AA214: NUMERICAL METHODS FOR COMPRESSIBLE FLOWS

Computational Fluid Dynamics (CFD)

Outline

1 What is CFD?

2 Why CFD?

3 What for CFD?
- A Tool for Research
- A Tool for Design Analysis and Optimization

4 Anatomy of CFD

- Abbreviation of Computational Fluid Dynamics
- Branch of fluid mechanics that uses mathematical models, numerical methods, and (powerful) computers to solve and analyze problems involving fluid flows
- Corresponding software is typically *verified* using academic problems, then *validated* using experimental (wind tunnel, flight test, etc.) data

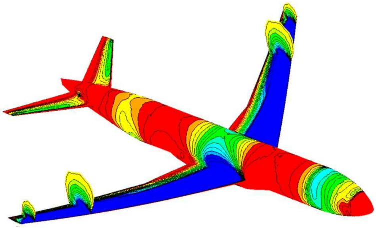


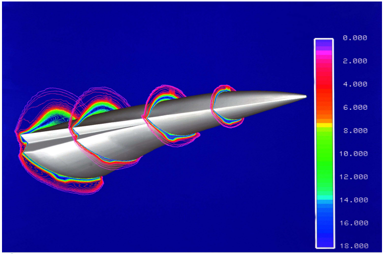
Figure: NASA Common Research Model (CRM): Pressure contour plots and iso-lines

- NASP (National AeroSpace Plane)

Figure: X-30 NASP: Early configuration (Rockwell, circa 1990)

- president Reagan announced the NASP project in his 1986 State of the Union message, calling for the development of "*... a new Orient Express that could, by the end of the next decade, take off from Dulles Airport and accelerate up to twenty-five times the speed of sound, attaining low earth orbit or flying to Tokyo within two hours...*"

- X-30 NASP (aka Copper Canyon Phase 2, HySTP (Hypersonic System Technology Program), Orient Express, X-30)



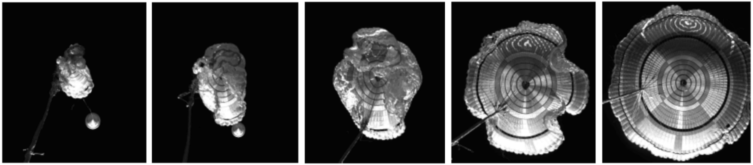
NASP–Computer Aided Design Computational Fluid Dynamics
NASA Langley Research Center 6/6/1988 Image # EL–1996–00191

  - SSTO (Single Stage To Orbit) winged orbital launch vehicle with air-breathing scramjet
  - horizontal takeoff/horizontal landing
  - **CFD for modeling the scramjet and designing the aerodynamic shape**
  - cancelled in 1990 due to cost and technical challenges (composite hydrogen tank, carbon-carbon aerodynamic surfaces, high temperature heat pipes as sharp leading edges)

- Why CFD for the X-30 NASP?
  - ground test facilities (wind tunnels) do not exist in all flight regimes covered by the hypersonic flight of such a vehicle
  - the prospect in the twenty-first century for wind tunnels that can simultaneously simulate the high Mach numbers and high flow field temperatures encountered by transatmospheric vehicles are not encouraging [1]
  - flight testing is particularly challenging in this case, because of the speeds at which such a vehicle operates: When anything unexpected occurs, the time to react is so short that the tested system must be destroyed

[1]See John D. Anderson Jr., Computational Fluid Dynamics, The Basics with Applications, Chapter 1, McGraw-Hill, Inc.

- (Supersonic) parachute inflation dynamics

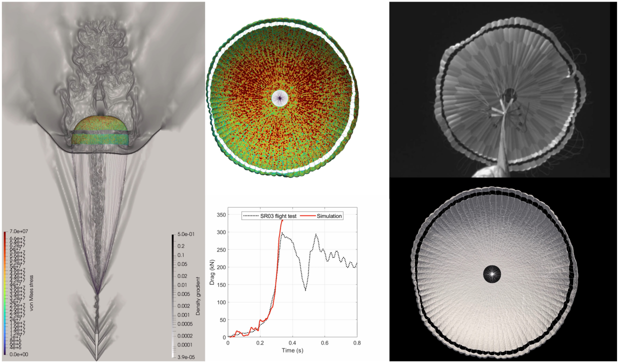


(a) 0.224 sec (c) 0.305 sec (e) 0.387 sec (g) 0.468 sec (i) 0.550 sec

Figure: Supersonic parachute inflation dynamics (O'Farell et al., AIAA 2016-3242)

