Development and validation of Computational Wind Field Model(Wind scape)

Donghyun Kim and Byoung-Yun Kim*
International Institute for Applied Systems Analysis, A2361, Laxenburg, Austria & Korea Forest Research Institute, Seoul 130-712, Rep. of Korea,
* NEXTFOAM Inc., Seoul 153-790, Rep. of Korea

Wind field data is essential for the accurate prediction of forest fire spread. For the accurate prediction of local wind field we developed computational fluid dynamics simulation procedure. Using terrain and wind information as input data three dimensional computational mesh is automatically generated. GIS data of the format DEM(Digital Elevation Map) is directly converted to quad surface mesh and full hexahedral space mesh is automatically generated. Around this mesh we made extended computational domain. Open source CFD program, OpenFOAM is used for the flow solver. Atmospheric flow boundary condition is used at inlet and roughness length is considered at terrain by rough wall function. Simple Graphic User Interface of mesh generation is developed for user convenience. Hill named Saebyul-Orum in Cheju Island is simulated and we compared the results with measured data. The validation cases shows high quality grid, easy of use, good robustness and accuracy. This program can be applied to wind environment analysis and forest fire spread prediction. For better accurate result Large Eddy Simulation will be studied in future research. And for more convenience, development of direct interface among this CFD program and GIS database program.

Keywords: CFD, Wind, Wind field, OpenFOAM, Forest landscape

Introduction

To predict forest fire spread wind field data is necessary. Measured meteorological data is insufficient for detail flow field. So numerical simulation of flow field is very important to acquire local flow field data.

CFD (Computational Fluid dynamics) is vest tool for numerical simulation of flow field. Now CFD is very widely used in aerospace, mechanical, marine engineering and so on. But there are some special features and difficulties for atmospheric flow simulation like forest.

Atmospheric flow has strong turbulence fluctuation and shows transient characteristics. So Large Eddy Simulation (LES) is vest for it. But generally computational domain of atmospheric flow is very large and need very large number of computational meshes. So for practical point of view LES is very difficult to use until now. Instead of LES, steady Navier-Stokes solver with RANS turbulence model is suitable for practical use.

Atmospheric boundary condition for velocity and turbulence dissipation at inlet face is essential. For accurate simulation roughness length definition for target terrain is important. But accurate value of roughness length cannot be easily defined. In turbulence model atmospheric wall function also needed.

Third feature of atmospheric CFD is difficulties in define computational domain. For accurate CFD simulation, inlet condition is clearly defined. But in forest region defining inlet surface is ambiguous.

Another difficulties in atmospheric CFD is mesh generation. Usually we use terrain geometry in the format of height contour or cad file of dxf or iges format. Creating terrain surface using this type of data is very time consuming and cumbersome. So we need new methodology for 3D mesh generation from GIS data.

In this study we made new methodology to make three dimensional mesh from GIS data.

Corresponding author : bykim@nens.co.kr
Byoung-Yun Kim: CEO in NEXTFOAM Inc.

Nomenclature

K	Turbulent kinetic energy	**Greek letters**	
P	Pressure	ε	Turbulent dissipation rate
U	Velocity vector	K	Von Karman constant
z	Height	**Subscripts**	
z_0	Roughness length	P	First cell value near wall
C_μ	Constant	Ref	Reference value

In previous study we developed CFD solver [1]. In this study we compared our solver with open source CFD code, OpenFOAM. Then we selected OpenFOAM for our new flow solver. A small hill is simulated and compared to measured data.

Research site

Site of this research is Saebyul-Orum, a small hill in Cheju island in Korea. It is located in west side of Cheju island near sea and there are no mountains or large city between sea and it. It is covered with low scrub and weeds. So inlet condition can be easily defined. And there are measure data in another research for validate simulation results. In Fig. 1 show Saebyul-Orum(hill).

Fig. 1 Landscape of Saebyul-Orum(hill).

Computational domain & mesh generation

We developed 3D mesh generation program. Our idea is that GIS data of terrain in the format of DEM(Digital Elevation Map) is quad surface mesh itself. We can obtain DEM data of ascii format from GIS program at intervals of given value. The given value becomes surface mesh size and DEM file can be converted easily to plot3d mesh file format. In this study we used 5m resolution. Using boundary-fitted curvilinear coordinate system we can make three dimensional hexahedral mesh preserving orthogonality and smoothness. Input values to this program are DEM file name, surface mesh resolution, first cell height, domain height, number of node and stretching ratio in normal to terrain direction. Fig.2 is the graphic user interface of this program. Total time consumed to generate mesh is just 1 or 2 minutes.

