REE 307
Fluid Mechanics II

Lecture 4

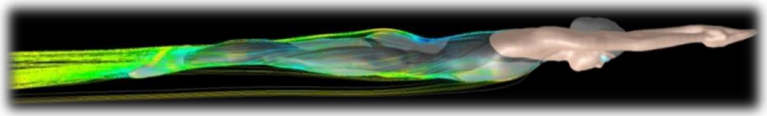
November 1, 2017

Dr./ Ahmed Mohamed Nagib Elmekawy

Zewail City for Science and Technology

Introduction to CFD

What is CFD?



Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena.

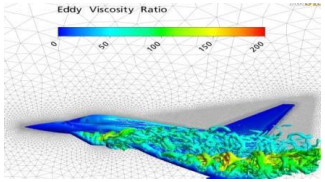
To predict these phenomena, CFD solves equations for conservation of mass, momentum, energy etc..

CFD can provide detailed information on the fluid flow behavior:

- Distribution of pressure, velocity, temperature, etc.
- Forces like Lift, Drag.. (external flows, Aero, Auto..)
- Distribution of multiple phases (gas-liquid, gas-solid..)
- Species composition (reactions, combustion, pollutants..)
- Much more

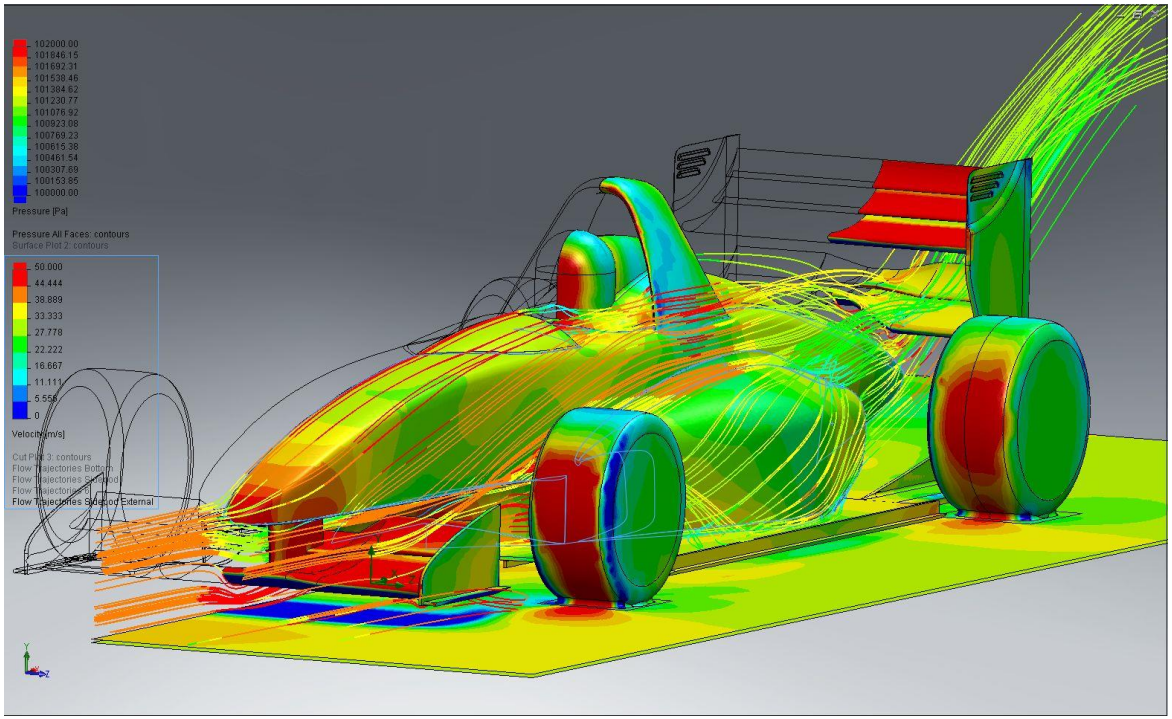
CFD is used in all stages of the engineering process:

- Conceptual studies of new designs
- Detailed product development
- Optimization
- Troubleshooting
- Redesign

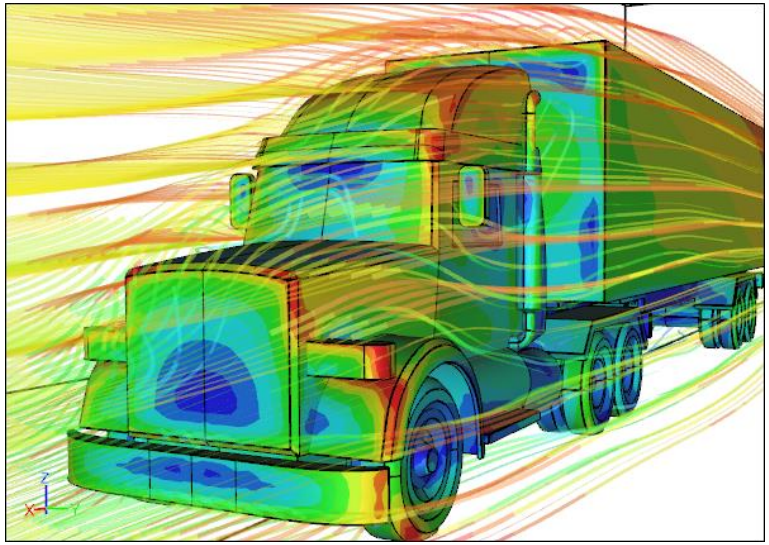


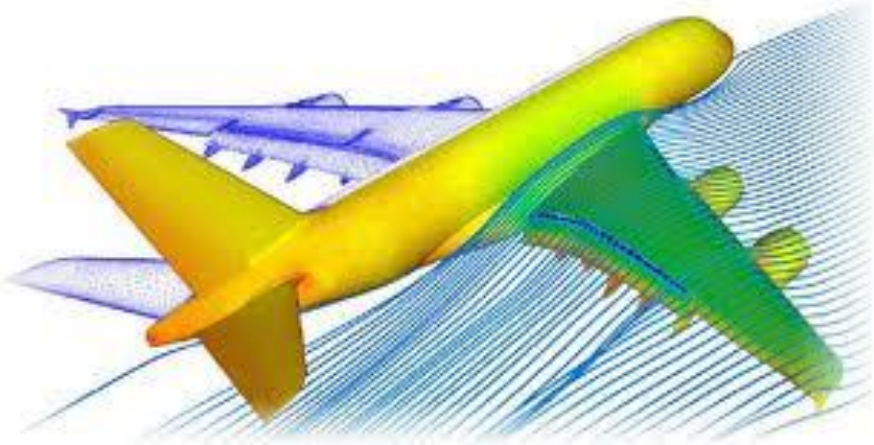
CFD analysis complements testing and experimentation by reducing total effort and cost required for experimentation and data acquisition

