Computers & Fluids 115 (2015) 75–85

Contents lists available at ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

Nesting an incompressible-flow code within a compressible-flow code: A two-dimensional study

Michael A. Sprague [a,*], Ignas Satkauskas [a,b]

[a] Computational Science Center, National Renewable Energy Laboratory, 15013 Denver West Parkway, Golden, CO 80401, USA
[b] Department of Applied Mathematics, University of Colorado, 526 UCB, Boulder, CO 80309, USA

ARTICLE INFO

Article history:
Received 24 February 2014
Received in revised form 15 January 2015
Accepted 9 March 2015
Available online 19 March 2015

Keywords:
Wind energy
Multi-scale
Model coupling
Computational fluid dynamics
Numerical weather prediction

ABSTRACT

We consider numerical algorithms appropriate for one- and two-way coupling between meso-scale and micro-scale fluid-dynamics codes for wind energy computing. At the meso-scale is a numerical weather-prediction code, which is typically based on the compressible-flow Euler equations. At the micro-scale, surrounding one or more wind turbines, is a computational fluid dynamics code, which is typically based on the incompressible-flow Navier–Stokes equations. When calculating short-duration flow around wind turbines, one-way coupling is sufficient, where the meso-scale computational model drives the micro-scale model. However, in long-duration simulations involving large wind farms, the influence of the wind farm on the meso-scale weather may no longer be insignificant and two-way coupling is warranted. In this study, we focus on a simple two-dimensional system, for which our goal is to devise one- and two-way coupling algorithms that can effectively transport a vortex propagating in laminar flow from one domain to the other. Two coupling schemes and their numerical implementation are described: partial-boundary coupling and projection coupling. In the former, the micro-scale-domain boundary is decomposed, based on the meso-scale solution, into sections corresponding to inflow and outflow. The micro-scale model has Dirichlet- and Neumann-type boundary conditions on these sections, respectively. In projection coupling, the meso-scale solution is projected onto the incompressible-flow solution space in the micro-scale domain, from which Dirichlet-type boundary conditions are derived. In these simulations, the uncoupled meso-scale solution is taken as the reference, and the best coupling method is that which produces solutions that deviate the least from the reference. In one-way coupling, under a simple two-dimensional laminar-flow test case, partial-boundary coupling was more effective than projection coupling. However, in two-way coupling, projection coupling was the best performer.

1. Introduction

Computational models present opportunities for reducing the cost of wind energy by enabling scientists and engineers to better predict the associated complicated multi-physics and to better optimize turbine designs and wind farm layouts. While it is clear that the show-stopping engineering problems surrounding a single wind turbine have largely been solved (after all, wind turbines are being built and installed), large-scale deployment of wind farms, composed of hundreds of wake-interacting wind turbines, faces daunting challenges due to poorly understood flow physics. For example [1], observed power loss can be 20–30% of that predicted in operational wind farms due to poorly understood turbine-wake interactions in large wind farm arrays [2,3], and turbines within wind farms experience significantly higher failure rates compared to isolated turbines due to uncertain aerodynamic loading [4]. Further, it is unclear in what ways large wind farms will affect regional weather, which is an important issue considering that wind farms are often centered in agricultural regions. Understanding and overcoming these challenges will be aided by high-fidelity computational models that account for interaction with the local weather system.

The physical systems governing production and extraction of wind energy have relevant time and length scales spanning many orders of magnitude [5]. Physically validated mathematical models and software are established at each *uncoupled* scale; however, no single model can treat all relevant scales. For example, meso-scale atmospheric dynamics are described well by solutions to the compressible Euler equations (with appropriate physics models), whereas turbine-vicinity (micro-scale) fluid dynamics are well described by solutions to the incompressible Navier–Stokes (NS)

* Corresponding author.
E-mail address: Michael.A.Sprague@nrel.gov (M.A. Sprague).

http://dx.doi.org/10.1016/j.compfluid.2015.03.005
0045-7930/© 2015 Published by Elsevier Ltd.

equations (with an appropriate subgrid-turbulence model). While the need for multi-model multi-scale coupled systems is clear, there is no obvious path for efficient and accurate coupling for holistic simulation across relevant scales.

In this paper, we are focused on numerical methods appropriate for coupling numerical weather prediction (NWP) codes (meso-scale) with turbine-local computational fluid dynamics (CFD) codes (micro-scale). Further, we are interested in codes for micro-scale flow that have unstructured-grid capability allowing accurate simulation over complex topography. We focus on the Weather Research and Forecasting (WRF) code [6] as our target meso-scale code and an incompressible-flow CFD code like OpenFOAM [7] as our target micro-scale code. Coupling such codes has an abundance of challenges. First, as mentioned above, there is an inherent mismatch of the mathematical models. The WRF model is, at its core, built on the compressible inviscid Euler equations. At the micro-scale, the turbine-vicinity model is built on the incompressible viscous Navier–Stokes equations. Thus, acoustic waves exist in the meso-scale model, while the micro-scale model is acoustically rigid. Second, there are numerical-model mismatches between WRF and, say, an OpenFOAM implementation: (i) the spatial and temporal grids will not match at their interface, and WRF is discretized in the vertical direction in pressure coordinates – the spatial grid moves vertically with variation in pressure; (ii) WRF employs an explicit time integrator, whereas OpenFOAM employs a semi-implicit time integrator; and (iii) WRF is spatially discretized with finite differences on a structured grid, whereas OpenFOAM is discretized with finite volumes on, in general, an unstructured grid.

