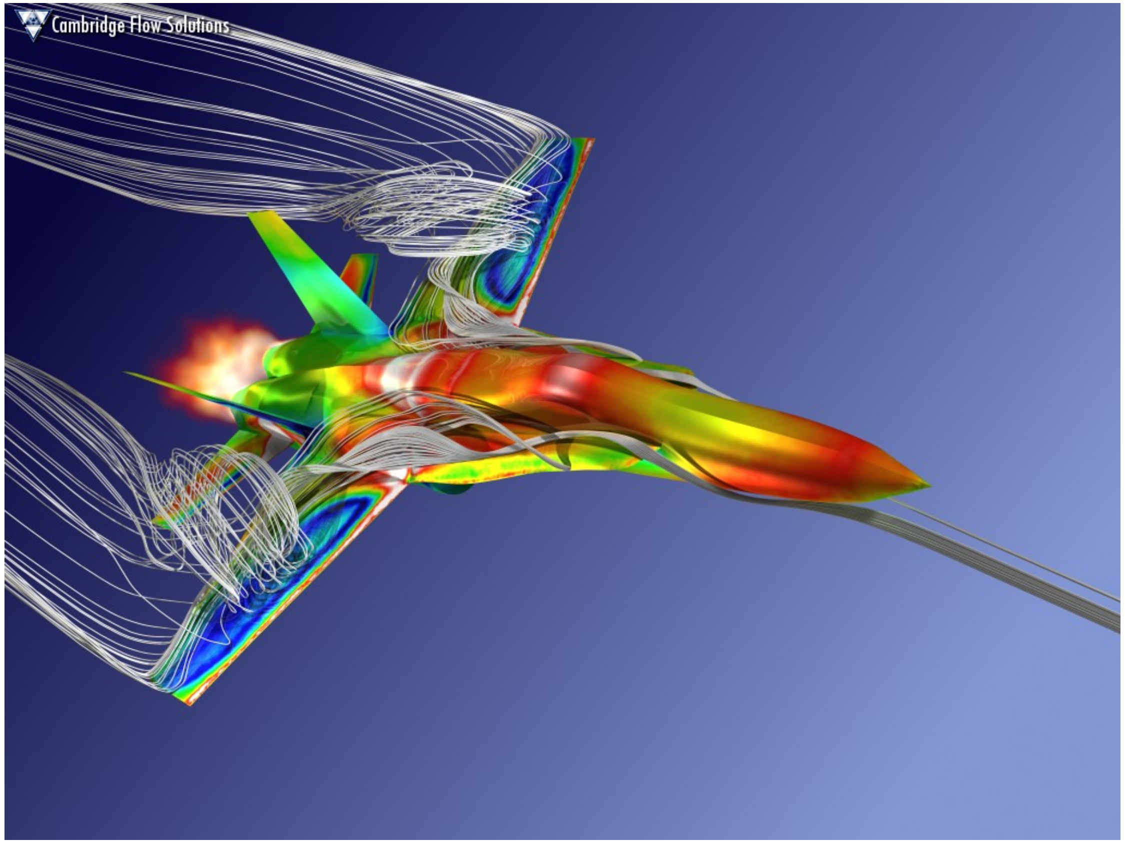
. . . but also, more interestingly

CFD in Aerospace Industry

Speed Range	Mach Number
low subsonic	< 0.3
high subsonic	$0.3 - 0.6$
transonic	$0.6 - 1.1$
supersonic	$1 - 5$
hypersonic	> 5



CFD in Aerospace Industry



- Use of CFD is no longer in question: definitely used throughout the design process
- Questions on fidelity and accuracy: can we get sufficiently reliable results?
- Roll-out of CFD continues with more complex requirements, increase of the computer power and applicability of new methods (optimisation)

Industries Using CFD

Fluid Flow Simulations

- Fluid flow is important over a range of simulations. I divide them info “smooth” and “rough” flow regimes
    - Smooth flows are well organised, always in closely controlled circumstances, where flow losses are critically important. Examples: aeroplane design, turbo-machinery components, car aerodynamics, etc.
    - In rough flows, flow organisation is much more important. Flow regime are uncertain or vary so much that we cannot talk about “design conditions”. Examples: electronics cooling, passenger compartment air management, flow between buildings, marine engineering