  - parachute systems are crucial for the success and safety of space exploration, particularly during the descent and landing phases of space missions
  - unfortunately, they are also a top risk to every mission/platform that depends on them
  - parachute modeling is almost exclusively empirical: some predictions, such as loads, torque, and terminal rate of descent, use physics-based models anchored to test reconstructions; other aspects, such as packing and integration, and until very recently, deployment and inflation, are not modeled with enough confidence

- Supersonic parachute inflation dynamics



Flight test

Numerical simulation

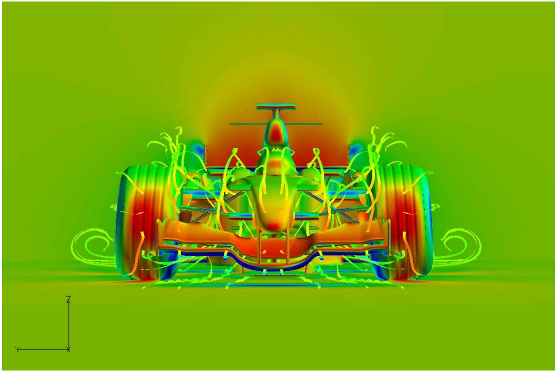
Figure: Numerical simulation of NASA's ASPIRE SR03 flight test using Stanford's AERO Suite for highly nonlinear fluid-structure interaction problems (circa 2023)

  - advanced CFD has only recently made a dent in the state of the art of the numerical modeling of parachute deployment and inflation

- Formula 1

  - the most widely watched annual sport event in the world, with viewing figures of 490 million per race
  - its commercial rights generate $1.827 billion in annual revenue
  - its top teams run on budgets of up to $400 million backed by global corporations such as BMW, Mercedes, Red Bull, and Renault
  - in 2022, 23 Grands Prix were held in Bahrain, Saudi Arabia, Australia, Italy (2), USA (2), Spain, Monaco, Azerbaijan, Canada, UK, Austria, France, Hungary, Belgium, Netherlands, Russia, Singapore, Japan, Mexico, Brazil, and UAE

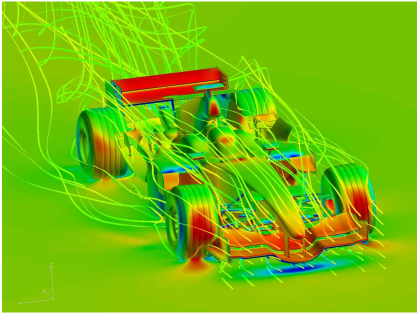
- Formula 1 (continue)
  - "*The first priority is aero, the second priority is aero, the third priority is aero, the fourth priority is tyres*" (Sergio Rinland: Williams, Brabham, Benetton, Sauber)



- Wind tunnels for Formula 1

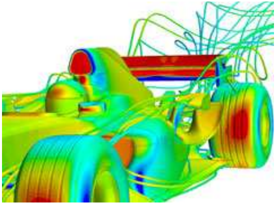
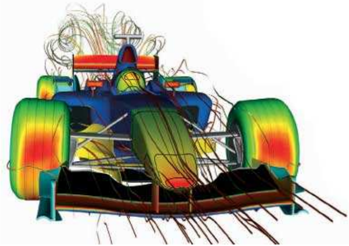
  - about $100 million investment
  - most are half-scale, but some are full-scale
  - moving carpets can be used to include the effect of the rotation of the wheels

- Why CFD for Formula 1?