While there have been limited efforts to two-way couple WRF and an incompressible-flow CFD code (see, e.g., [8]), there is no evidence of successful interactive coupling. Alternatively, there has been more success in one-way coupling, where a meso-scale NWP model is used to drive an incompressible-flow CFD model. Li et al. [9] used the RAMS NWP model to drive a FLUENT incompressible-flow CFD simulation in a Reynolds averaged Navier–Stokes (RANS) framework. WRF-calculated velocity and temperature fields were used on all lateral and top boundaries in the FLUENT model. Boutanios et al. [10] performed OpenFOAM RANS simulations where the boundary conditions were taken from a WRF simulation. Zajaczkowski et al. [11] performed a preliminary study where WRF results were used as boundary conditions for an AcuSolve [12] CFD simulation, again with a RANS model. WRF velocities were used as inflow boundary conditions on two sides of the CFD box; other boundaries had simple outflow boundary conditions. A Newtonian-relaxation data assimilation technique was used in transferring WRF data to AcuSolve. Recently, Churchfield et al. [13] used WRF and WRF-LES (large-eddy simulation), to drive an OpenFOAM incompressible-flow CFD simulation with a standard Smagorinsky LES model in their Simulator fOr Wind Farm Applications (SOWFA). WRF data were used as OpenFOAM boundary conditions, where the mean-flow direction was into the OpenFOAM domain; outflow boundary conditions were used elsewhere. Gopalan et al. [16] and Sitaraman et al. [14] used WRF data to one-way drive a wind farm compressible-flow CFD simulation in the HELIOS computational platform [15] (with RANS and detached-eddy-simulation techniques) where the full rotors were resolved. WRF-HELIOS coupling was accomplished via the meso-scale/micro-scale interface (MMCI [16]), which is a python-based infrastructure. Yang et al. [17] used WRF at the meso-scale to drive their Virtual Wind Simulator, which is a LES framework for simulating incompressible turbulent flow around wind turbines in complex terrain. Further discussion of coupling NWP meso-scale models and CFD micro-scale models for wind-engineering applications can be found in Yamada and Koike [18]. Peet and Lele [19] examined methods for coupling a compressible-flow code with either a low-Mach-number-flow or incompressible-flow code. In Pete and Lele's preferred approach, domains interacted at their boundaries; variables were transferred between codes through a simple interpolation. In their analysis of coupling between compressible- and incompressible-flow-models, they restricted simulations to where the underlying test flow was incompressible. Peet and Lele [20] applied their approach to two-dimensional LES of film cooling where a low-Mach-number-flow code was coupled to a compressible-flow code.

There have been successful efforts to one- and two-way couple a nested-grid large-eddy-simulation inside a meso-scale simulation grid, where the same underlying mathematical model is used at both scales (e.g., compressible flow at both scales). Sullivan et al. [21] examined LES of planetary-boundary-layer (PBL) incompressible flows where a refined grid resided in a coarse grid. One- and two-way coupling was examined. Moeng et al. [22] examined two-way grid nesting for LES PBL compressible flows in the WRF model. Recently, Mirocha et al. [23,24] showed that the use of one-way-coupled nested grids in WRF LES simulations can improve accuracy. Harris and Durran [25] used an idealized one-dimensional model to study one- and two-way grid nesting.

In this paper, we focus on a simple two-dimensional system that captures large-scale flow characteristics of the meso- and micro-scale systems. The two-dimensional compressible inviscid Euler equations (with no additional physics models) constitute our meso-scale model and the two-dimensional incompressible Navier–Stokes equations (with no turbulence modeling) constitute our micro-scale model. Numerical solution of these models mimics that of WRF and OpenFOAM, i.e., finite-differences spatial discretization and explicit time integration for the meso-scale code and finite-volume spatial discretization and semi-implicit time integration for the micro-scale code. With these codes, we test two coupling schemes in one- and two-way coupling. This work is meant to be a foundation-creating step towards the development of robust, accurate, and efficient coupling algorithms appropriate for production computing. Toward that end we examine our coupling schemes with a simple propagating vortex and quantify the errors introduced due to model coupling and the discrepancies between the underlying models. With an effective coupling strategy that can propagate simple flows between models with high fidelity in a physically meaningful way, we can then move towards solving the challenging issues associated with, e.g., the transfer of sub-grid turbulent energy and turbulent-flow structures.

The paper is organized as follows. In Section 2 we describe our two-dimensional model system and our coupling algorithms. In Section 3 we describe our numerical models. In Section 4 we present numerical results for one- and two-way coupling for the propagation of a laminar vortex. Section 5 is our conclusion.

2. Formulation and implementation

2.1. Test system

Fig. 1 shows our idealized two-dimensional test system, which consists of two domains: (*i*) an inner square domain, denoted $\Omega_{\rm NS}$, with boundary denoted $\partial\Omega_{\rm NS}$, where we are interested in solutions to the NS equations, and (*ii*) a larger, partially coincident square domain, denoted $\Omega_{\rm E}$ ($\Omega_{\rm NS} \subset \Omega_{\rm E}$) with external boundary denoted $\partial\Omega_{\rm E}$, where we are interested in solutions to the Euler equations. Position (nondimensionalized with a reference length ℓ^*) is denoted $\mathbf{x} = (x, y) \in \Omega_{\rm E}$, and the coordinate-system origin is at the center of the aligned square domains $\Omega_{\rm NS}$ and $\Omega_{\rm E}$.