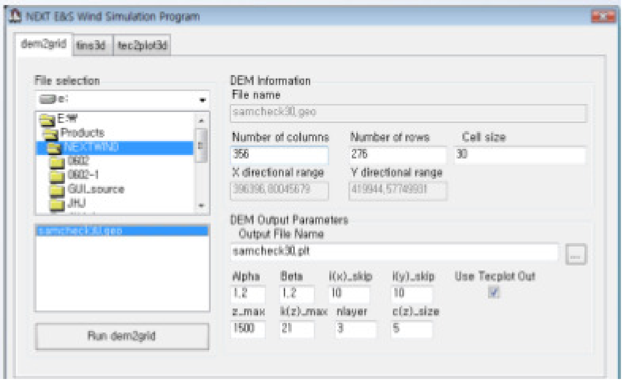

Fig. 2 GUI of mesh generation program.

If boundary faces of the mesh are flat, far away from the mountains and wind direction is normal to the face the mesh file is very good for CFD simulation. But in other cases we need expansion of the computational region and make boundary faces flatten and make inlet boundary normal to wind direction.

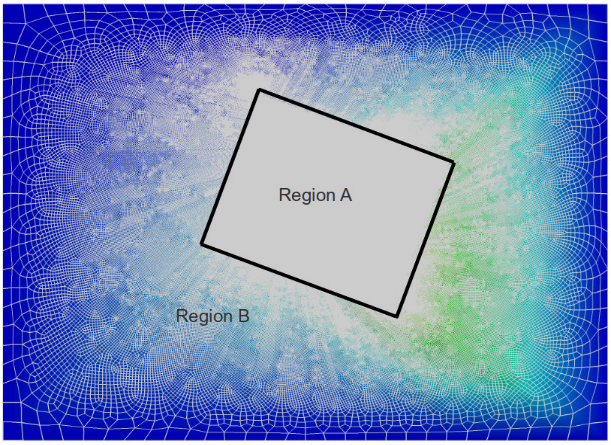

Fig. 3 Computational domain & surface mesh.

So we rotated generated mesh file with the angle of wind direction and made extended computational domain. In Fig. 3 the "Region A" is made from DEM data and "Region B" is extended region. It also shows surface mesh at terrain.

First cell height from terrain is 2m. 50 nodes is located in normal to terrain direction with stretching ratio 1.1. Min/max z-coordinate is 323 and 1500m.

All the cells are hexahedral and total number of cells is 11,091,900. About 70% of total cells is in “Region A”.

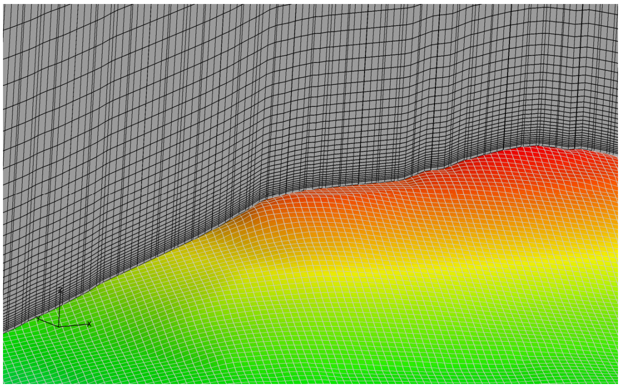

Fig. 4 Mesh detail for Saebyul-Orum(hill).

Solver & Boundary Conditions

Previously we developed flow solver for windscape simulation. But this solver is not parallelized yet. We considered two methods, parallelizing our previous solver and using open source CFD code, OpneFOAM [2] Two solvers are using the same governing equations and numerical schemes but OpenFOAM is parallelized very well. So in this study we used OpenFOAM standard solver simpleFoam of OpenFOAM-2.1.1.

Governing equation of simpleFoam is Navier-Stokes equation. Steady state and incompressible assumption is applied. For pressure-velocity coupling uses SIMPLE algorithm.