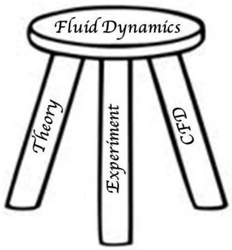
- to gain understanding and insight into the behavior of the car in configurations that are difficult or impractical in a wind tunnel
  - yaw (cross-wind, cornering)
  - steer with the front wheels turned
  - roll (ride-height variations)
- wind tunnel models are usually (too) stiff, while the front and rear wings are (very) flexible

- Why CFD for Formula 1? (continue)

  - In 2007, Renault announced an investment of $50 million in CFD because the work it would enable at the Formula 1 factory in Enstone, England, would find its way back to street car technology
  - "Success in F1 requires world-class CFD and a cutting-edge aerodynamic team that is armed with the best tools available ..." (Willy Rampf, Sauber Petronas)

- Modern viewpoint: Fluid dynamics has three pillars, namely, *theory*, *experimentation*, and *computation*
  - "Together with *theory* and *experimentation*, *computational science* now constitutes the 'third pillar' of scientific inquiry, enabling researchers to build and test models of complex phenomena" [2]
  - CFD is the third pillar of fluid dynamics



[2] United States President's Information Technology Advisory Committee, Computational Science: Ensuring America's Competitiveness, National Coordination Office for Information Technology Research & Development, 2005

- Counter-view point: Fluid dynamics has two pillars *which are thoroughly computational* [3]
  - *Experimental Fluid Dynamics*: Foundations were laid in France and England in the 17^{th} century
    - today, computation is an integral part of experimentation (for example, for analyzing measurements)
    - today, a CFD simulation is by itself a virtual fluid dynamics experiment
  - *Theoretical Fluid Dynamics*: Foundations were developed primarily in Europe during the 18^{th} and 19^{th} centuries
    - to be useful, a theory should be able to explain some existing observation and/or generate a meaningful prediction
    - in fluid dynamics a theory is typically mathematical in nature, and its application for prediction typically requires a CFD simulation

[3] Adaptation of Moshe Y. Varid's Editor's Letter "Science Has Only Two Legs", Communications of the ACM, Vol. 53, No. 9, September 2010

Outline

1 What is CFD?

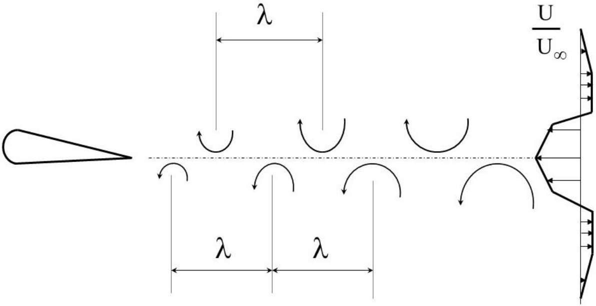
2 Why CFD?

3 What for CFD?
- A Tool for Research
- A Tool for Design Analysis and Optimization

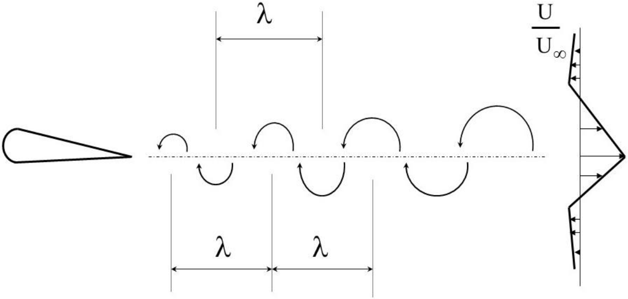
4 Anatomy of CFD

- "What if" numerical experiments
  - what if the flow was laminar?
  - what if the flow was turbulent?
- Numerical sensitivity analysis
  - what happens to the lift/drag ratio if the angle of attack is slightly increased?
  - what happens to the induced drag coefficient if the Reynolds number is slightly decreased?

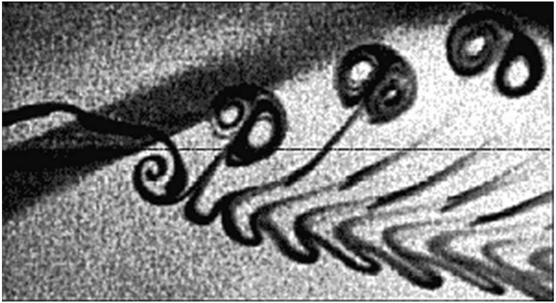
- Exploration of flow physics
  - the Knoller (1909, theory)-Betz (1912, theory), or Katzmayr (experimental verification, Vienna, 1922) effect: reduction in drag of an airfoil when the air stream is oscillating
    - low frequency plunging of an airfoil (momentum deficit, indicative of drag)



