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

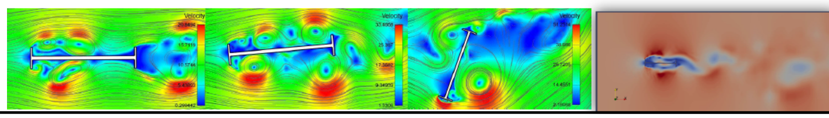
Work at the Human-Technology Frontier: Shaping the Future

nsf.gov

Ahsan Kareem

SimCenter

CyberInfrastructure



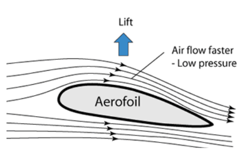
CFD Challenges and Perception

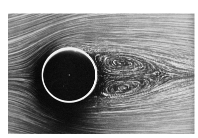
- **Advances in CFD** with applications to wind and its effects are critical in enhancing the performance of built environment under natural hazards.
- Currently **experiments in wind tunnel/vortical flow facilities** ……… ***scale effects*** may impact the availability and quality of results
- **CFD 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***.

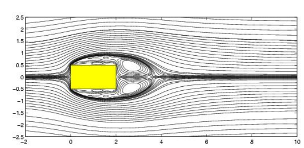
CFD Group Goals

- 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:

Convective acceleration

Inertial force per unit volume

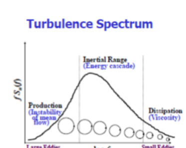
$$\rho\left[\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right]$$

Substantial acceleration

Local (Instantaneous) acceleration

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

Pressure gradient *Viscous stress*



Computational Fluid Dynamics(CFD) Basics

- **RANS**

Reynolds Averaged Navier Stokes

» Complete Model of Turbulence, very effective

- **LES**

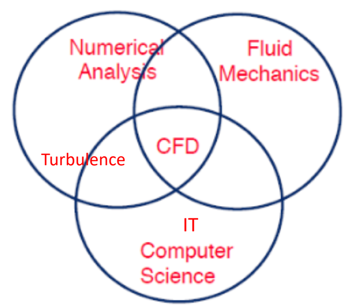
Large Eddy Simulation

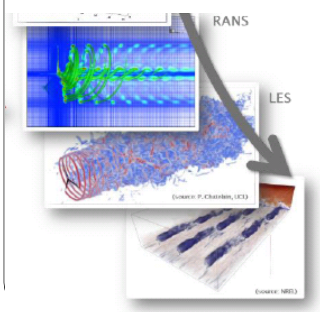
» Model small scales, calculate large scales

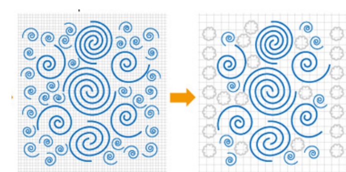
- **DNS**

Direct Solution of Navier Stokes-Equations

CFD is an interdisciplinary topic







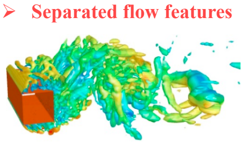
Cannot capture every detail   Coarse graining via SGS Model

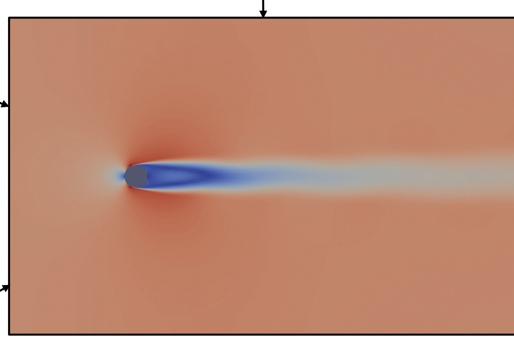
Turbulence modeling

LES
- Sub grid-scale model
- Dynamic SGS model
- ……

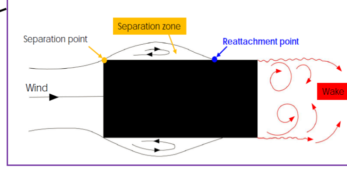
RANS
- k-ε model
- k-ω model
- Reynolds stress model
- ……

Sharp edges
- Separated flow features

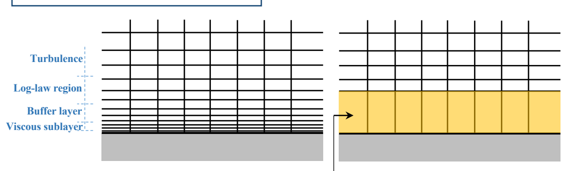




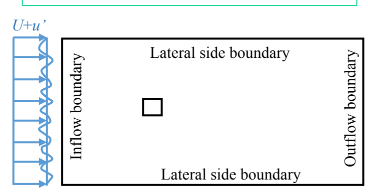
Complexity of flow field
- Separation
- Reattachment
- Vortex shedding
- ……