For turbulence modeling we used standard k-epsilon turbulence model. Linear solver for momentum and turbulence is smooth solver with GaussSeidel smoother. Linear solver for pressure is GAMG solver with Gauss-Seidel smoother. Second order upwind discretization scheme is used for momentum and 1st order upwind scheme is used for turbulence.

Boundary condition for inlet face(min. x) is atmospheric velocity inlet. Reference velocity was get from the AWS data of Korea Meteorological Administration. Velocity magnitude is 2.9m/s at the height of 10m and direction is 250deg.

Inlet velocity profile is in Eq. (1). Velocity profile is shown at Fig. 5.

$$U = \frac{U^*}{K} \ln\left(\frac{z}{z_0} + 1\right) \tag{1}$$

where, $$U^* = \frac{KU_{ref}}{\ln\left(\frac{z_{ref}}{z_0} + 1\right)}$$

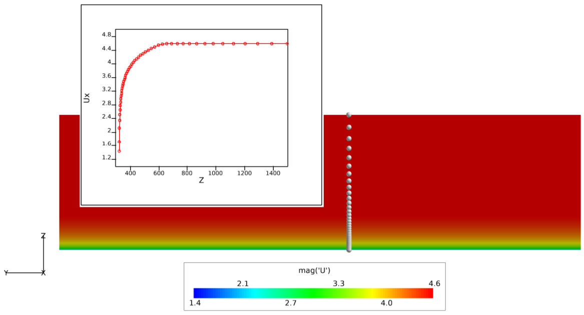

Fig. 5 Inlet velocity profile.

Roughness length, z_0 of terrain is set to 0.03. Pressure, p at inlet face is set to zero gradient. Turbulence kinetic energy, k at inlet face is assumed as constant value 1.3. Turbulence dissipation rate, ε at inlet face is set as atmospheric profile in the form of Eq. (2).

$$\varepsilon = \frac{U^{*3}}{K(z + z_0)} \tag{2}$$

Outlet face(maximum x) boundary condition for p is fixed value as 0. Boundary condition for side(min/max y) and top(max z) face is slip condition.

At the terrain face U is set to no-slip condition and p is set to zero gradient condition. Wall function is set to k and ε. For ν_t atmospheric rough wall function is used [3]. Velocity at first cell center is defined from Eq. (3).

$$U_p = \frac{U^*}{K \ln(z/z_0)} \tag{3}$$

Experiment data

In Fig. 6 shows anemometer and three measured location [4]. Measured height of velocity is 10m. P2 in Fig. 6 is the top of the hill and p1 is northwest side and P3 is southeast side of the hill.

Measured velocity magnitude and direction at this condition is at Table 1.

Table 1 Measured data in Saebyul-Orum(hill)

	direction	Velocity
P1	251 degree	3.3 m/s
P2	249 degree	4.9 m/s
P3	189 degree	0.9 m/s

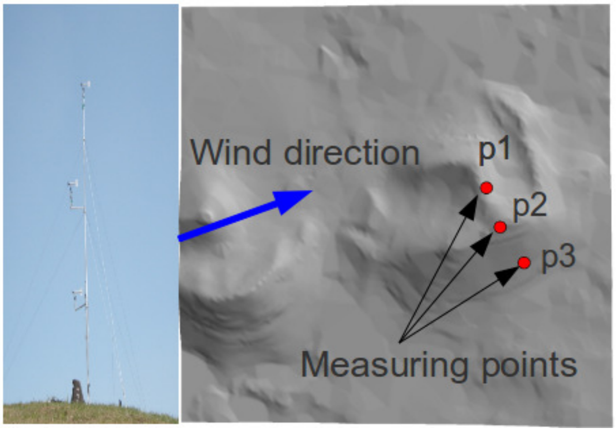

Fig. 6 Anemometer & measuring points in Sae-byul-Orum(hill).

Simulation Results

Simulation is done with 24 CPU cores at our linux cluster server using Intel 6-core CPUs. Computation time for get converged solution is about 4 hours.

Fig.7 and Fig. 8 shows velocity magnitude at terrain and flow pathline. In this two figures, we can see very strong circulating flow and high velocity magnitude gradient near the ridge of hills.