- Exploration of flow physics (continue)
  - the Knoller (1909, theory)-Betz (1912, theory), or Katzmayr (experimental verification, Vienna, 1922) effect: reduction in drag of an airfoil when the air stream is oscillating
    - higher frequency plunging of an airfoil (momentum surfeit, indicative of thrust)



- Exploration of flow physics (continue)
  - the Knoller-Betz (or Katzmayr) effect (continue)
    - increasingly higher frequency plunging of an airfoil

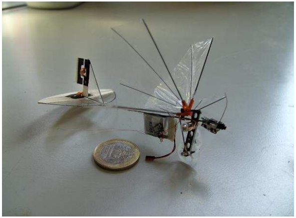


- Exploration of flow physics (continue)
  - the Knoller-Betz (or Katzmayr) effect (continue)
    - low frequency plunging of an airfoil: Positive drag

- Exploration of flow physics (continue)
    - the Knoller-Betz (or Katzmayr) effect (continue)
        - higher frequency plunging of an airfoil: Thrust production

- Exploration of flow physics (continue)
  - the Knoller-Betz (or Katzmayr) effect (continue)
    - increasingly higher frequency plunging of an airfoil: Thrust production with nonsymmetric wake patterns

- Exploration of flow physics (continue)
  - the Knoller-Betz (or Katzmayr) effect (continue)
    - implications for flapping wing propulsion
    - applications for Micro Air Vehicles (MAVs)



Outline

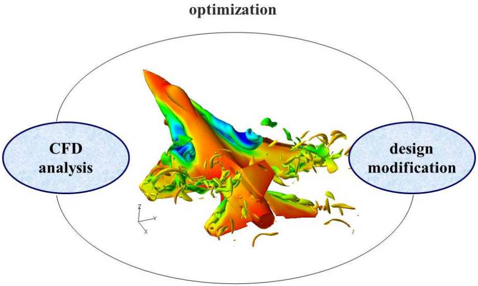
1 What is CFD?

2 Why CFD?

3 What for CFD?
- A Tool for Research
- A Tool for Design Analysis and Optimization

4 Anatomy of CFD

- Today: *Preliminary* and *detailed* designs
- Today: Design and design optimization by analysis



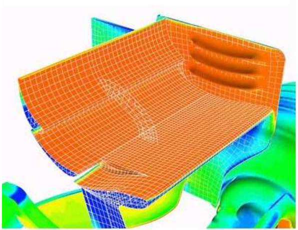
- iterative process
- resource intensive

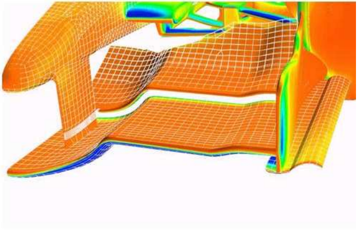
  $\Longrightarrow$ component design

  $\Longrightarrow$ detail design

  $\Longrightarrow$ model reduction (CME 345)

- Example: Formula 1 aerodynamic shape design
  - wind tunnels will probably remain the most important overall design tool for a relatively long time
  - however detail designs like wing mirrors and/or front/rear wings and end plates are currently designed using only CFD





- Sample impact: Formula 1 arrangement optimization
  - simulation of the turbulent flow past a complete car configured in its wind tunnel $\Longrightarrow$ move the rear-view mirrors to the wake of the tires

- Sample impact: Formula 1 arrangement optimization (continue)
  - position of the rear-view mirrors before performing the turbulent flow simulations

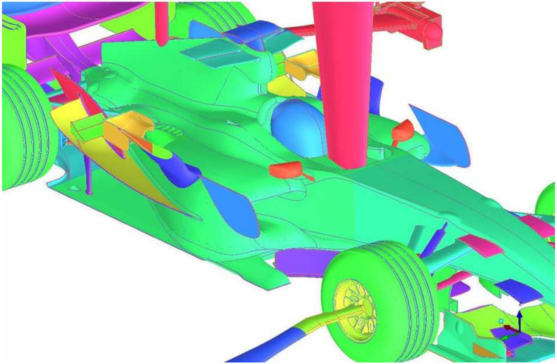


Figure: CAD (Computer-Aided Design) model (initial)

- Sample impact: Formula 1 arrangement optimization (continue)
  - "optimized" position of the rear-view mirrors