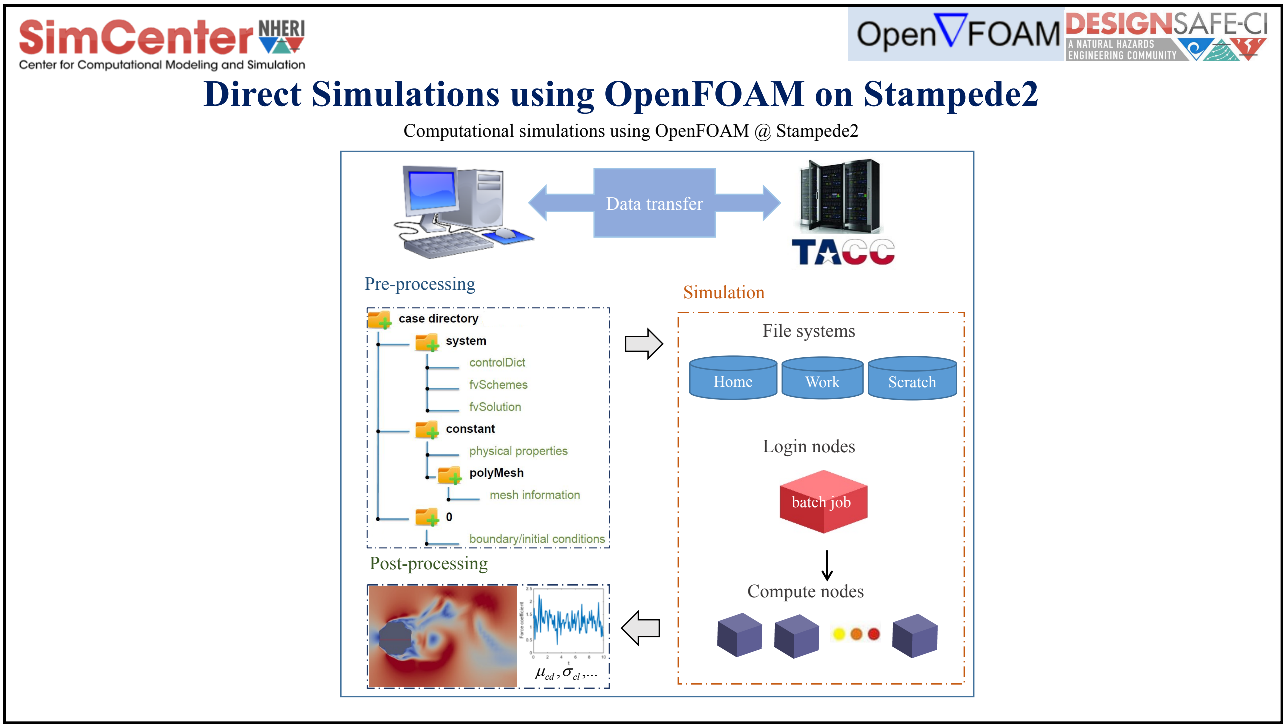


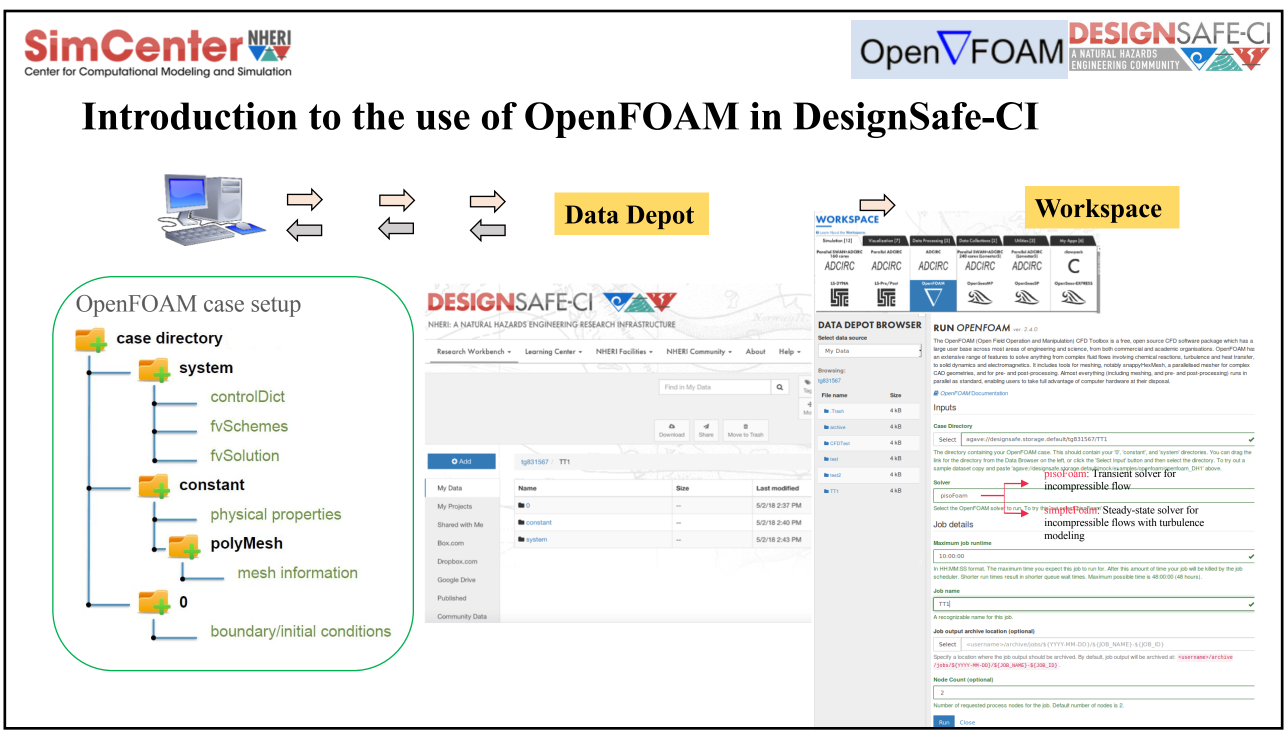
High Reynolds number
- Grid resolution
- Wall boundary condition



