Nearfield Plume Modeling using Large Eddy Simulation

Thorsten Stoesser, Dongjin Kim, Mehtap Cakmak
Near Field Simulation Team

Thorsten Stoesser  
Dongjin Kim  
Mehtap Cakmak
Contents

• Near Field Model Domain
• Numerical Model
• First Test Case: Jet In Cross Flow (JICF)
• Outlook to year 2
Near Field Model Domain

- Turbulence
- Entrainment/Detrainment
- 3-Phase Flow

Laboratory Experiments
Separate laboratory experiments address droplet and bubble scale processes, including bubble and droplet formation with and without dispersant application, dissolution, droplet-turbulence interaction, and evaporation with associated air quality assessment.
Numerical Model

• Unsteady, incompressible Navier-Stokes solver
• Finite difference method on a Cartesian grid with staggered variable arrangement
• Local mesh refinement technique
• Multigrid/Multiblock method
• Wall-Adapting Local Eddy Viscosity (WALE) subgrid scale model
• Immersed Boundary Method (IBM)
• Lagrangian/Eulerian multiphase capabilities
Local Mesh Refinement

• Purpose is to decrease the computational effort by placing more grid points around critical regions to get higher resolution and fewer grids points to regions where a coarse grid is adequate.

• Block-based spatial refinement

• No temporal refinement

• Efficient and easy combination with multigrid method

• Use of local corrections after computations, implementation of approximate projection operator and higher order spatial interpolations to prevent the discontinuity problem across the fine-coarse interface.
Local Mesh Refinement

- local mesh refinement between blocks
- here only 2:1
- currently overlap region of 1 coarse cell to 4 fine cells
- values in the ghost cell are interpolated

Ghost cell for fine grid block

Ghost cell for coarse grid block
Multigrid Method

- An efficient method for solving the Poisson equation, especially for three-dimensional problems

Basic Algorithm of Multigrid Method for linear systems:
1. Compute residuals of Poisson Equation on the finest level iteratively. High frequency errors disappear in this step.
2. Restriction of residuals from fine to coarse levels and then iterations are performed on this new level in order to eliminate low frequency errors.
3. Interpolation of errors from coarse to fine levels
4. Correction and final iterations on the finest level
5. Locally refined meshes have one more level
Multigrid Method – Efficiency testing

- Gauss-Seidel or TDMA iterative methods are tested. V-cycle is preferred to decrease the number of level changes. The improvement in the convergence rate can be seen in the figure when compared with SIP solver.

- Multigrid method is approximately 3-4 times faster than SIP
• In the filtered Navier-Stokes equations, large scale eddies are simulated directly and small scale eddies are modeled. In our code, WALE Model is used due to numerous advantageous.
  – Simpler than Dynamic SGS Model
  – No need damping function like Smagorinsky Model
  – Automatically accounts for wall effects
  – Easily applied to complex geometry problems
  – Can simulate flows with transition or re-laminarization

• Preliminary tests have been successful
In order to validate the model, fully developed low Reynolds number turbulent flow through a straight square duct is tested. Wall shear stress distribution agrees well with DNS data. Outperforms Smagorinsky model on the same grid.
Multiphase Flow Treatment

- 2-Way Coupled Particle Tracking Method → NSF Rapid Grant – Bubble Column → Method described in Hu and Celik, 2008
Year 1 Program and Goals

• Develop Methodology – test algorithms and models
  ⇒ Months 1 - 6

• Validate Model for 2 basic test cases using available experimental data
  - Jet in Cross Flow (Experiment¹, LES, DNS²,³)  ⇒ Months 7 - 9
  - Bubble plume experiments at Texas A&M  ⇒ Months 10 - 12

**Testcase #1 ➔ Jet in Cross Flow**

- Computational domain $L_x=13.7D$, $L_y=8D$, $L_z=12D$
- Circular Pipe of length $l_z=6D$
- Inflow condition for jet by simultaneously computing a pipe flow with cyclic boundary conditions
- Velocity ratio $U_{jet}/U=3.3$
- Crossflow at upstream boundary assumed laminar featuring a boundary layer with a thickness of $\delta_{99}=0.5D$ using a Blasius profile.
- $Re_{D,U_\infty} = 2100$ yielding and $Re_{D,U_{jet}} = 6930$
- Well documented testcase – accurate laboratory data
- Successful LES for comparison
Testcase #1  ➔ Jet in Cross Flow

LES\(^2\) compared with experimental velocity data

Streamlines, Streaklines and Vortex Lines
Outlook to year 2

- Very preliminary multi-phase flow predictions (left) and comparisons with experimental observations (right)

- Refine NSF Rapid grant model using current multigrid/multiblock model
THANKS