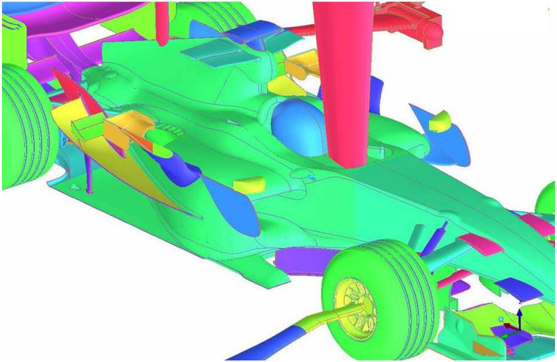


Figure: CAD model (modified)

- Sample impact: Formula 1 arrangement optimization (continue)
  - car before design optimization

Figure: Rear-view mirrors are on the car body in the vicinity of the cockpit

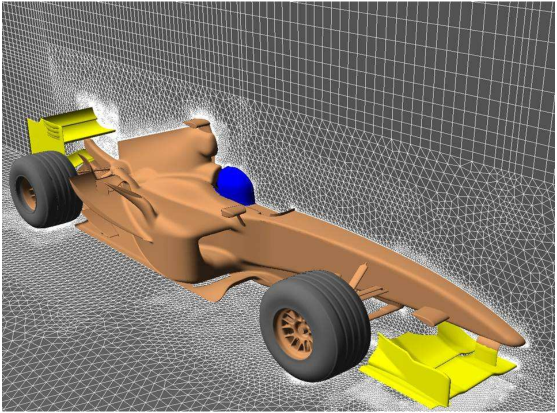
- Sample impact: Formula 1 arrangement optimization (continue)
  - car after design optimization

Figure: Rear-view mirrors are on sticks in the wake of the tires

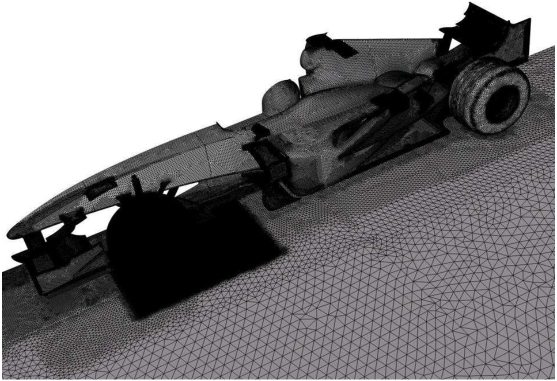
Anatomy of CFD

- Anatomy of an overall process
  - clean CAD model
  - specify mathematical models (inviscid, viscous laminar, viscous turbulent, turbulence model, etc.)
  - generate grid (or mesh) (discretize the computational domain)
    - structured grid
    - multi-block structured grid
    - unstructured
    - Chimera (overset) grids
  - specify equations of state
  - prescribe boundary conditions
  - select steady or unsteady flow solution
  - define initial conditions
  - specify output
  - specify semi-discretization process (spatial discretization)
    - finite difference approximation
    - finite volume approximation
    - finite element approximation
  - specify time or (pseudo-time) discretization
    - explicit ODE (Ordinary Differential Equation) solver
    - implicit ODE solver and equation solver
  - perform post-processing

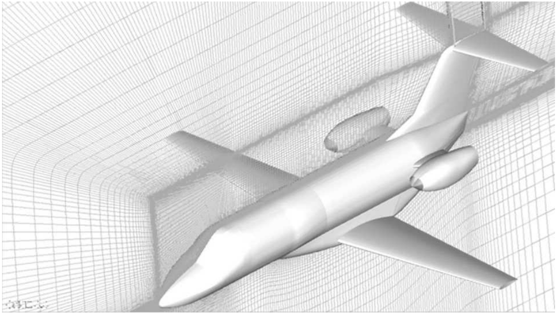
- Partial view of a CAD model and an unstructured grid



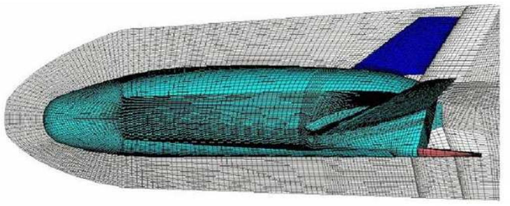
- Partial view of an unstructured surface grid



- Partial view of a structured grid



- Partial view of a multi-block structured grid



- Partial view of a set of overset grids