Boundary conditions
- Inflow turbulence characteristics



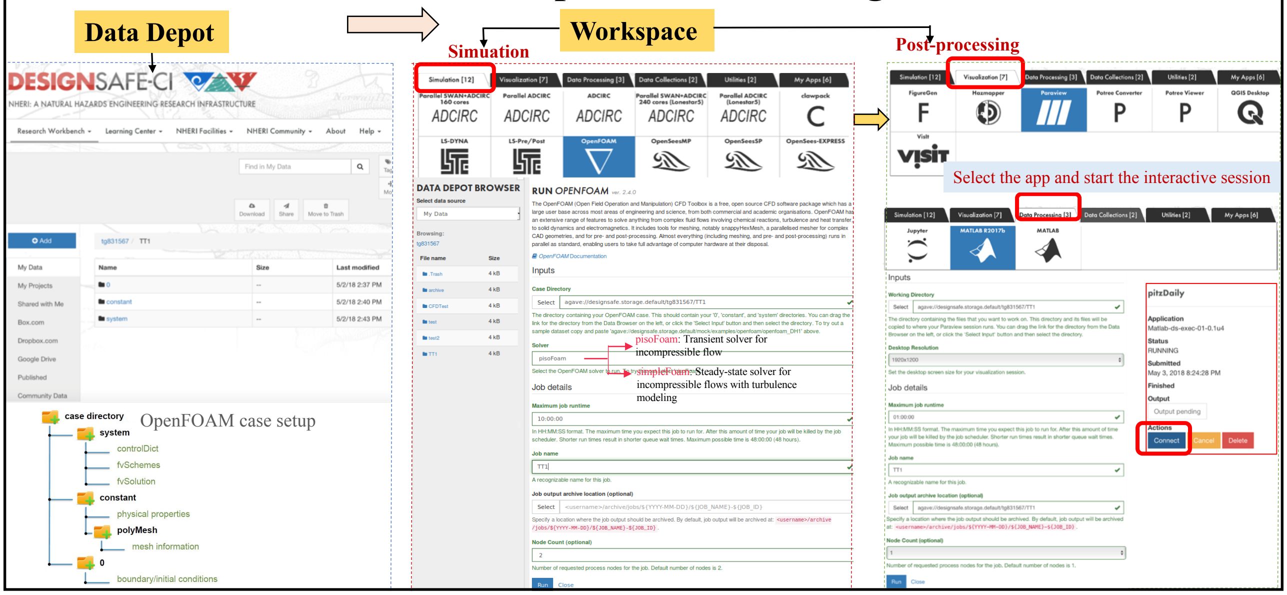

SimCenter NHERI
Center for Computational Modeling and Simulation
OpenFOAM
DESIGNSAFE-CI
A NATURAL HAZARDS ENGINEERING COMMUNITY
Direct Simulations using OpenFOAM on Stampede2
Computational simulations using OpenFOAM @ Stampede2
Data transfer
TACC
Pre-processing
case directory
system
controlDict
fvSchemes
fvSolution
constant
physical properties
polyMesh
mesh information
0
boundary/initial conditions
Simulation
File systems
Home
Work
Scratch
Login nodes
batch job
Compute nodes
Post-processing
$\mu_{cd}, \sigma_{cl}, \ldots$


SimCenter NHERI
Center for Computational Modeling and Simulation
OpenFOAM
DESIGNSAFE-CI
A NATURAL HAZARDS ENGINEERING COMMUNITY
Introduction to the use of OpenFOAM in DesignSafe-CI
Data Depot
Workspace
OpenFOAM case setup
case directory
system
controlDict
fvSchemes
fvSolution
constant
physical properties
polyMesh
mesh information
0
boundary/initial conditions
DESIGNSAFE-CI
WORKSPACE
DATA DEPOT BROWSER
RUN OPENFOAM
Inputs
pisoFoam: Transient solver for incompressible flow
simpleFoam: Steady-state solver for incompressible flows with turbulence modeling