CFD Applications

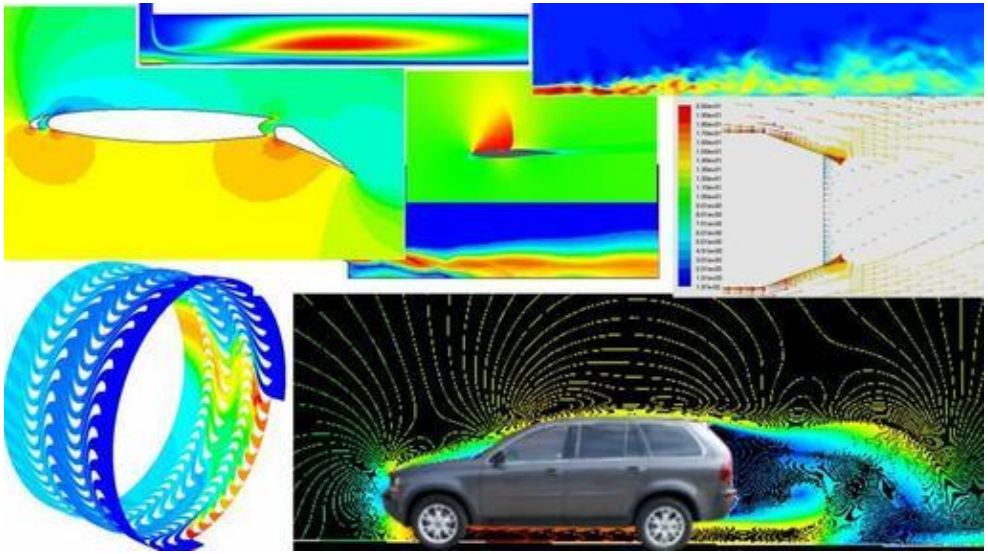


CFD Applications

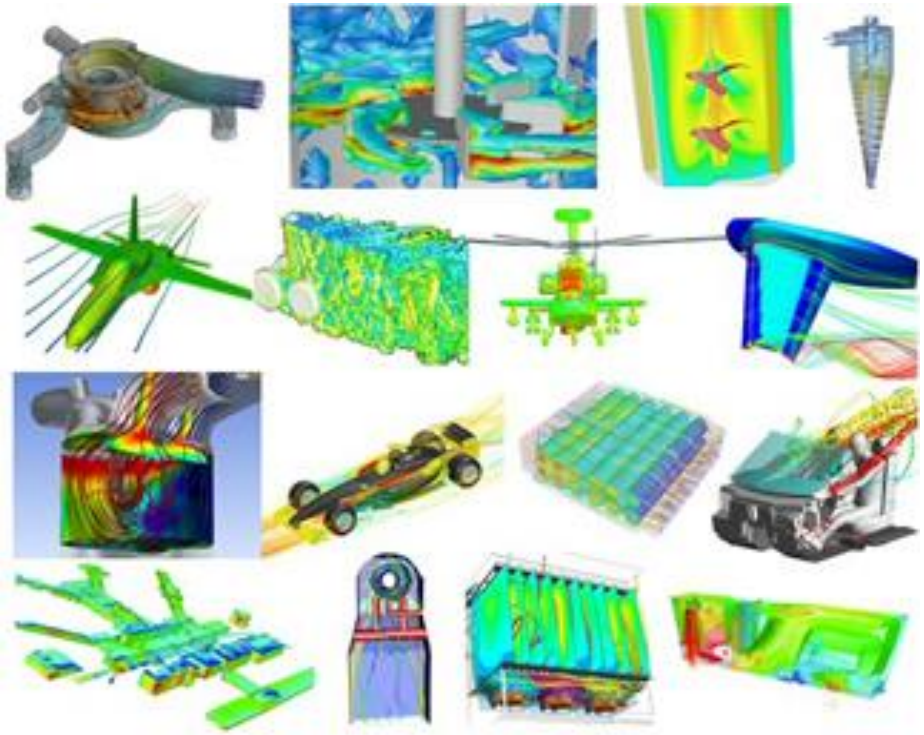




CFD Applications



CFD Applications

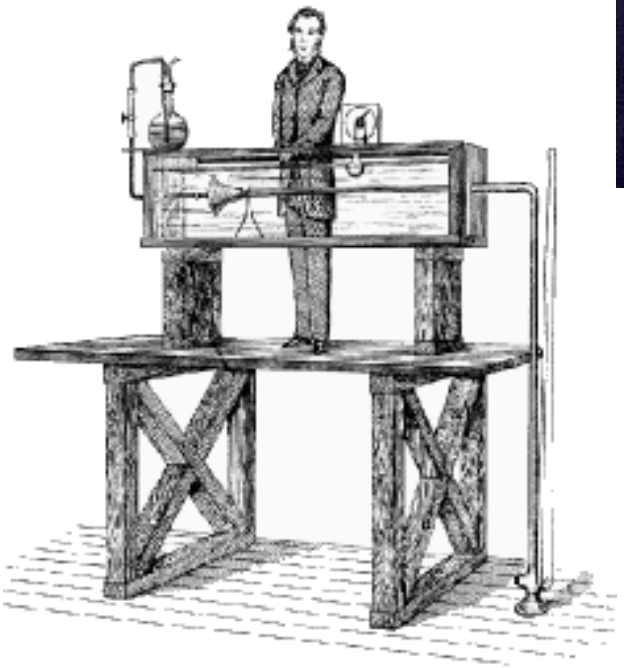


Euler and Navier-Stokes equations

- In 1759 Leonhard Euler published equations of motion for a fluid, applying Newton's second law of motion, F = ma. Euler's idea to express knowledge about fluid dynamics in the form of partial differential equations was a major breakthrough. A practical shortcoming of his flow model, however, was that it did not consider friction forces.
- George Stokes came with more advanced equations in 1845. These equations were already introduced in 1822 by Claude Navier, but only for incompressible fluids. With the freshly introduced equations, today called Navier-Stokes equations, understanding and controlling a large class of fluid flows seemed close at hand. The problem was reduced to the mathematical solution of these basic differential equations.
- Although the Navier-Stokes equations meant a considerable theoretical advance, the analytic mathematical solution of the full equations proved one bridge too far.

