Coupling micro-scale CFD simulations to meso-scale models
Outline

- Introduction
- O.F.Wind® micrositing software
- Mesoscale and CFD Models
- wrf2Foam utility
- Coupling strategy
- Test case results
- Conclusions and outlook
O.F.Wind is an OpenFOAM® based CFD tool for site assessment. It has a dedicated GUI and accounts for automising the process for using the different tools of the OpenFOAM® toolbox. Its key features are:

- Reliable and robust: extremely rare divergence problems
- Handles really complex terrains, unstructured mesh
- Very flexible meshing with local refinement: accuracy where needed
- Whole CFD part of O.F.Wind (= the modified OpenFOAM®) is available open source
- Forest and wake modelling available, thermal sover under development
- Fully parallelized
- Proven accuracy
- Links to other programs: WindPro (.map and time series), wrg, Surfer
- Comprehensive post-processing:
  - Real 3D analysis of results boosts understanding of flow behaviour
  - Access to any data point in flow field
  - Calculation and analysis of Speedups, TI, IEC parameters, time series etc.
MesoScale – MicroScale models

Mesoscale simulations (NWP):

- Solve velocity, pressure, density and temperature, … (compressible)
- Pressure coordinate system (geopotential height)
- Variables are both staggered and collocated
- Models soil influence, vegetation growth and solar heat flux

- Usually weak mass conservation
- Coarse grid resolution (a few km to 500m)
- Ignore local effects (hills)

Microscale simulations (CFD):

- Solve velocity and pressure, sometimes temperature (incompressible)
- Cartesian coordinate system
- Variables are usually collocated (Rhie-Chow)
- Mass-consistent
- Have a high grid resolution (a few meters)
- Detailed ground roughness models and forest

- Computationally expensive
- Ignore a lot of atmospheric phenomena
- Usually use idealized wind profiles
Why coupling?

We can combine advantages of both models

- Use advanced physical models
  - Forest modelling
  - Roughness
  - Wakes

- Provide more realistic inlet profiles
  - No need for NABL profiles
  - Thermo-stratification can be included

- Data clustering:
  - Extract statistically relevant days
  - Enhanced resource assessment

- Accurate steady-state / transient simulations
  - Investigate extreme events (blade damage)
  - Simulate diurnal-cycle
wrf2Foam

- **Objective:**
  - Convert WRF outputs into readable OpenFOAM® cases
  - WRF (Weather Research Forecast) is a widely used NWP software

- **Main difficulties:**
  - Read WRF outputs and convert them to OpenFOAM® format
  - Map the fields on the cartesian grid
  - Merge several WRF outputs (several time folders)

- **Method:**
  - Create a blockMesh mesh in OpenFOAM®
  - Move the mesh points to exact xyz-coordinates
  - Unstagger WRF's variable on the collocated grid

NetCDF files → Format conversion i-j-k blockMesh → spatial transformation x-y-z blockMesh
wrf2Foam

- **Example: Tracking Hurricane “Katrina”**
  - Outputs spread over multiple files
  - Multiple time-steps per files
  - Moving computational domain
  - A different OpenFOAM® mesh at each time-step
Coupling strategy – Meshes

- **WRF simulation (by Vortex®)**
  - Complex terrain
  - 83x83x10km (3km resolution, 37 vertical levels)
  - Approx 800 cells in CFD domain

- **O.F.Wind® models**
  - Generated using a high resolution SRTM map
  - 25km radius (<70m resolution)
  - Constant roughness length ($z_0 = 0.03m$)

---

Coarse CFD mesh
1.6M cells

Fine CFD mesh
6.5M cells
Coupling strategy – Orography blending

- **Orography blending:**
  - Big difference in terrain altitudes (450m)
  - Buffer region (4km) – Cosine blending
  - Outer CFD elevations matches WRF elevations

- **Data mapping:**
  - CFD profiles are translated along the absolute coordinates of the microscale grid
Special boundary conditions:

- At each time-step, face values are read from WRF twin simulation (linear time-interpolation)
- Based on velocity flux, switches from Neumann to Dirichlet
- Turbulence profiles based on WRF boundary layer height, decreasing shear stress
NWP provides accurate representation of the fields away from the ground. CFD models usually ignore phenomena such as the Ekmann effect or low level jets.

In upper region of the CFD domain, we use the results of the WRF simulation.

Data assimilation (space and time)

- Body force (Newtonian relaxation)

\[
\rho \frac{\partial u}{\partial t} + \rho u \cdot \Delta u + \Delta p = \Delta \cdot \tau + \rho b
\]

\[
b(\vec{x}, t) = G \cdot R(\vec{x}, t) \cdot (u_{\text{wrf}} - u)
\]
Coupling strategy — Data Assimilation

WRF

$G = \frac{1}{50}$

$G = \frac{1}{500}$

$G = 0$
Results: Steady-state – Case A

- CFD model reproduces qualitatively well the field obtained from WRF simulation
- Mesh resolution has a small effect

Uwrf interpolated on 1.6M cfd mesh

Hill has no influence

Horizontal velocity (m/s) - Uwrf

Ucfd – 1.6M mesh

Ground effects

Ucfd – 6.5M mesh
Results: Steady-state – Case B

- CFD model exhibit much higher speeds than the WRF solution (relief has no influence)
- Not an artefact of the nudging
Conclusion and future tasks

What has been achieved?

- Coupled mesoscale data with a micro-scale simulation
  - Transient and steady-state coupling
- Added newtonian relaxation in OpenFOAM®
- Results look promising
- Fully automated procedure

Still to be done:

- Compare against field measurements
- Include thermal effects, forest models and wakes
- Use data-clustering etc. for improving efficiency