SimCenter NHERI
Center for Computational Modeling and Simulation

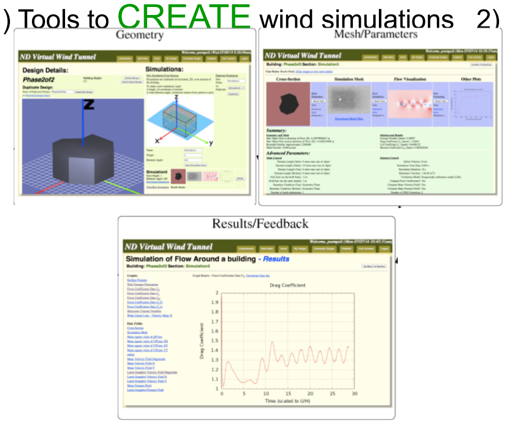
OpenFOAM DESIGNSAFE-CI A NATURAL HAZARDS ENGINEERING COMMUNITY

Introduction to the use of OpenFOAM in DesignSafe-CI



Virtual Wind Tunnel

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



Agave

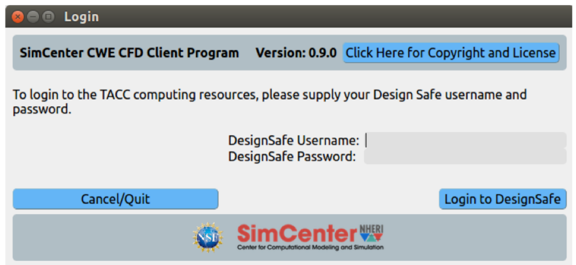
DESIGNSAFE-CI A NATURAL HAZARDS ENGINEERING COMMUNITY

TACC

- Establishing Virtual Community Users
- Shared Software
- Shared Hardware
- Collective Knowledge
- Crowd Sourcing

