Large Eddy Simulation for Atmospheric Boundary Layer (ABL)

Ashvinkumar Chaudhari

Centre of Computational Engineering and Integrated Design (CEID)
Lappeenranta University of Technology (LUT)
P.O.Box 20, FI-53851 Lappeenranta, Finland
Introduction

- The development of renewable energies, particularly energy from wind, water, and solar power, is a wide public interest nowadays.

  - Renewable energy has an important role to play in reducing Carbon Dioxide (CO2) emissions in environment.
  - Increasing the share of renewable energy in the energy balance system enhances sustainability.
  - It also helps to reduce the other energy sources, such as oil and natural gas.

- Numerical simulation of wind flow in Atmospheric Boundary Layer (ABL) is a great interest for engineers to help in the modeling of wind energy issues.
Atmospheric Boundary Layer (ABL)

- What is ABL?
  - Lowest region of the atmosphere, directly affected by the Earth’s surface

- The ABL plays an important role in many fields, including air pollution and the dispersal of pollutants, aeronautical, meteorology, weather forecasting, and climate studies, etc.

- Very high-$R_e$ ($\sim 10^7$) turbulent flow, over rough wall, strongly affected by buoyancy forces
CFD in environmental flows

- Modeling of a wind flow in environment is of great interest in terms of wind energy applications, such as it helps us to locate, control, and optimize the wind farms, as well as power stations, airports, etc.

- Several theoretical, experimental and numerical studies have been reported about modelling of a wind flow in complex terrains contain hills, forest, lake, etc.

- Numerical calculation of a wind flow using computational fluid dynamics (CFD) has been becoming a popular technique.

- It can provide significant cost benefits for optimizing engineering design solutions related to environmental concerns.
Turbulence models

- Most of the research currently taking place in the field of CFD concerns the study of turbulent flows. Almost any naturally occurring flow is turbulent, and hence it is important to be able to model turbulent flows accurately.

- Different approaches to make turbulence computationally tractable:
  - Reynolds-Averaging Navier-Stokes Equations (RANS)
    - Gives a prediction of the mean velocity and the mean level of turbulent quantities
  - Direct Numerical Simulations (DNS)
    - It captures all of the relevant scales of the turbulent motion. But this approach is extremely expensive,
    - Computational cost requires for DNS is proportional to $R_e^3$ (Turbulent Reynolds number)
  - Large Eddy Simulations (LES)
  - Detached Eddy Simulations (DES)
Large Eddy Simulation (LES)

- Large Eddy Simulation (LES) is a model to simulate the turbulent flows in CFD where the smaller eddies are filtered and are modeled using a sub-grid scale (SGS) model, while the larger energy carrying eddies are simulated.

- It was initially proposed in 1963 by Joseph Smagorinsky to simulate atmospheric air currents.

- Why LES ??
- Some applications need explicit computation of accurate unsteady fields.
  - Bluff body aerodynamics, where the flow is governed by large turbulent scales
  - Aerodynamically generated noise (sound)
  - Atmospheric boundary layer (ABL)
  - Mixing
  - Combustion
- Examples:

Figure courtesy of Ansys Fluent
Unsteady flow

- Turbulent flow over a complex terrain contains complex flow characteristics such as, separation, reattachment, aerodynamic instabilities, etc. Therefore, an advance approach should be applied.
- Due to inherent unsteady phenomena of this flow, it is difficult to model by RANS approach. Thus, time dependent computations such as DNS or LES are required.
- Relative to RANS, LES can be computationally expensive, requiring about 1000 times greater computational resources, however, it yields fidelity solutions for flow configurations where RANS fails.

Figure courtesy of Ansys Fluent
Test case:
LES for Channel flow
(Re\_tau=180)

- Information on Geometry and Grid resolution in LES
  - $L_x = 2\pi\delta$
  - $L_y = 2\delta$
  - $L_z = \pi\delta$, where $\delta$ is the boundary layer depth (=1).
- Mesh resolution:
  - Coarse: $(60 \times 61 \times 60) = 219,600$
  - Fine: $(100 \times 101 \times 100) = 10,100,000$
- Flow time=21.78 s, time step= 0.0015 sec

Moser et al. (1999) have performed the DNS of fully developed channel flow at Re\_tau=180, 395, 590.
Domain: $4\pi\delta \times 2\delta \times (4/3)\pi\delta$
Mesh: $(128 \times 129 \times 128) = 2113536$
Stream-wise velocity

u- instantaneous

u- mean

u- RMS

08 Dec 2011
Mean Velocity profile in wall-unit
(Re\_tau=180)

- In fine mesh case, LES has a better agreement in mean velocity with DNS.
RMS velocity and Reynolds stress

- Values are normalized with $u_t$.
- Only resolved Reynolds stress
- Reasonably good agreements with DNS again.
LES for air flow over 2D-hill

- Standard hill geometry in wind tunnel experiments carried out by Khurshudyan et al. (1981).