Osborne Reynolds - England (1842-1912)

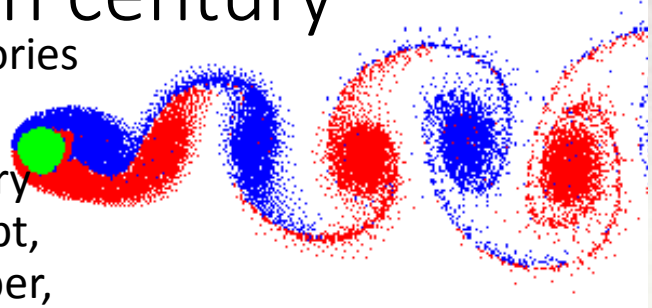
- Reynolds was a prolific writer who published almost 70 papers during his lifetime on a wide variety of science and engineering related topics.
- He is most well-known for the Reynolds number, which is the ratio between inertial and viscous forces in a fluid. This governs the transition from laminar to turbulent flow.

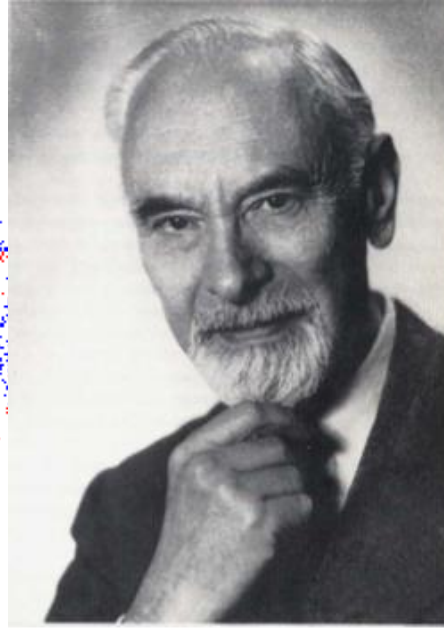


- Reynolds' apparatus consisted of a long glass pipe through which water could flow at different rates, controlled by a valve at the pipe exit. The state of the flow was visualized by a streak of dye injected at the entrance to the pipe. The flow rate was monitored by measuring the rate at which the free surface of the tank fell during draining. The immersion of the pipe in the tank provided temperature control due to the large thermal mass of the fluid.

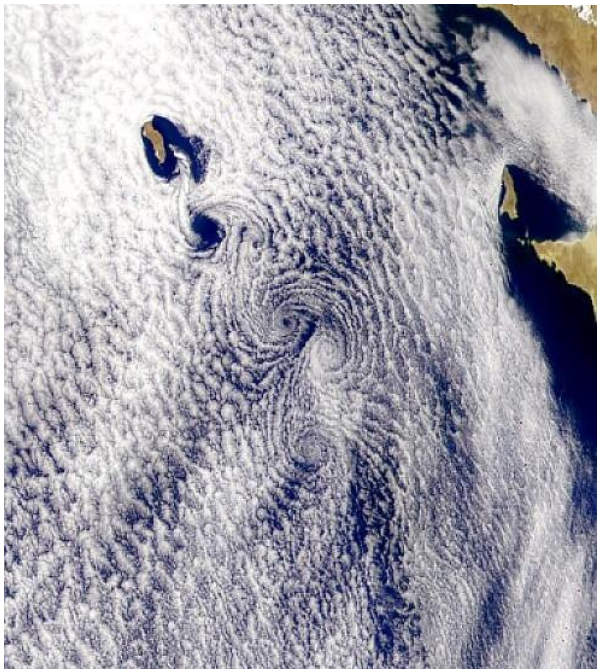
First part of the 20th century

- Much work was done on refining theories of boundary layers and turbulence.
- Ludwig Prandtl (1875-1953): boundary layer theory, the mixing length concept, compressible flows, the Prandtl number, and more.
- Theodore von Karman (1881-1963) analyzed what is now known as the von Karman vortex street.
- Geoffrey Ingram Taylor (1886-1975): statistical theory of turbulence and the Taylor microscale.
- Andrey Nikolaevich Kolmogorov (1903-1987): the Kolmogorov scales and the universal energy spectrum.
- George Keith Batchelor (1920-2000): contributions to the theory of homogeneous turbulence.





Ludwig Prandtl
4.2.1875 – 15.8.1953

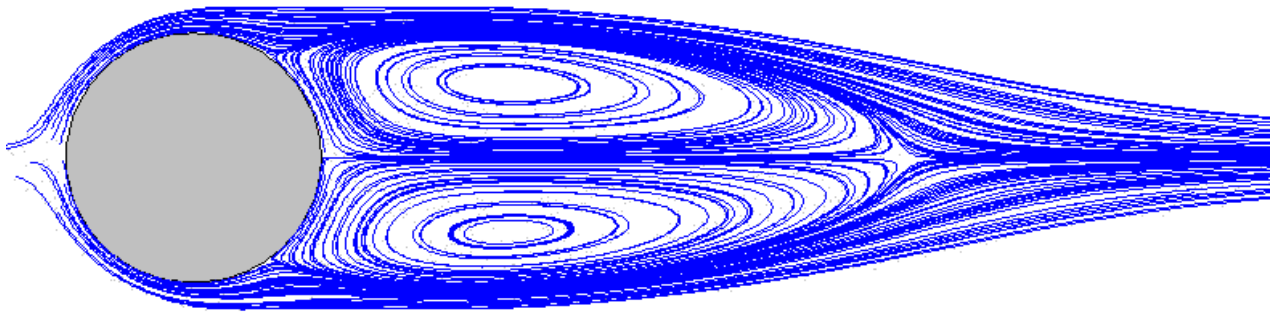


Lewis Fry Richardson (1881-1953)