Client Program Installs on Windows, Mac or Linux

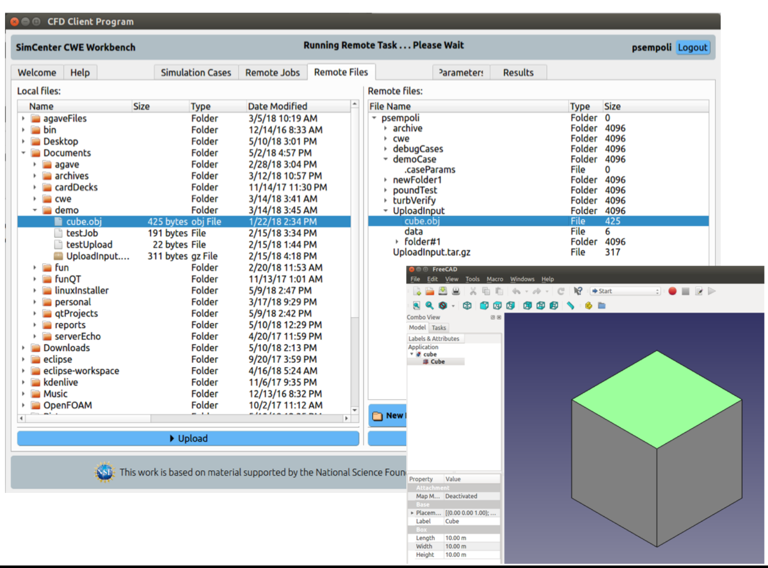
Windows



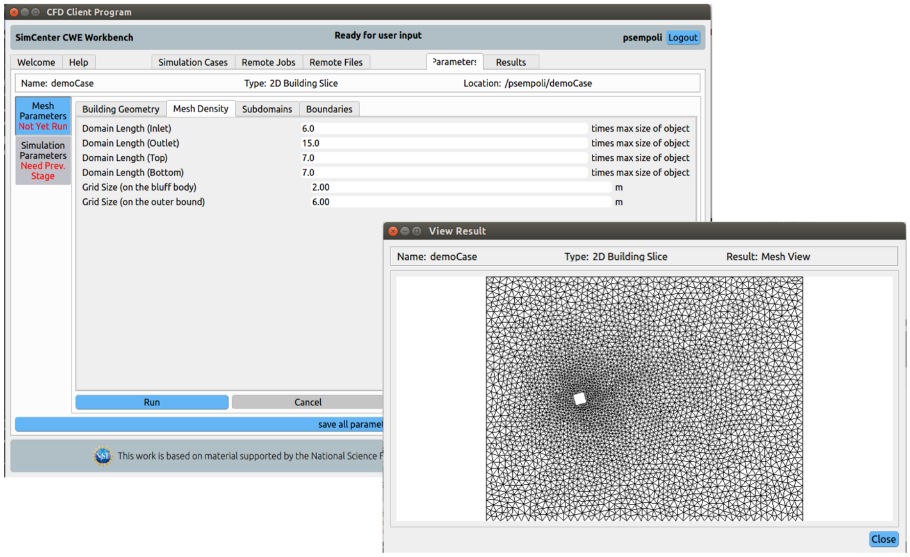
Mac OS

Provides Remote access to DesignSafe, runs simulations on TACC

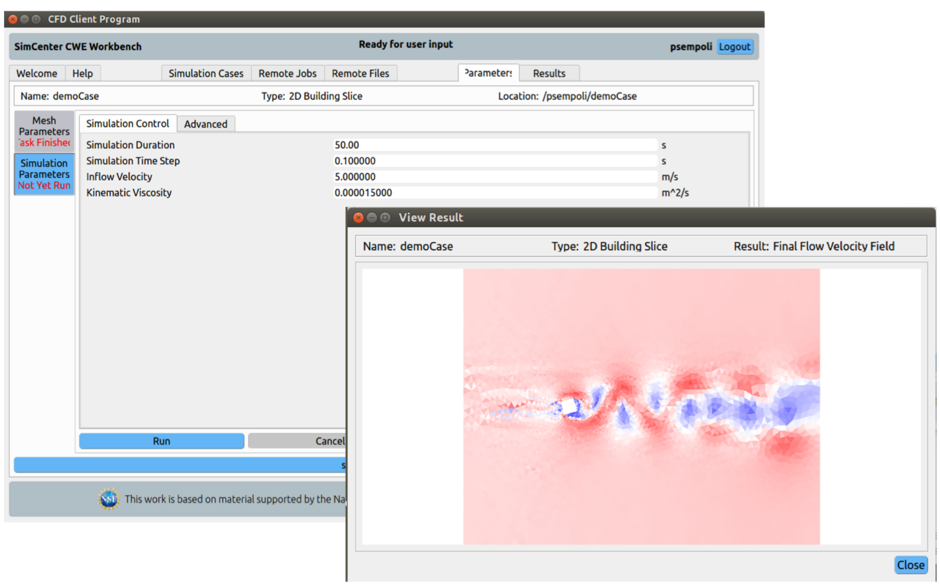
Can upload geometries created in FreeCAD



Performs Mesh Generation



. . . and Simulation

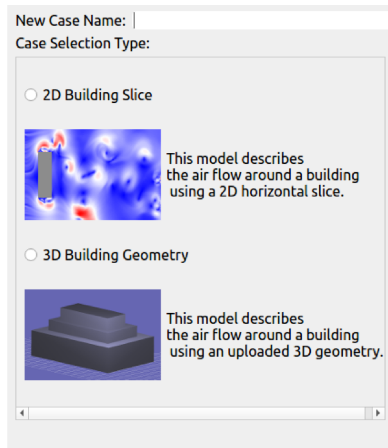


Extensible Template Framework

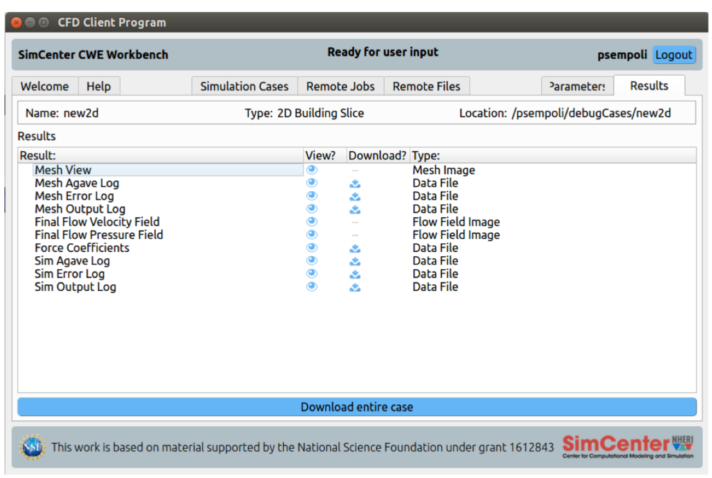
Current templates for 2D and 3D rigid body simulations.

Planned future templates for:

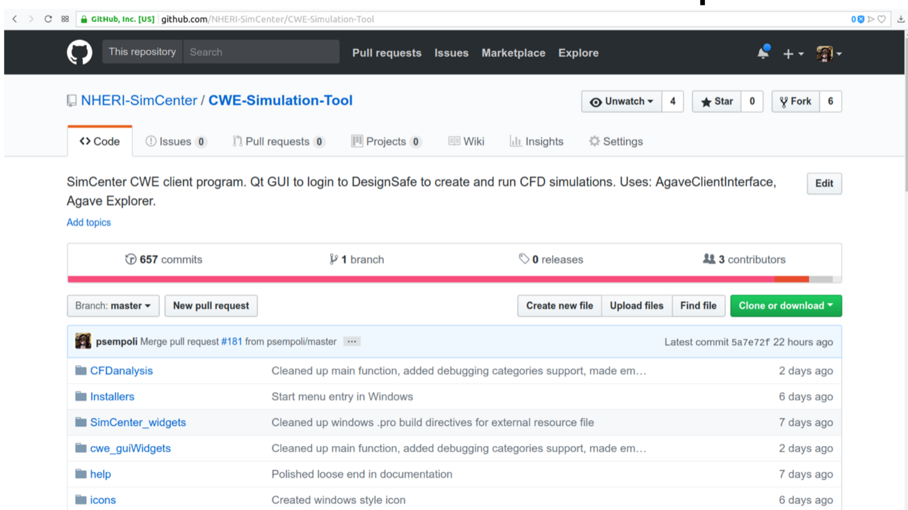
- Custom inflows
- UQ
- Vibrating Structures
- Other user interests?



