Computational Fluid Dynamics, Simulation, and Computational tools in Wind Engineering

Ahsan Kareem

SimCenter
CyberInfrastructure

CFD Challenges and Perception

• **Advances in CFD** with applications to wind and its effects are critical in enhancing the performance of built environment under natural hazards.

• Currently **experiments in wind tunnel/vortical flow facilities** ........ **scale effects** may impact the availability and quality of results

• **CFD offers**, in principle, a most effective means of overcoming some of the challenges and it combined with advanced data analytics approaches has the promise to **provide data with great details** that are not possible with limited accessibility of sensors in experiments

• **Major challenges** include numerically capturing the complexity of massively separated flows around structures compounded by **multi-scale fluctuations** in the flow due to turbulence and their **nonlinear interactions**.
CFD Group Goals

- To enhance the quality and accessibility of computational tools to build a large community of users who in turn will help usher new advances
- Facilitate sharing of computational and data resources through an extensible set of CFD software suites
- Ease the use of advanced CFD models, methods and codes in real-world complex problems
- Enable detailed validation of computational models
- Create interfaces to establish linkages between a range of existing and future interoperable software suites without extensive training in each suites and their input/data needs
- Facilitate advanced analytics, e.g., uncertainty quantification, machine learning and optimization
- Provide students access to advanced computational fluid dynamics tools
- Lowering or eliminating the perceived barriers to using CFD based codes
- Facilitate collaboration in wind engineering research and especially between computational and experimental researchers

Increasing Complexity in flow around Bodies

Navier Stokes Equations:

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

Convective acceleration

Turbulence Spectrum
Computational Fluid Dynamics (CFD) Basics

- **RANS**
  - Reynolds Averaged Navier Stokes
  - Complete Model of Turbulence, very effective
- **LES**
  - Large Eddy Simulation
  - Model small scales, calculate large scales
- **DNS**
  - Direct Solution of Navier Stokes-Equations

Turbulence cannot capture every detail. Coarse graining via SGS Model.

Turbulence modeling

- **LES**
  - Sub grid-scale model
  - Dynamic SGS model
  - ... ...
- **RANS**
  - $k$-$\varepsilon$ model
  - $k$-$\omega$ model
  - Reynolds stress model
  - ... ...

Complexity of flow field

- Separation
- Reattachment
- Vortex shedding
- ...

Boundary conditions

- Inflow turbulence characteristics

High Reynolds number

- Grid resolution
- Wall boundary condition

Viscous sublayer

Buffer layer

Log-law region

Turbulence

Lateral region

Wall function

Separation point

Reattachment point

Inflow boundary

Lateral side boundary

Outflow boundary
Direct Simulations using OpenFOAM on Stampede2

Computational simulations using OpenFOAM @ Stampede2

Introduction to the use of OpenFOAM in DesignSafe-CI
Introduction to the use of OpenFOAM in DesignSafe-CI

Data Depot  →  Simulation  →  Workspace  →  Post-processing

Select the app and start the interactive session

Virtual Wind Tunnel

1) Tools to **CREATE** wind simulations  2) Interface to **RUN** wind simulations

- Establishing Virtual Community Users
- Shared Software
- Shared Hardware
- Collective Knowledge
- Crowd Sourcing
Client Program Installs on Windows, Mac or Linux

Provides Remote access to DesignSafe, runs simulations on TACC

Can upload geometries created in FreeCAD
Performs Mesh Generation

... and Simulation
Extensible Template Framework
Current templates for 2D and 3D rigid body simulations.

Planned future templates for:
- Custom inflows
- UQ
- Vibrating Structures
- Other user interests?

Extensible Result Framework

This work is based on material supported by the National Science Foundation under grant 1612843.
Software is Open-Source
And under iterative development

Turbulence inflow generation for wind engineering application

**Design Parameters**
- Mean velocity (e.g., power-law wind profile)
- Turbulent intensity
- Turbulent Spectra (e.g., von Karman turbulent spectra)
- Coherency

**Governing Equations**
- Navier-Stokes equation
\[ \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \nu \nabla^2 \mathbf{u} \]
- Continuity equation
\[ \mathbf{u} \cdot \nabla \rho = 0 \]

**Recycling Method**
- Weighted Amplitude Wave Superposition (WAWS)
  - Non-divergence-free
  - Hoshiya, 1972
- Random Flow Generation (RFG)
  - Divergence-free with Gaussian spectra not compatible with the spectra in the ABL
  - Kraichnan, 1970

**Synthetic Turbulence**
- Discrete RFG (DRFG)
  - Divergence-free with ABL turbulent spectra
  - Huang et al., 2010
- Consistent DRFG (CDRFG)
  - An improvement to DRFG
  - Aboshoza et al., 2015
Inserting the synthetic velocity field into the Navier-Stokes equation to compute the pressure field.

