Lecture (1) Introduction to Computational Fluid Dynamics (CFD)

What is CFD?

- CFD is the simulation of fluids engineering systems using modeling (mathematical physical problem formulation) and numerical methods (discretization methods, solvers, numerical parameters, and grid generations, etc.)
- Historically only Analytical Fluid Dynamics (AFD) and Experimental Fluid Dynamics (EFD).
- CFD made possible by the advent of digital computer and advancing with improvements of computer resources

Why use CFD?

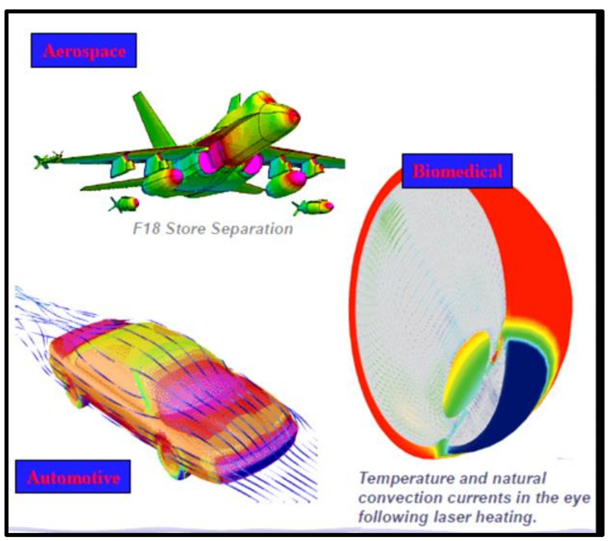
- Analysis and Design

1. Simulation-based design instead of "build & test"
    - More cost effective and more rapid than EFD
    - CFD provides high-fidelity database for diagnosing flow field
    - Provides guidance for planning experiments

2. Simulation of physical fluid phenomena that are difficult for experiments
    - Full scale simulations (e.g., ships and airplanes)
    - Environmental effects (wind, weather, etc.)
    - Hazards (e.g., explosions, radiation, pollution)
    - Physics (e.g., planetary boundary layer, stellar evolution)
    - Knowledge and exploration of flow physics

Where is CFD used?

- Aerospace
- •Automotive
- •Biomedical
- •Chemical Processing
- •HVAC
- •Hydraulics
- •Marine
- •Oil & Gas
- •Power Generation



- •Sports

Modeling

Modeling is the mathematical physics problem formulation in terms of a continuous initial boundary value problem (IBVP)•IBVP is in the form of Partial Differential Equations (PDEs) with appropriate boundary conditions and initial conditions.

Modeling includes:

1. Geometry and domain
2. Coordinates
3. Governing equations
4. Flow conditions
5. Initial and boundary conditions
6. Selection of models for different applications

Modeling (geometry and domain)

Simple geometriescan be easily created by few geometric parameters (e.g. circular pipe)