Extensible Result Framework



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

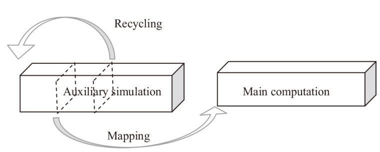
$$\frac{\partial u}{\partial t} + u \cdot \nabla u = \upsilon \nabla^2 u - \nabla p$$

- Continuity equation

$$\nabla \cdot u = 0$$

Method types

Recycling Method



Recycling technique (Lund et al., 1998)

Computational Costly
Mesh dependent

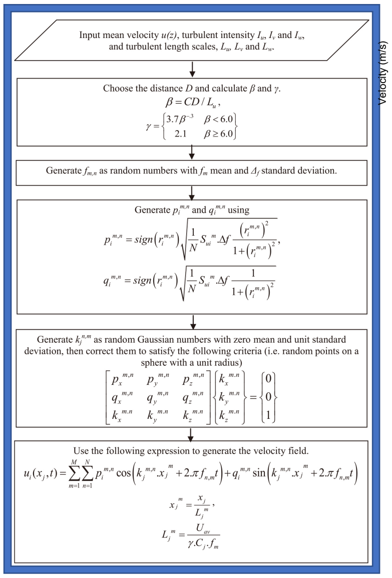
Synthetic Turbulence

Weighted Amplitude Wave Superposition (WAWS)
Hoshiya, 1972
Non-divergence-free

Random Flow Generation (RFG)
Kraichnan, 1970
divergence-free with Gaussian spectra
not compatible with the spectra in the ABL

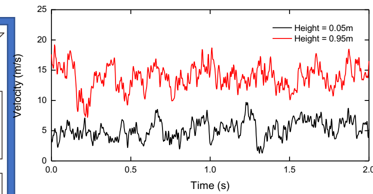
Discrete RFG (DRFG)
Huang et al., 2010
divergence-free with ABL turbulent spectra

Consistent DRFG (CDRFG)
Aboshosha et al., 2015
An improvement to DRFG

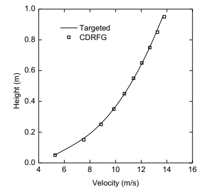


CDRFG technique flow chart (H. Aboshosha et al., 2015)

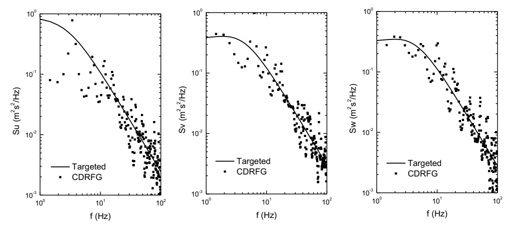
Inserting the synthetic velocity field into the Navier-Stokes equation to compute the pressure field



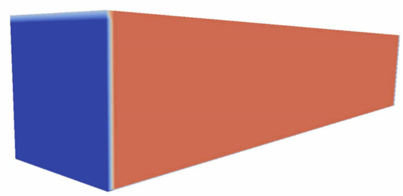
velocity sample



mean velocity profile



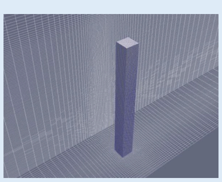
spectral plots



Numerical Simulation Example

FSI in OpenFOAM: Partitioned strategy

Fluid domain Ω_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

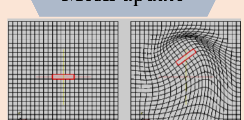
Interpolation

(patchToPatchInterpolation)