Fig. 9 shows surface flow at 3m and 10m height from terrain. Surface flow at 3m height is drawn with white color and at 10m height is with black. They shows similar flow pattern but at 10m height flow are somewhat stretched comparing at 3m. These figures show that complicated flow pattern is successfully captured with relatively low computing price. Calculated velocity magnitude and direction at location p1 is 3.5m/s and 247 deg. At location P2, velocity is 4.7m/s and direction is 253 deg. At location P3, velocity is 3.8m/s and direction is 246 deg. At location p1, p2 simulated results are very similar to experimental data. But location p3 shows very different results.

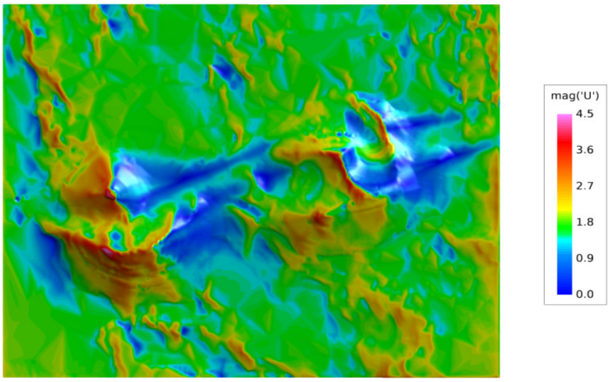

Fig. 7 Velocity magnitude at terrain.

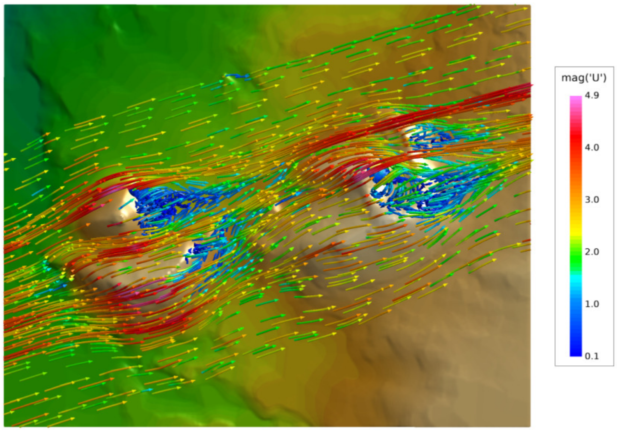

Fig. 8 Flow pathline.

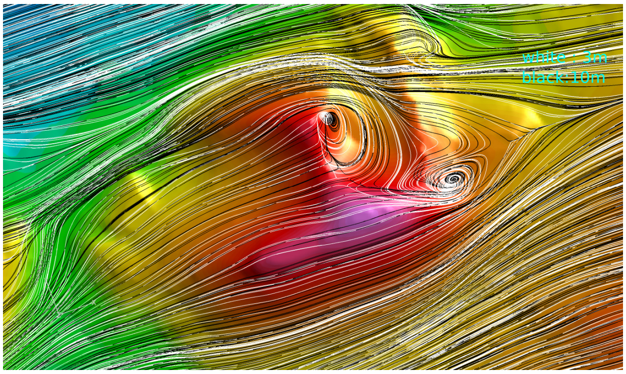

Fig. 9 Surface flow.

In Fig. 10 and Fig. 11 we draw velocity vector at two plane near location P1. P1 is at the ridge of the hill, so near this point flow separates and re-circulating region is created as shown in Fig. 10.

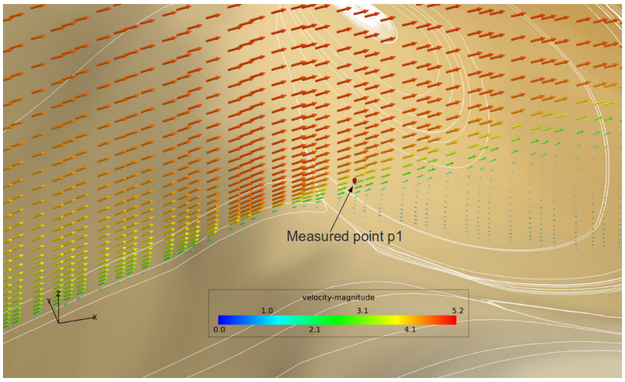

Fig. 10 Velocity near p1.