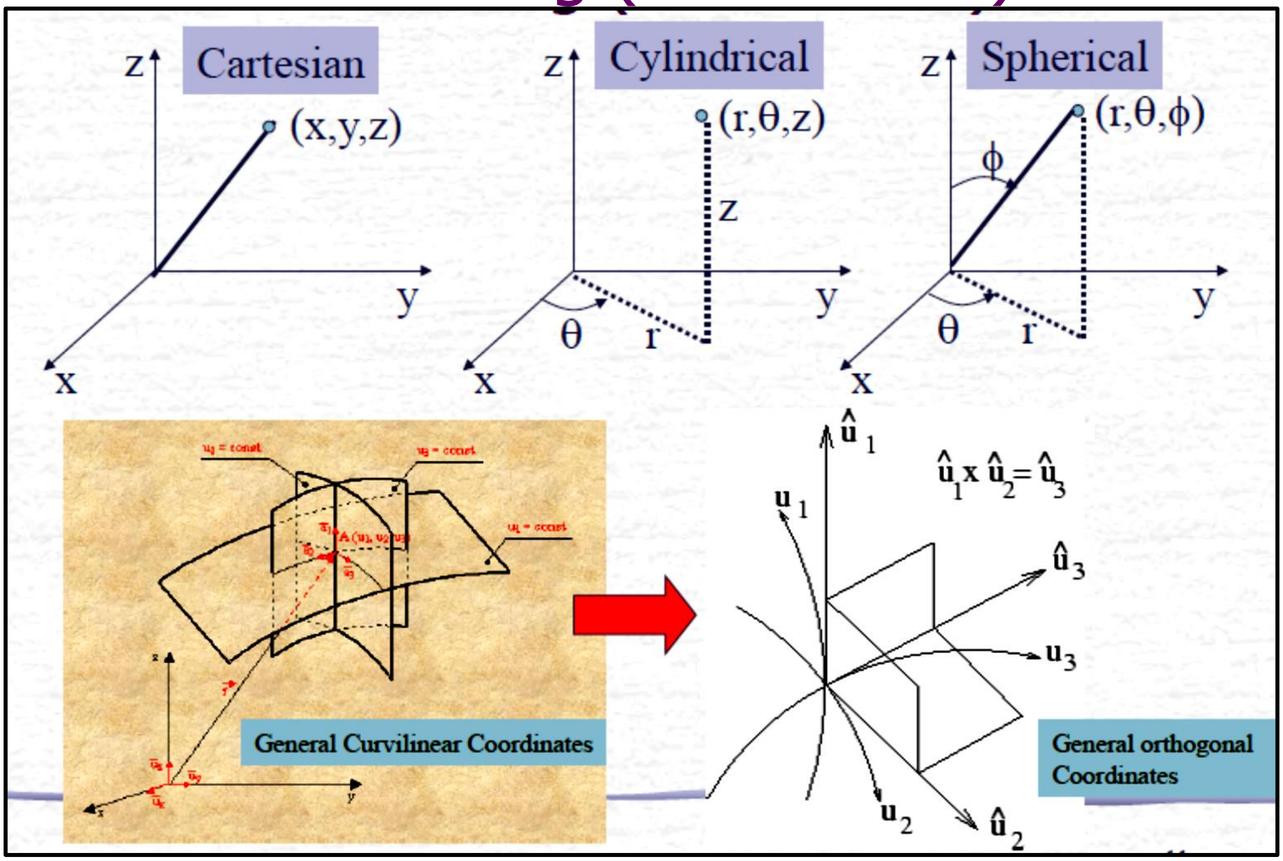
Complex geometriesmust be created by the partial differential equations or importing the database of the geometry(e.g. airfoil) into commercial software

Domain: size and shape

- Typical approaches •Geometry approximation
- CAD/CAE integration: use of industry standards such as Parasolid, ACIS, STEP, or IGES, etc.

- The three coordinates: Cartesian system (x,y,z), cylindrical system (r, θ, z), and spherical system(r, θ, Φ) should be appropriately chosen for a better resolution of the geometry (e.g. cylindrical for circular pipe).

Modeling (coordinates)



Modeling (governing equations)

- Navier-Stokes equations (3D in Cartesian coordinates)

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

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

$$\underbrace{\rho\frac{\partial w}{\partial t}}_{\text{Local acceleration}}+\underbrace{\rho u\frac{\partial w}{\partial x}+\rho v\frac{\partial w}{\partial y}+\rho w\frac{\partial w}{\partial z}}_{\text{Convection}}=\underbrace{-\frac{\partial \hat{p}}{\partial z}}_{\text{Piezometric pressure gradient}}+\underbrace{\mu\left[\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right]}_{\text{Viscous terms}}$$

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

$$p=\rho RT$$ Equation of state

$$R\frac{D^2R}{Dt^2}+\frac{3}{2}\left(\frac{DR}{Dt}\right)^2=\frac{p_v-p}{\rho_L}$$ Rayleigh Equation

Modeling (flow conditions)