Displacement, Velocity

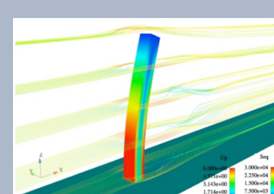
Interpolation

Mesh update

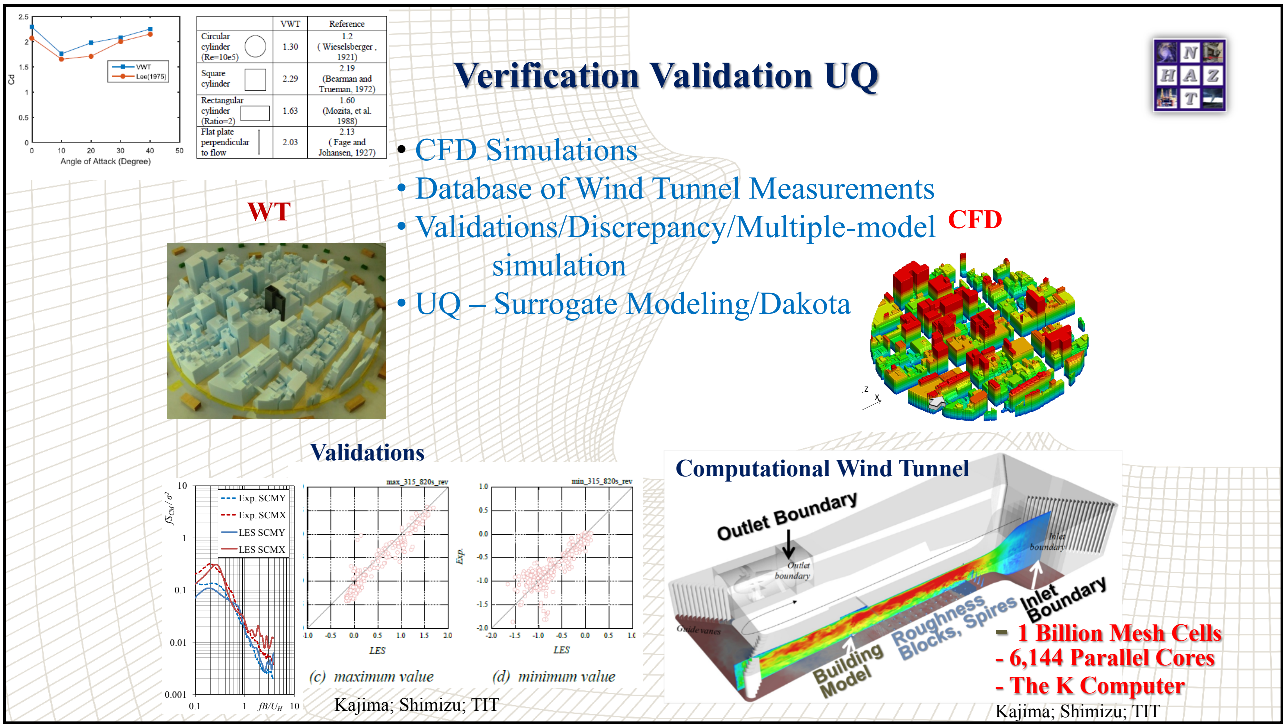


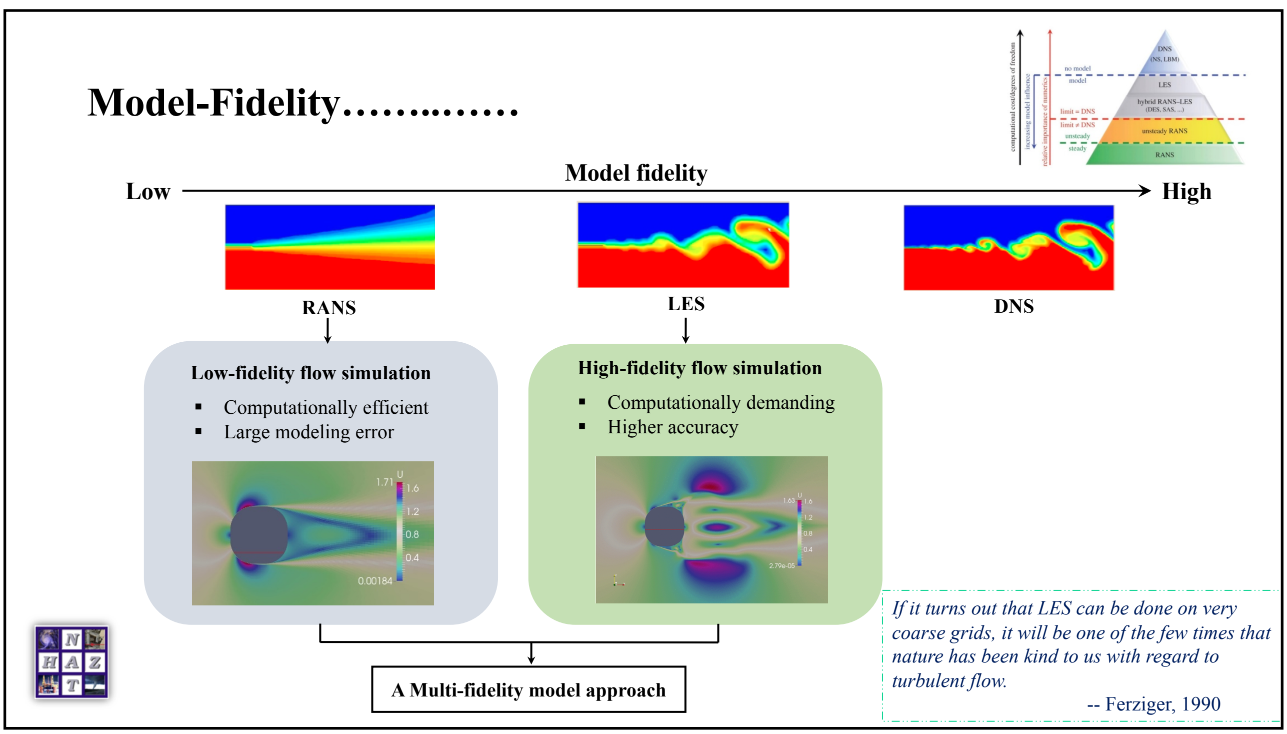
(dynamicMotionSolverFvMesh)

Solid domain Ω_S
(Structural mechanics computation)



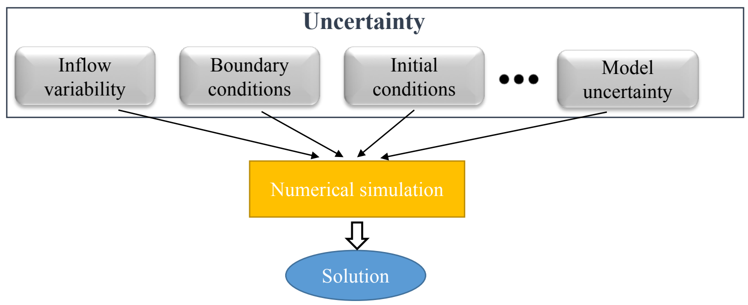
Mesh motion solver: generic motion solvers in OpenFOAM (Spring analogy, Laplace equation, linear pseudo-solid equation), user-defined solvers