- In 1922, Lewis Fry Richardson developed the first numerical weather prediction system.
  - Division of space into grid cells and the finite difference approximations of Bjerknes's "primitive differential equations."
  - His own attempt to calculate weather for a single eight-hour period took six weeks and ended in failure.
- His model's enormous calculation requirements led Richardson to propose a solution he called the "forecast-factory."
  - The "factory" would have filled a vast stadium with 64,000 people.
  - Each one, armed with a mechanical calculator, would perform part of the calculation.
  - A leader in the center, using colored signal lights and telegraph communication, would coordinate the forecast.

1930s to 1950s

- Earliest numerical solution: for flow past a cylinder (1933).
  - A.Thom, 'The Flow Past Circular Cylinders at Low Speeds', Proc. Royal Society, A141, pp. 651-666, London, 1933
- Kawaguti obtained a solution for flow around a cylinder, in 1953 by using a mechanical desk calculator, working 20 hours per week for 18 months, citing: "*a considerable amount of labour and endurance*."
  - M. Kawaguti, 'Numerical Solution of the NS Equations for the Flow Around a Circular Cylinder at Reynolds Number 40', Journal of Phy. Soc. Japan, vol. 8, pp. 747-757, 1953.



1960s and 1970s

- During the 1960s the theoretical division at Los Alamos contributed many numerical methods that are still in use today, such as the following methods:
  - Particle-In-Cell (PIC).
  - Marker-and-Cell (MAC).
  - Vorticity-Streamfunction Methods.
  - Arbitrary Lagrangian-Eulerian (ALE).
  - k-ε turbulence model.
- During the 1970s a group working under D. Brian Spalding, at Imperial College, London, develop:
  - Parabolic flow codes (GENMIX).
  - Vorticity-Streamfunction based codes.
  - The SIMPLE algorithm and the TEACH code.
  - The form of the k-ε equations that are used today.
  - Upwind differencing.
  - ‘Eddy break-up’ and ‘presumed pdf’ combustion models.
- In 1980 Suhas V. Patankar publishes *Numerical Heat Transfer and Fluid Flow*, probably the most influential book on CFD to date.

1980s and 1990s

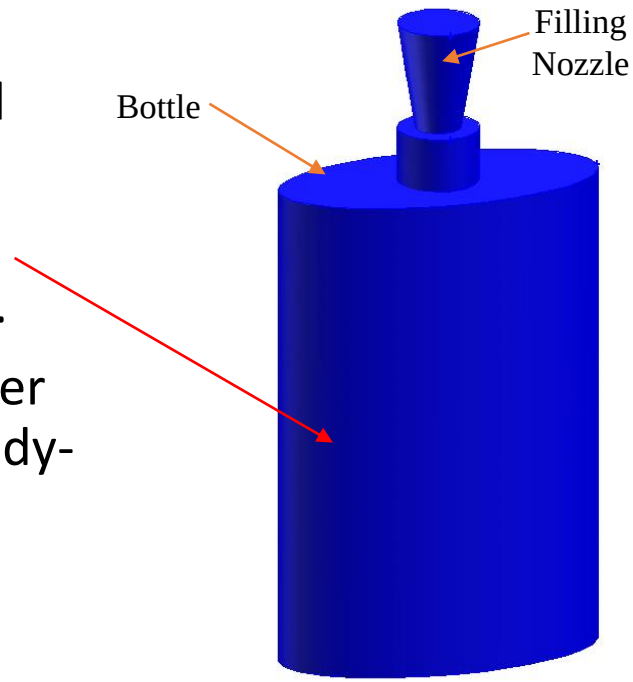
- Previously, CFD was performed using academic, research and in-house codes. When one wanted to perform a CFD calculation, one had to write a program.
- This is the period during which most commercial CFD codes originated that are available today:
  - Fluent (UK and US).
  - CFX (UK and Canada).
  - Fidap (US).
  - Polyflow (Belgium).
  - Phoenix (UK).
  - Star CD (UK).
  - Flow 3d (US).
  - ESI/CFDRC (US).
  - SCRYU (Japan).
  - and more, see www.cfdreview.com.

What is computational fluid dynamics?

- 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.
- The result of CFD analyses is relevant engineering data used in:
  - Conceptual studies of new designs.
  - Detailed product development.
  - Troubleshooting.
  - Redesign.
- CFD analysis complements testing and experimentation.
  - Reduces the total effort required in the laboratory.

CFD - how it works

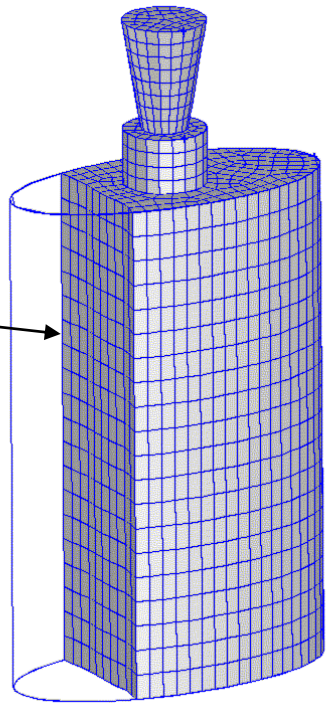
- Analysis begins with a mathematical model of a physical problem.
- Conservation of matter, momentum, and energy must be satisfied throughout the region of interest.
- Fluid properties are modeled empirically.
- Simplifying assumptions are made in order to make the problem tractable (e.g., steady-state, incompressible, inviscid, two-dimensional).
- Provide appropriate initial and boundary conditions for the problem.



Domain for bottle filling problem.