Industries Using CFD

Aerospace/Defence	Food and Beverage	Oil and Gas
Home Appliances	Polymer processing	Automotive
Power Generation	Heating and Ventilation	Pumps
Chemical Processing	Marine and Off-Shore	Semiconductor
Sports Equipment	Electronic Cooling	Turbo-machinery
Nuclear Power	Internal Combustion Engines	Environmental Flows

Capabilities of CFD Solvers

General Modelling Specifications: Fluent, Version 6.2

- 2D planar, 2D axisymmetric, 2D axisymmetric with swirl, and 3D flows
- Unstructured mesh (triangle and quadrilateral elements for 2D; tetrahedral, hexahedral, prism and pyramid elements for 3D)
- Steady-state or transient flows
- All speed regimes (low subsonic, transonic, supersonic, and hypersonic flows)
- Inviscid, laminar, and turbulent flows
- Newtonian or non-Newtonian flows
- Full range of turbulence models including k-epsilon, k-omega, RSM, DES, and LES
- Heat transfer including forced, natural, and mixed convection; conjugate (solid/fluid) heat transfer; and radiation, including solar loading
- Chemical species mixing and reaction, including homogeneous and heterogeneous combustion models and surface deposition/reaction models
- Free surface and multiphase models, including heat transfer and reactions
- Lagrangian trajectory calculation for dispersed phase (particles/droplets/bubbles), including spray and wall film models

Capabilities of CFD Solvers

General Modelling Specifications: Fluent, Version 6.2, *cont'd.*

- Phase change model for melting/solidification applications, cavitation model and wet steam model
- Porous media with non-isotropic permeability, inertial resistance, solid heat conduction, and option to compute interstitial velocities
- Lumped parameter models for fans, radiators, and heat exchangers
- Dynamic mesh capability for modelling flow around moving objects
- Inertial (stationary) or non-inertial (rotating or accelerating) reference frames
- Multiple reference frame (MRF) and sliding mesh options
- Mixing-plane model for rotor-stator interactions
- Comprehensive suite of aeroacoustics modelling tools
- Volumetric sources of mass, momentum, heat, and chemical species
- Material property database
- Dynamic (two-way) coupling with GT-Power and WAVE
- Add-on modules for fuel cells, magnetohydrodynamics, and continuous fibre modelling
- Extensive customisation capability via user-defined functions

Numerical Simulation Software

In the last 10 years, CFD performance and use coming together

- Computers power is a cheap commodity. Massively parallel computers are commonplace today and can be easily handled in software
- In aerospace, understanding the physics is typically not a problem
- Numerical methods cleaned up of systemic errors and gross failures
- Sufficient experience in research departments
- Validation against “trusted” experimental data
- Understanding of simplifications and assumptions

In other industries, roll-out of numerical simulation tools limited by experience

Numerical Simulation Software

Phases of Integration of CFD in the Design Process

1. Research and development departments: validation and assessment of capabilities. Typically involves detailed study of old designs or production pieces and comparison with available measured data.
2. Pre-design: experimenting with early prototypes and new ideas away from the current development line
3. Design and pre-production: new product development.
4. Production: optimisation of existing components and incremental development of the running design

Flow Analysis Goals

- No longer sufficient to make a plane fly
- Main objectives: economy and fuel consumption
- Government regulations: noise and pollution levels. Example: noise pollution caused by the supersonic shock wave on the ground killed supersonic flight! Simulation objective: dissipate the shock between the plane and the ground
- Passenger comfort. Includes both oscillatory and non-oscillatory flows around the aircraft, as well as cabin heating and air-conditioning
- Military applications: agile manoeuvring system and unstable aerodynamics

Phases of a CFD Simulation

Describing Geometry and Mesh Generation

- Define the flow geometry and create the computational mesh

Pre-Processing

- Choice of mathematical model: what kind of fluid, what equations are of interest: compressible or incompressible, steady-state or transient, energy equation, turbulence, buoyancy effects, chemical reactions etc.
- Initial and boundary condition setup

Flow Solution

- Setting up solution and solver parameters: discretisation schemes, relaxation parameters, choice of models, linear equation solver etc.
- Running the solver

Post-Processing and Data Analysis

- Extracting global flow parameters, *e.g.* lift and drag, pressure drop
- Flow visualisation: vectors, contours, iso-surfaces, streamlines or stream ribbons. Used to help understand the flow behaviour
- Using simulation results in the design process
  - Geometry or flow condition variations; parametric studies or sensitivity analysis; error estimation and mesh refinement studies

CFD Studies

Objectives of CFD Simulations in the Design Phase

- **Integral studies.** In simple lift and drag studies, we could be looking at a small number of integral properties.
- **Flow organisation**, where global characteristics of the flow are controlled to achieve stability or a desired pattern
- **Management of detailed flow structure.** Example: remove the vortex depositing dirt on a part of the windshield
- **Sensitivity and robust design studies.** Usually cannot be seen in results without experience or require specialised simulations.