And near the terrain up to about 5m height there are weak cross flow directing north as shown in Fig. 11. Fig.12 shows detail flow field near P2. Near these points flow fields are relatively uniform comparing p1. Fig. 13 shows detail flow field near P3. Near P3 the flow path is smooth and seems there is not high velocity gradient or severe velocity fluctuation. So the flow direction and magnitude cannot be like measured data. We guess there were some problems with the anemometer at that time.

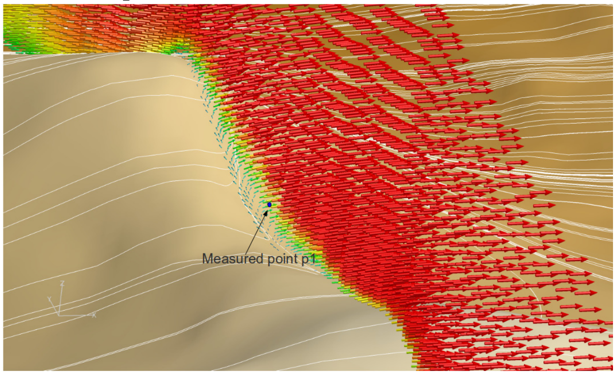

Fig. 11 Velocity near p1.

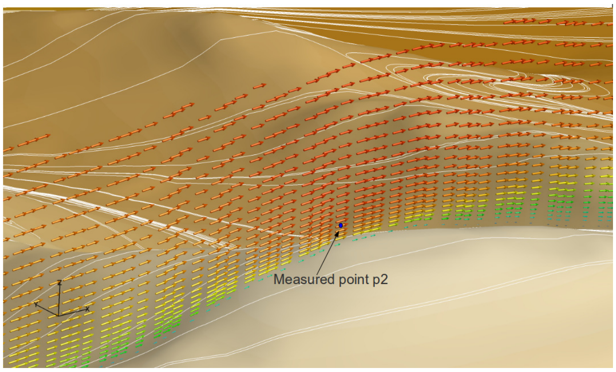

Fig. 12 Velocity near P2.

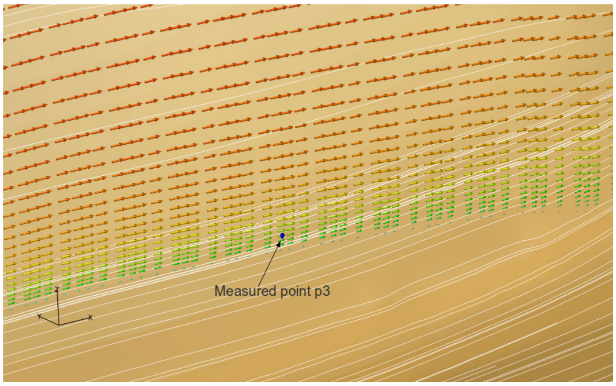

Fig. 13 Velocity near P3.

At P1 and P2 measured values are very similar with simulated results. So we think that this modeling method and numerical solution is good to predict wind field. But we also have some limits about the validation. First limit is that acceptable measured points are only two. Second limit are that because the two points are located at the ridge of the hill the flow direction will be similar to inlet direction.

Conclusion

In this study we made automatic 3D mesh generation program using DEM data. With this program we can obtain fully hexahedral high quality mesh very easily.

To remove the effect of geometry of terrain near boundary face, we expanded computational domain.

We used OpenFOAM as flow solver. And we can obtain very stable flow solutions.

As though some limit with measured value, we think this methodology of prediction flow field is good for our purpose.

Automatic 3D mesh generation program is based on windows and OpenFOAM solver run only on linux. Creating expanded computational domain should be done by manual. So for more convenience we will automate creating expanded computational domain and convert mesh generation program to linux. And then integrate all this job in a single graphic user interface. Integrating DEM database to simulation program and LES simulation will be done in the futue.

References

[1] D.H.Kim, M.B.Lee, B.Y.Kim : Development of Computational Wind Field Model (Wind scape) and Applicability Assessment, XXIII IUFRO world congress, (2010.)

[2] www.openfoam.com

[3] J.Sumner, C.Masson, Y.Odemark, M. Cehlin : OpenFOAM Simulation of atmospheric flow over complex terrain – performance in a blind comparison of CFD codes, 5th OpenFOAM Workshop (2010)

[4] KFRI : Annual Report of Study on Forest Fire Risk Map, (2011)