CFD - how it works (2)

- CFD applies numerical methods (called discretization) to develop approximations of the governing equations of fluid mechanics in the fluid region of interest.
  - Governing differential equations: algebraic.
  - The collection of cells is called the grid.
  - The set of algebraic equations are solved numerically (on a computer) for the flow field variables at each node or cell.
  - System of equations are solved simultaneously to provide solution.
- The solution is post-processed to extract quantities of interest (e.g. lift, drag, torque, heat transfer, separation, pressure loss, etc.).



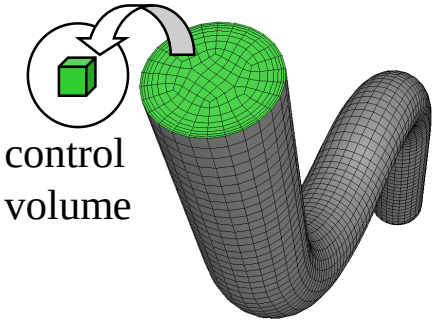
Mesh for bottle filling problem.

Discretization

- Domain is discretized into a finite set of control volumes or cells. The discretized domain is called the “grid” or the “mesh.”
- General conservation (transport) equations for mass, momentum, energy, etc., are discretized into algebraic equations.
- All equations are solved to render flow field.

$$\underbrace{\frac{\partial}{\partial t}\int_V \rho\phi dV}_{\text{unsteady}} + \underbrace{\oint_A \rho\phi\mathbf{V}\cdot d\mathbf{A}}_{\text{convection}} = \underbrace{\oint_A \Gamma\nabla\phi\cdot d\mathbf{A}}_{\text{diffusion}} + \underbrace{\int_V S_\phi dV}_{\text{generation}}$$

Eqn.	ϕ
continuity	1
x-mom.	u
y-mom.	v
energy	h



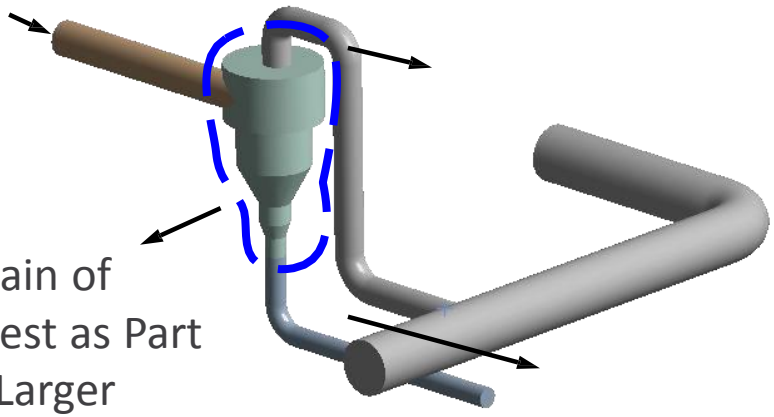
Fluid region of pipe flow discretized into finite set of control volumes (mesh).

Step 2. Identify the Domain You Will Model

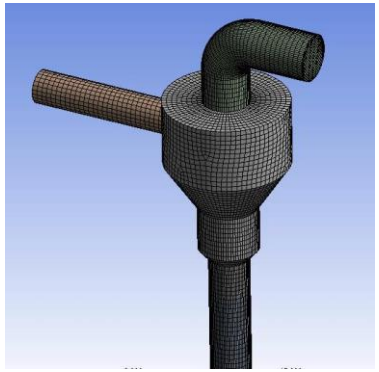
- **How will you isolate a piece of the complete physical system?**

- **Where will the computational domain begin and end?**
  - Do you have boundary condition information at these boundaries?
  - Can the boundary condition types accommodate that information?
  - Can you extend the domain to a point where reasonable data exists?

- **Can it be simplified or approximated as a 2D or axi-symmetric problem?**



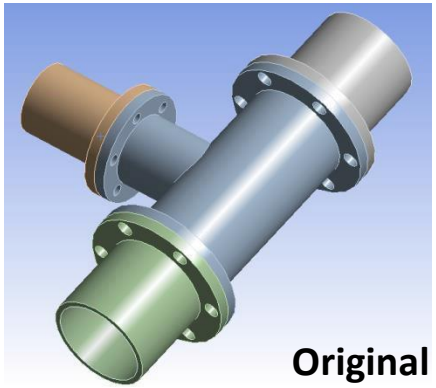
Domain of Interest as Part of a Larger System (not modeled)



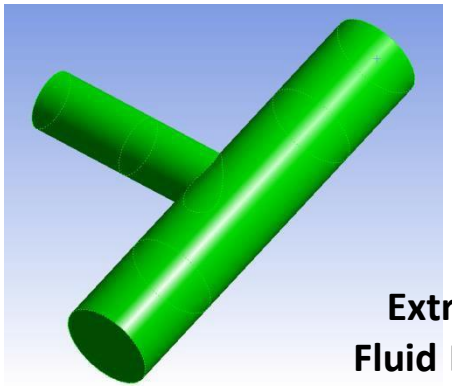
Domain of interest isolated and meshed for CFD simulation.

Step 3. Create a Solid Model of the Domain

- **How will you obtain a model of the *fluid* region?**
  - Make use of existing CAD models?
  - Extract the fluid region from a solid part?
  - Create from scratch?
- **Can you simplify the geometry?**
  - Remove unnecessary features that would complicate meshing ;fillets, bolts…?
  - Make use of symmetry or periodicity?
    - Are both the flow <u>and</u> boundary conditions symmetric / periodic?
- **Do you need to split the model so that boundary conditions or domains can be created?**