CFD Software Development

Who Writes Numerical Simulation Software?

- Small experimental codes: playing around with physics and numerical methods
- In-house “general” CFD solver development
- In-house custom-written software for specific purposes: *e.g.* wing-nacelle engine system, turbine blade optimisation, simulation of unstable manoeuvres in military jets, calculation of directional derivatives and solution stability, matching computations with measured data sets etc.
  - Complex and tuned panel method codes
  - Simplified physics, *e.g.* potential flow and boundary layer codes
  - Hooked-up mesh generation and parametrisation
  - Special purpose codes: sensitivity, aero-acoustics etc.
  - In-house development kept secret: competitive advantage
- Government-sponsored (National Labs) developments
- General-purpose CFD packages: from a fridge to a stealth plane
- University research codes; public-domain software
- “Write-your-own” CFD solver
- Software getting increasingly complex: you need a PhD to join the game

Current Challenges

Current CFD Work

- Mesh generation, especially parallel mesh generation
- Handling massively parallel simulations
- Integration into the CAD-based design process
- Fluid-structure interaction and aeroacoustics
- On the cusp between two generations of general-purpose CFD solvers: procedural programming, Fortran and C against object orientation
- The push for bigger, faster, more accurate simulations in external aerodynamics not so strong in the aerospace market: meshes are already sufficiently large. Also, extensive experience of the required size of the model, mesh resolution and locally fine meshes from the days when computer power was expensive
- In aircraft engine design, the opposite is the case. **ASC Project** (Advanced Simulation and Computing), US Dept of Energy, Los Alamos, Livermore, Sandia, Stanford University and other partners
  `http://www.stanford.edu/group/cits/research/index.html`
  `http://www.llnl.gov/PAO/news/asc/`

Current Challenges

ASC Project

- Tip-to-toe simulation of a turbo-fan aircraft engine, including fan, turbo compressor, combustion chambers and turbine. Preferred modelling technique: Large Eddy Simulation
- Integrated Multi-code Simulation Framework: software challenges
- As a part of the project, world's biggest parallel computers have been built:
  - ASC Red, Sandia 1996
  - ASC Blue Livermore, Los Alamos, 1998
  - ASC White, Livermore, 2001
  - ASC Q, Los Alamos, 2003
  - ASC Red Storm, Sandia, 2004, 40-TeraOps
  - ASC Purple, Livermore, 2005, 100-TeraOps
  - ASC Blue Gene Livermore, Los Alamos, : 130 000 CPUs at 360 TeraFlops
- For comparison, ASC Linux, 960 node-Linux box with 1920 processors and $3.8\,\mathrm{T}B$ produces peak performance of 9.2 TeraFlops/s
- The idea of doing a complete engine is somewhat abandoned: not enough power for LES on compressor or turbine. Using combined RANS/LES simulation approach with coupling on interfaces

Scope of Computations

Aerospace CFD Simulations

- **Aerodynamic Drag**
  - Drag varies with the velocity squared: major influence at aerospace speed. Narrow improvements in drag lead to considerable advances:

    A 15% drag reduction on the Airbus A340-300B would yield a 12% fuel saving, other parameters being constant.
    (Mertens, 1998)
- **High-Lift Aerodynamics**
  - High-lift wing configuration very important: lower take-off and landing speed, higher pay-load etc.
  - Study of multi-element airfoil configuration: high flow curvature, flow separation, wakes from upstream elements, laminar-to-turbulent boundary layer transition etc.