- Height of Hill (H) = 0.117 m
- Length of Hill = a = 3H
- The shape of hill given by following parametrical formula:

\[
x = \frac{1}{2} \xi \left[ 1 + \frac{a^2}{\xi^2 + m^2(a^2 - \xi^2)} \right]
\]

\[
z = \frac{1}{2} m \sqrt{a^2 - \xi^2} \left[ 1 - \frac{a^2}{\xi^2 + m^2(a^2 - \xi^2)} \right], \quad |\xi| \leq a
\]

where \( m = n + \sqrt{n^2 + 1} \) and \( n = \frac{H}{a} \) is average slope.
CFD model and Grid resolutions

Full Domain

<table>
<thead>
<tr>
<th></th>
<th>Streamwise (X)</th>
<th>Vertical (Y)</th>
<th>Spanwise (Z)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Domain length</td>
<td>45.64 H (3hx + 20H)</td>
<td>10 H</td>
<td>6H</td>
</tr>
<tr>
<td>Total nodes</td>
<td>300</td>
<td>141</td>
<td>46</td>
</tr>
<tr>
<td>Mesh Resolution</td>
<td>0.153H</td>
<td>0.0043H-0.251H</td>
<td>0.133H</td>
</tr>
</tbody>
</table>

Total number of hexahedral nodes=1945800
Boundary conditions

- **Inflow condition**: Velocity given by logarithmic flow profile,

\[
U = \begin{cases} 
\frac{U_{inf}}{k} \ln \left( \frac{z}{z_0} \right), & z \leq h \\
U_{inf}, & z > h 
\end{cases}
\]

\[V = 0, \quad W = 0.\]

- Where, \(z\) = The height above the ground, \(z_0\) = Roughness (1.57E-04),
- \(K\) = Von Karman constant (=0.4), \(u^*\) = friction velocity (=0.178 m/s),
- \(U_{inf}\) = Free stream velocity (=4 m/s), \(h\) = Boundary layer depth (=1.17 m)

- **Inflow turbulence**: Turbulent intensity 12 % (was given to generate the velocity fluctuations at inflow)
- **Outlet** = Outflow
- **Span-wise direction** = Periodic boundary condition
- **Upper boundary** = Symmetry
- **Lower boundary** = No-slip (Smooth Wall)
Numerical method and flow statistics

- FVM based commercial code ANSYS Fluent 13.0 with the Smagorinsky-Lilly sub-grid-scale model has been used.

- Reynolds number based on hill height and free stream velocity, \( \text{Re}_H = 3040 \).
- Frictional Reynolds number (\( \text{Re}_{\tau} \)) = 1187

- Pressure-velocity coupling scheme: PISO
- Second order implicit solver
- Discretization scheme:
  - Pressure: standard
  - Momentum: Bounded central differencing
- Total averaging time = 37.5 sec (i.e. more than 10 x \( L_x \))

- Results are time and space averaged in span-wise direction.
Mean velocity and turbulence

Contours of mean u-velocity

RMS u-velocity

UV Reynolds stress
LES-animation

- Instantaneous stream-wise velocity (mean + fluctuation)
- This shows the flow separation and reattachment behind hill
Comparison with measurement
[Khurshudyan et al. (1981)]

- Mean velocity profile at several locations before and after hill.
- Results are normalized by free stream velocity, $u_{\text{inf}} = 4.1$ (m/s)

- Even with the different Reynolds number ($Re_H$), the overall mean velocity profile has a quantitatively good agreement at several locations.
Reynolds stress \((-uv/u_{\text{inf}}^2)\)
Reynolds stress \((-uv/u_{inf}^2)\)

- Not so good agreement in Reynolds shear stress \((-UV)\) near wall region, after the hill
Conclusions and future work

- Flow seems to be completely unsteady after the hill, as expected highly turbulent flow occurs with a large recirculation bubble in wake region.

- Flow separation occurs after it passes the summit, a reverse flow can be seen behind the hill and finally it reattach at X/H=5.5. The separation region in LES is found to be a bit smaller than measurement (X/H=6.5).

- Even with the completely different Reynolds number (Re_H), overall mean velocity profile has a quantitatively good agreement with the measured data at several locations.

- Still poor agreement with measurements for turbulence quantities near wall region, which need to be investigated.
References

- ANSYS FLUENT 12.0 Theory Guide, April 2009
- ANSYS FLUENT 12.0 User’s Guide, April 2009
- CFD-Wiki: www.cfd-online.com/Wiki/
References


- Thank you for your attentions!!!

- Comments are appreciated!!!