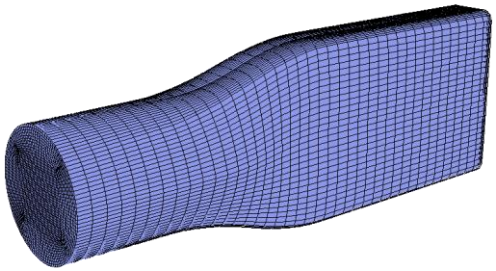
Original CAD Part

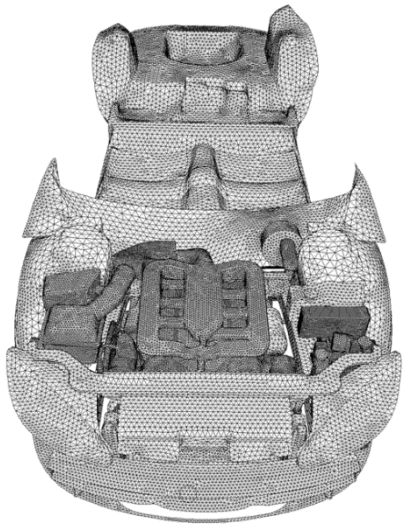


Extracted Fluid Region

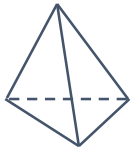
Step 4. Design and Create the Mesh

- **What degree of mesh resolution is required in each region of the domain?**
  - Can you predict regions of high gradients?
    - The mesh must resolve geometric features of interest and capture gradients of concern, e.g. velocity, pressure, temperature gradients
  - Will you use adaption to add resolution?
- **What type of mesh is most appropriate?**
  - How complex is the geometry?
  - Can you use a quad/hex mesh or is a tri/tet or hybrid mesh suitable?
  - Are non-conformal interfaces needed?
- **Do you have sufficient computer resources?**
  - How many cells/nodes are required?
  - How many physical models will be used?

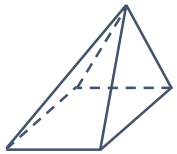




- Should you use a quad/hex grid, a tri/tet grid, a hybrid grid, or a non-conformal grid?
- What degree of grid resolution is required in each region of the domain?
- How many cells are required for the problem?
- Will you use adaption to add resolution?
- Do you have sufficient computer memory?

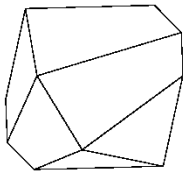


tetrahedron

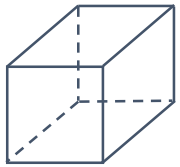


pyramid

triangle



arbitrary polyhedron



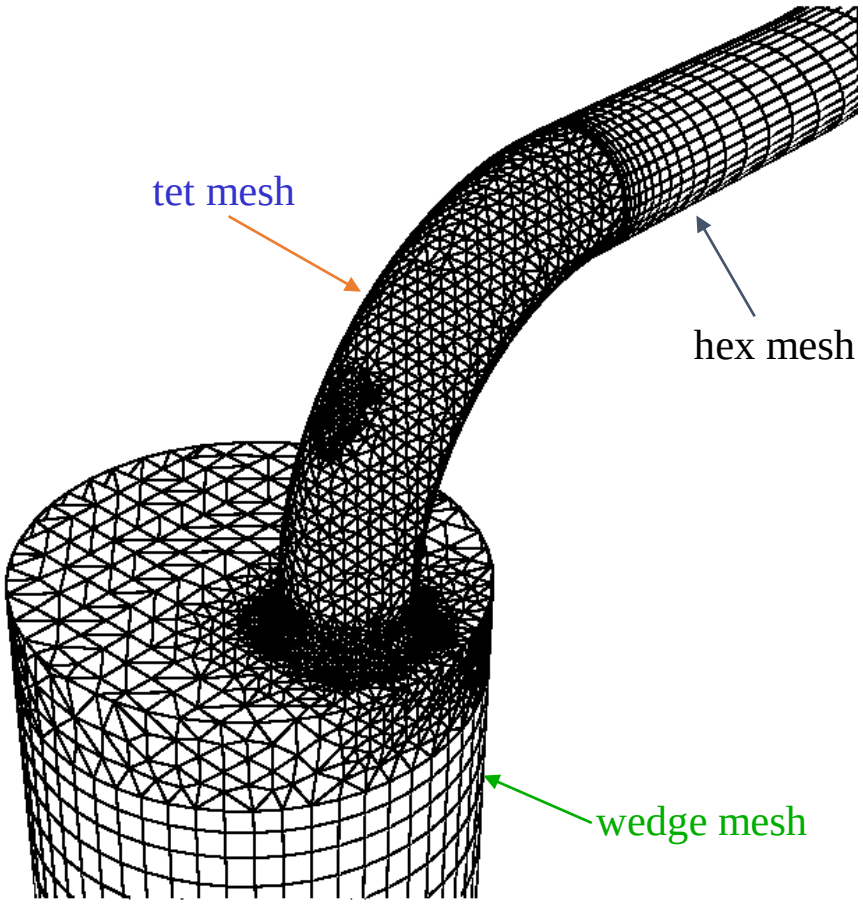
hexahedron

prism or wedge

quadrilateral

Hybrid mesh example

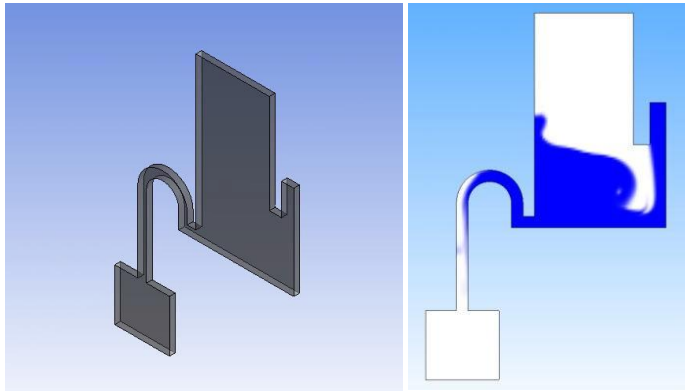
- Valve port grid.
- Specific regions can be meshed with different cell types.
- Both efficiency and accuracy are enhanced relative to a hexahedral or tetrahedral mesh alone.