FSI in OpenFOAM: Partitioned strategy

**Fluid domain** $\Omega_F$ (Fluid dynamics computation)
- Solver: transient solver (PisoFoam, SimpleFoam)
- Turbulent model: LES model, RANS model, or user-defined turbulence model

**Interface: data transfer**
- Pressure, Velocity Interpolation (patchToPatchInterpolation)

**Solid domain** $\Omega_S$ (Structural mechanics computation)
- Mesh motion solver: generic motion solvers in OpenFOAM (Spring analogy, Laplace equation, linear pseudo-solid equation), user-defined solvers

**Numerical Simulation Example**

**Inserting the synthetic velocity field into the Navier-Stokes equation to compute the pressure field**

**Spectral plots**

**Numerical Simulation Example**
Verification Validation UQ

- CFD Simulations
- Database of Wind Tunnel Measurements
- Validations/Discrepancy/Multiple-model simulation
- UQ – Surrogate Modeling/Dakota

Validation

Computational Wind Tunnel

1 Billion Mesh Cells
- 6,144 Parallel Cores
- The K Computer

Kajima; Shimizu; TIT

Model-Fidelity

Low Model fidelity

- RANS
  - Low-fidelity flow simulation
  - Computationally efficient
  - Large modeling error

High Model fidelity

- LES
  - High-fidelity flow simulation
  - Computationally demanding
  - Higher accuracy

- DNS

A Multi-fidelity model approach

If it turns out that LES can be done on very coarse grids, it will be one of the few times that nature has been kind to us with regard to turbulent flow.

-- Ferziger, 1990
Uncertainty Quantification in CFD-based Simulation

• Aleatory variability
  • Natural randomness in a process
  • e.g. Variability in inflow free stream turbulence

• Epistemic (Model-form) uncertainty
  • Discrepancies between the mathematical model and the physical reality due to incomplete knowledge or limited data
  • e.g. Parameter calibration error in turbulence modeling

• Uncertainty propagation
  • Intrusive Methods
  • Non-Intrusive Methods
    – Only multiple solution of original version
    – Stochastic emulation (ROM, Surrogates)
  • Dakota offers a host of schemes

Stochastic emulation

• Design of experiments
  • Design variables: Inflow parameters \( I_u, L_u \) + Shape parameters \( \Delta y_1^*, \Delta y_2^* \)
  • Method: Latin Hypercube Sampling (89 samples)

• Computational simulation
  • URANS: \( k-\omega \) SST model with wall functions

\[
k = \frac{3}{2} \left( \frac{U_l y'^*}{T} \right)\]

\[
\omega = \frac{c_{\mu} \sqrt{k}}{L_u}
\]

• Surrogate calibration
  • Method: Ordinary kriging

Turbulence intensity \( I_u \)  Integral length scale \( L_u \)
Shape parameterization

Surrogate

Multi-objective optimization

Genetic algorithm

Output

Pareto front

Problem formulation

RANS

LE

Multi-fidelity model

Inflow Uncertainty

Workflow of the aerodynamic shape optimization considering inflow and model-form uncertainties separately

CFD Tools --- Plans and Progress

- OpenFOAM based simulations Directly via Stampede/DesignSafe Portal
  - Direct RANS URANS LES DES Models
- OpenFOAM based Virtual Wind Tunnel via DesignSafe Platform
- Inflow Simulator
- Aeroelastic: Fluid-Structure Interaction
- Validation
- Multi-Fidelity Modeling using OpenFOAM
  - Multi-Fidelity Modeling
  - Combining Multi-Fidelity Modeling with Machine Learning Modeling
- OpenFOAM-Dakota linkage (CFD-UQ)
  - RANS – input parameters -UQ
  - LES – Inflow parameters -UQ
  - Stochastic emulation UQ tool
- CFD-based input to PBE for wind, waves/surge/tsunami
- Digital Learning Hubs
- Collaboration with DesignSafe: Simulation and Data Groups (VORTEX-Winds)
Digital Virtual Learning Hub

- Remove the "fear out of CFD" in structural engineering community
- Virtual Wind Tunnel-VWT
- Using VWT promote crowdsourcing to get the masses help advance the field and build a community of users who can generate simulation results for a variety of cases to populate a database....e.g., "Mozak"
- Promote solid understanding of fluid mechanics appropriate for the simulation study and source of uncertainties and level of accuracy through an extensive on-line tutorials/webinars/seminars/blogs---CFDSI----Digital Learning Hubs
- Establish CFD-Wind Hazard “Chatrooms”
- Forge International collaborations...Japan, China, EU, UK, ....
- Validation and Benchmarking
- Hybrid use of NHERI-EFs (Wind Tunnels) for cross platform validation

Mozak: a game that crowdsources mapping of brain-cells