Scope of Computations

High-Lift Devices: Leading Edge Extension

Scope of Computations

Aerospace CFD Simulations

- **Unsteady Aerodynamics**
  - Oscillatory instability: dynamic stall on helicopter rotor blades in forward flight; vortex shedding behind bluff bodies
  - Non-oscillatory flows: flow separation at the high angle of attack. Turbulence effects are critical for accurate modelling
- **Rotary Aerodynamics**
  - Simulation of helicopter rotor blades usually considered a specialised area of research: special assumptions and modelling regimes
- **High-Speed Aerodynamics**
  - At high speed, the equation of state and ideal gas assumptions break down. In other aspects, the flow is becoming easier to handle. Generally refers to speed of $Ma = 5$ and above
  - For high speed, and due to the real gas effects we speak of *aerothermodynamics* rather than aerodynamics.

Scope of Computations

Aerospace CFD Simulations

- **Rudder and Steering Diagrams**
  - In automated steering/targeting systems, the aircraft/missile is controlled by a computer: given target or flight path
  - Automatic control systems rely on the diagrams showing the response on steering commands: in practice, large look-up tables or fitted functional data. Consider a case of a rotating missile with 2×4 control surfaces.
- **Internal Flows and Auxiliary Devices**
  - Internal flows: incompressible, low speed, aerodynamics forces typically of no consequence
  - A number of specialist devices associated with safe operation of aircraft. Example: air intakes for air conditioning systems operating at -50 C and at low pressure get blocked by snow-like particles

Scope of Computations

Aerospace CFD Simulations

- **Stability and Robust Design**
  - Stability analysis takes into account the effects of uncertainly (noise) in the input parameters. Example: how much will the lift coefficient on the airfoil change with a 5% change in the angle of attack?
    - Away from stall point: lift is stable to small change in conditions
    - At stall: catastrophic change
    - What about a NACA 0012 (symmetric airfoil profile) at zero angle of attack?
  - Stability of the solution on small perturbations of inlet conditions
- **Fluid-Structure Interaction.** Example: wing flutter
  - Aerodynamic forces from fluid flow determine the load on the wing. Wing itself is an elastic structure and deforms under load
  - Deflection of the elastic wing changes the flow geometry: a new solution produces different surface load
  - Interaction between the two may be stable or unstable: flutter

Finite Volume or Finite Element?

Spatial Discretisation

- Finite Difference Method (FVM): Not used commercially. Important in aero-acoustic simulations; problems with high-order boundary conditions
- Finite Volume Method: dominates the fluid simulation arena
- Finite Element Method. No particular reason why it cannot be used; however, the bulk of the numerical method development targeted to FVM I do not know any FEM fluid flow aerospace solvers, but FEM dominates the structural analysis arena
- Discontinuous Galerkin: a formal unification of the FEM and FVM ideas. Strongly conservative and consistent, but extensions are still impractical (control of matrix properties, solution techniques etc.) Consider it work-in-progress
- Monte Carlo Methods: extensively used in low-density high-speed aerodynamics (Space Shuttle re-entry). Techniques are specialised for high efficiency
- Spectral techniques: special purposes only. Extremely efficient and accurate for “box in a box” and cyclic matching simulations, *e.g.* DNS

Finite Volume or Finite Element?

Temporal Discretisation

- Steady state: no temporal discretisation required
- Time domain: bulk of transient flow simulations
- Frequency domain: special purposes. Example: in turbo-machinery simulations, it is possible to extract the dominant frequencies. Instead of solving a time-dependent problem, a series of steady simulations is set up, each for a selected frequency (effects of the temporal derivative now convert into a source/sink term). The time-dependent behaviour is recovered from the combination of frequency solutions.

Simplified Solvers

Simplified Solvers of Industrial Importance

- It is not always necessary to run a full Navier-Stokes solver to obtain usable results. Also, the simulation time is sometimes critical: approximate result now.
- **Panel method.** Combination of source, sinks, doublets and vortex elements used to assemble a "zero streamline" form which represents the body. Extremely fast and capable of producing indicative solutions with experience.
  `http://www.engapplets.vt.edu/fluids/vpm/`
- **Potential Flow Solvers.** Incompressible formulation considered too basic. However, the compressible potential formulation, or even a transient compressible potential can be very useful. The main effect missing in the simplified form is the viscosity effect in the boundary layer: effective change of shape for the potential region. Potential flow solver can be used to accelerate the solution to steady-state for more complex solver: initialisation of the solution
- **Potential Flow with Boundary Layer Correction.** Here, a combination of the compressible potential and boundary layer correction takes into account the near-wall effect: the geometry is corrected for displacement thickness in the boundary layer
- **Euler Flow Solver.** Neglects the viscous effects but the compressibility physics can be handled in full.