Hybrid mesh for an
IC engine valve port

Step 5. Set Up the Solver

- **For a given problem, you will need to:**
  - Define material properties
    - Fluid
    - Solid
    - Mixture
  - Select appropriate physical models
    - Turbulence, combustion, multiphase, etc.
  - Prescribe operating conditions
  - Prescribe boundary conditions at all boundary zones
  - Provide initial values or a previous solution
  - Set up solver controls
  - Set up convergence monitors



For complex problems solving a simplified or 2D problem will provide valuable experience with the models and solver settings for your problem in a short amount of time

Step 6. Compute the Solution

- **The discretized conservation equations are solved iteratively until convergence**
- **Convergence is reached when:**

– Changes in solution variables from one iteration to the next are negligible

- Residuals provide a mechanism to help monitor this trend

– Overall property conservation is achieved

•Imbalances measure global conservation

– Quantities of interest (e.g. drag, pressure drop) have reached steady values

•Monitor points track quantities of interest

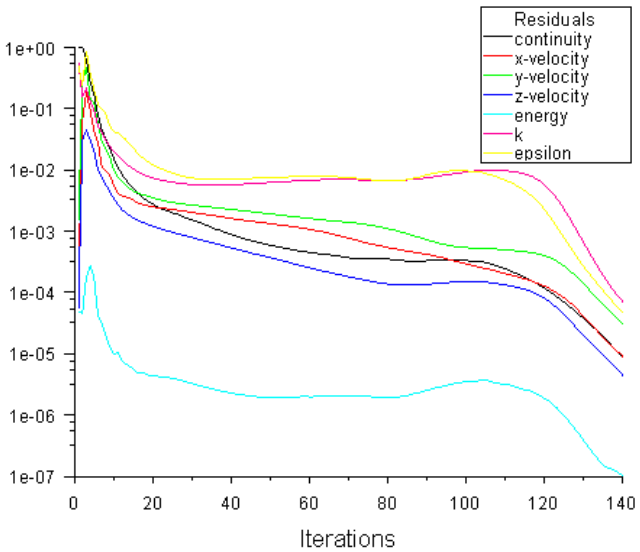
- **The accuracy of a *converged* solution is dependent upon:**

– Appropriateness and accuracy of physical models

– Assumptions made

– Mesh resolution and independence

– Numerical errors



A converged and mesh-independent solution on a well-posed problem will provide useful engineering results!

Step 7. Examine the Results

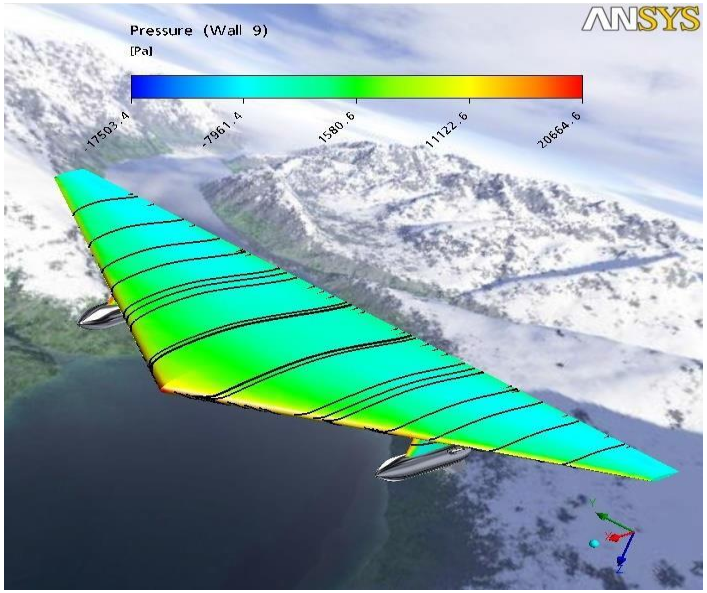
- **Examine the results to review solution and extract useful data**

– Visualization Tools can be used to answer such questions as:

  - What is the overall flow pattern?
  - Is there separation?
  - Where do shocks, shear layers, etc. form?
  - Are key flow features being resolved?

– Numerical Reporting Tools can be used to calculate quantitative results:

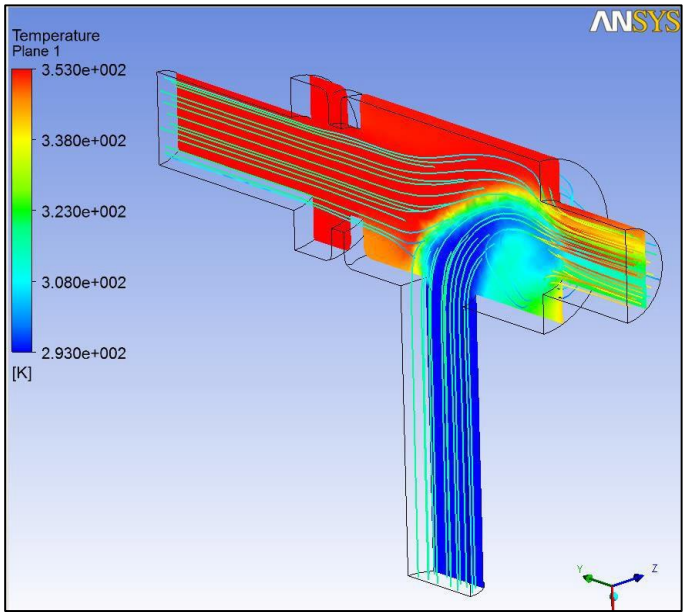
  - Forces and Moments
  - Average heat transfer coefficients
  - Surface and Volume integrated quantities
  - Flux Balances



Examine results to ensure correct physical behavior and conservation of mass energy and other conserved quantities. High residuals may be caused by just a few poor quality cells.

Step 8. Consider Revisions to the Model

- **Are the physical models appropriate?**
  - Is the flow turbulent?
  - Is the flow unsteady?
  - Are there compressibility effects?
  - Are there 3D effects?
- **Are the boundary conditions correct?**
  - Is the computational domain large enough?
  - Are boundary conditions appropriate?
  - Are boundary values reasonable?
- **Is the mesh adequate?**
  - Does the solution change significantly with a refined mesh, or is the solution mesh independent?
  - Does the mesh resolution of the geometry need to be improved?
  - Does the model contain poor quality cells?