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

- Viscous vs. inviscid (Re: Reynolds No.)
- External flow or internal flow (wall bounded or not)
- Turbulent vs. laminar (Re)
- Incompressible vs. compressible (Ma: Mach No.)
- Single-vs. multi-phase (Ca: Cavitation No.)
- Thermal/density effects (Pr: PrandtlNo., γ, Gr: Grashof No., Ec)

- Free-surface flow (Fr) and surface tension (We)
- Chemical reactions and combustion (Pe, Da)
- etc…

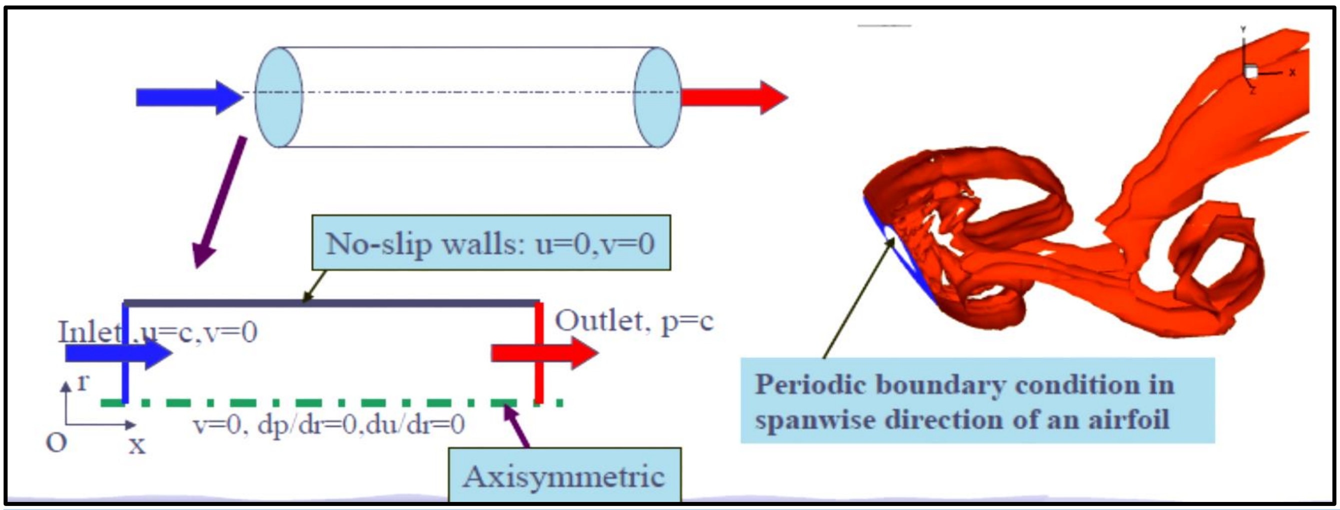
Modeling (initial conditions)

Initial conditions (ICS, steady/unsteady flows)

- ICs should not affect final results and only affect convergence path, i.e. number of iterations (steady) or time steps (unsteady) need to reach converged solutions.
- More reasonable guess can speed up the convergence
- For complicated unsteady flow problems, CFD codes are usually run in the steady mode for a few iterations for getting a better initial conditions

Modeling (boundary conditions)

Boundary conditions: No-slip or slip-free on walls, periodic, inlet (velocity inlet, mass flow rate, constant pressure, etc.), outlet (constant pressure, velocity convective, numerical beach, zero-gradient), and non-reflecting (for compressible flows, such as acoustics), etc.



Modeling (selection of models)

CFD codes typically designed for solving certain fluid phenomenon by applying different models

- Viscous vs. inviscid (Re)
- Turbulent vs. laminar (Re, Turbulent models)
- Incompressible vs. compressible (Ma, equation of state)
- Single-vs. multi-phase (Ca, cavitation model, two-fluid model)
- Thermal/density effects and energy equation (Pr, γ, Gr, Ec, conservation of energy)
- Free-surface flow (Fr, level-set & surface tracking model) and surface tension (We, bubble dynamic model)
- Chemical reactions and combustion (Chemical reaction model)
- etc…