Verification Validation UQ
• CFD Simulations
• Database of Wind Tunnel Measurements
• Validations/Discrepancy/Multiple-model simulation
• UQ – Surrogate Modeling/Dakota
WT
CFD
Angle of Attack (Degree)
VWT
Lee(1975)
VWT
Reference
Circular cylinder (Re=10e5)
1.30
1.2 (Wieselsberger , 1921)
Square cylinder
2.29
2.19 (Bearman and Trueman, 1972)
Rectangular cylinder (Ratio=2)
1.63
1.60 (Mozita, et al. 1988)
Flat plate perpendicular to flow
2.03
2.13 (Fage and Johansen, 1927)
Validations
Exp. SCMY
Exp. SCMX
LES SCMY
LES SCMX
(c) maximum value
(d) minimum value
LES
Exp.
Kajima; Shimizu; TIT
Computational Wind Tunnel
Outlet Boundary
Inlet Boundary
Roughness Blocks, Spires
Building Model
- 1 Billion Mesh Cells
- 6,144 Parallel Cores
- The K Computer
Kajima; Shimizu; TIT


Model-Fidelity……..……
Model fidelity
Low
High
RANS
LES
DNS
Low-fidelity flow simulation
▪ Computationally efficient
▪ Large modeling error
High-fidelity flow simulation
▪ Computationally demanding
▪ Higher accuracy
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



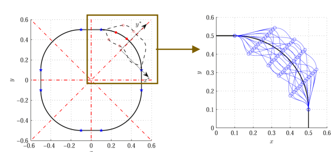
Stochastic emulation

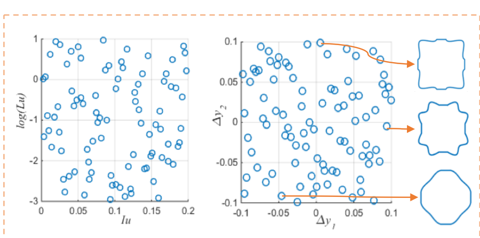
- **Design of experiments**
  - Design variables: Inflow parameters I_u, L_u + Shape parameters Δy_1^*, Δy_2^*
  - Method: Latin Hypercube Sampling (89 samples)
- **Computational simulation**
  - URANS: k-ω SST model with wall functions

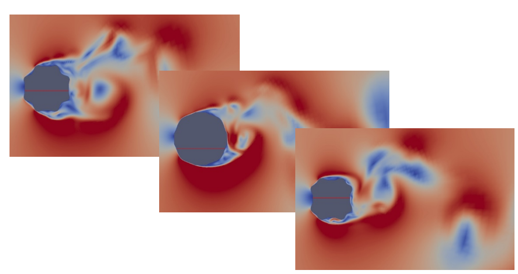
$$k = \frac{3}{2}(UI_u)^2$$

$$\omega = C_\mu^{-1/4}\frac{\sqrt{k}}{L_u}$$

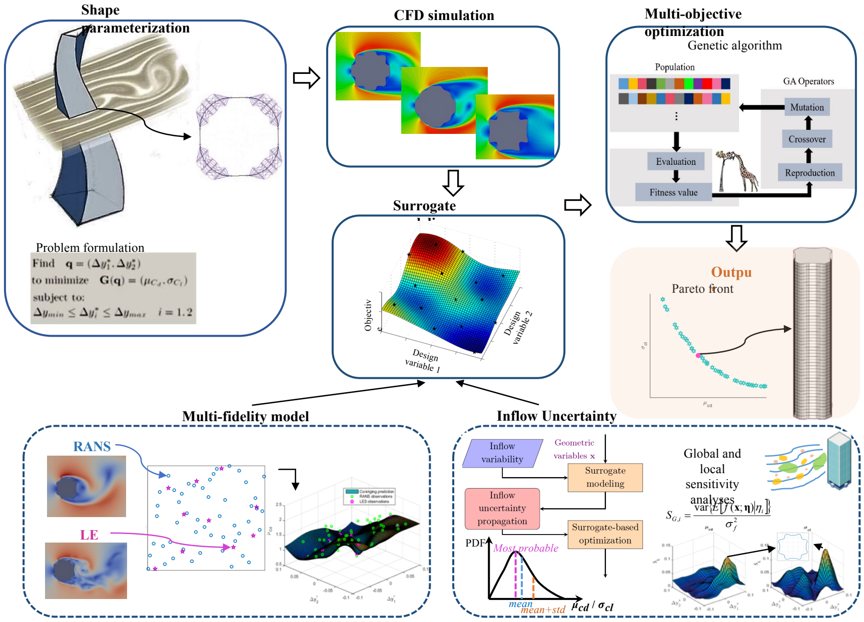
- **Surrogate calibration**
  - Method: Ordinary kriging







Turbulence intensity I_u *Integral length scale L_u*



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 validations

NASA Crowdsourcing Efforts To Speed Supercomputer Program.
The New York Times (5/8, Fortin, Subscription Publication) reported that NASA is crowdsourcing its efforts to improve the speed of its Pleiades supercomputer's FUN3D program, which is written in the decades-old computer language Fortran. The agency is offering cash prizes totaling up to $55,000. NASA executive Michael Hetle said, "This is the first time we've had a challenge like this." He added, "And so far, we have quite a bit of interest."

Mozak: a game that crowdsources mapping of brain-cells

NSF NHERI