High residuals may be caused by just a few poor quality cells

Use CFD with Other Tools to Maximize its Effect

Problem Identification

1. Define goals
2. Identify domain

Pre-Processing

3. Geometry
4. Mesh Physics
5. Solver Settings
6.

Solve

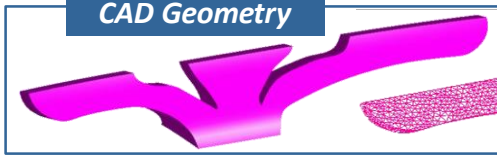
7. Compute solution

Post Processing

8. Examine results

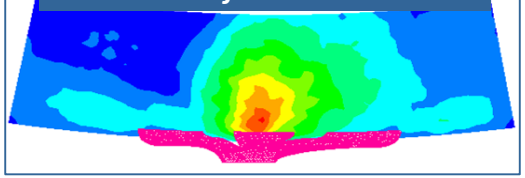
9. Update Model

CAD Geometry



Mesh

Thermal Profile on Windshield



Automated Optimization of Windshield Defroster with ANSYS DesignXplorer

Final Optimized Design

Prototype Testing → *Manufacturing*

Summary and Conclusions

- **Summary:**

– All CFD simulations (in all mainstream CFD software products) are approached using the steps just described

– Remember to first think about what the aims of the simulation are prior to creating the geometry and mesh

– Make sure the appropriate physical models are applied in the solver, and that the simulation is fully converged (more in a later lecture)

– Scrutinize the results, you may need to rework some of the earlier steps in light of the flow field obtained

1. **Define Your Modeling Goals**
2. Identify the Domain You Will Model
3. **Create a Geometric Model of the Domain**
4. Design and Create the Mesh
5. **Set Up the Solver Settings**
6. Compute the Solution
7. **Examine the Results**
8. Consider Revisions to the Model

Advantages of 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.

Advantages of CFD (2)

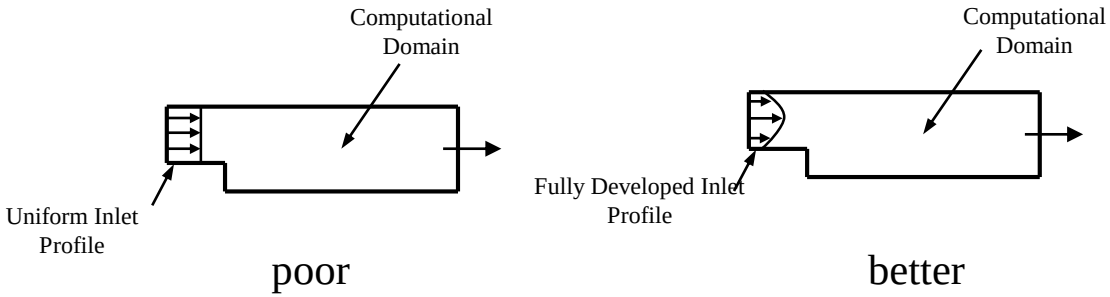
- Ability to simulate ideal conditions.
  - CFD allows great control over the physical process, and provides the ability to isolate specific phenomena for study.
  - Example: a heat transfer process can be idealized with adiabatic, constant heat flux, or constant temperature boundaries.
- Comprehensive 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.

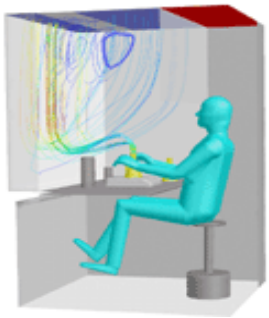
Limitations of CFD (2)

- 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.
  - Example: flow in a duct with sudden expansion. If flow is supplied to domain by a pipe, you should use a fully-developed profile for velocity rather than assume uniform conditions.

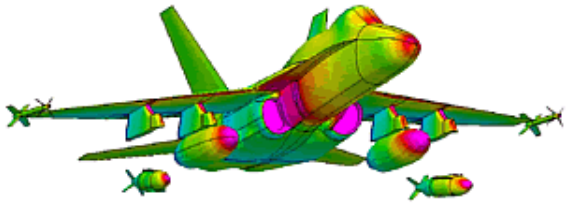


CFD is the simulation of fluids engineering systems using computers and modeling (mathematical/physical problem formulation) together with numerical methods (discretization methods, solvers, numerical parameters, and grid generations, etc.)

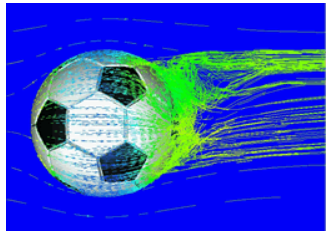
- CFD Applications
  - Aerospace
  - Automotive
  - Biomedical
  - Chemical Processing
  - HVAC
  - Hydraulics
  - Marine
  - Oil & Gas
  - Power Generation
  - Sports



Streamlines for workstation ventilation

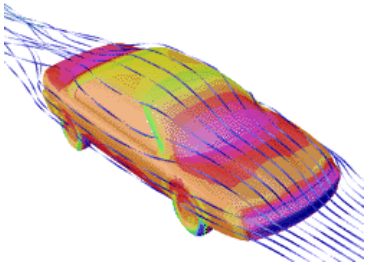


Flow over F18 fighter

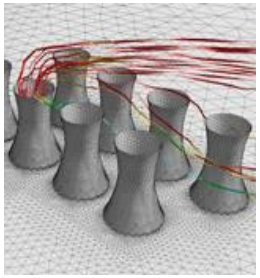


Flow over a football

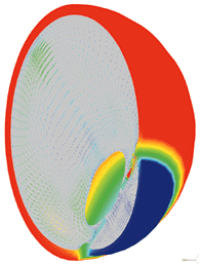
Lubricating mud flow over a drill bit



Aerodynamics of cars



Flow around cooling towers



Temperature in the eye following laser heating

Summary

- CFD is a method to numerically calculate heat transfer and fluid flow.
- Currently, its main application is as an engineering method, to provide data that is complementary to theoretical and experimental data. This is mainly the domain of commercially available codes and in-house codes at large companies.
- CFD can also be used for purely scientific studies, e.g. into the fundamentals of turbulence. This is more common in academic institutions and government research laboratories. Codes are usually developed to specifically study a certain problem.