Abstract
Recently, prediction of the wind environment around a high-rise building using Computational Fluid Dynamics (CFD) has been carried out at the practical design stage. However, very few studies have examined the accuracy of CFD including the velocity distribution at pedestrian level. Thus, a working group for CFD prediction of the wind environment around a building was organized by the Architectural Institute of Japan (AIJ). This group consisted of researchers from several universities and private companies. In the first stage of the project, the working group planned to carry out cross comparison of CFD results of flow around a single high-rise building model placed within the surface boundary layer and flow within a building complex in an actual urban area obtained from various numerical methods. This was done in order to clarify the major factors affecting prediction accuracy. This paper presents the results of this comparison.

Keywords: CFD; wind environment assessment; cross comparison; revised k-ε models; actual urban area

Introduction
Recently, prediction of the wind environment around a high-rise building using Computational Fluid Dynamics (CFD) has been carried out at the practical design stage. The performance of CFD prediction of flow around a bluff body based on various turbulence models has been investigated by many authors [1-5]. However, these previous researches focused mainly on the prediction accuracy of the separating flow and pressure distribution around the roof. Few have examined the accuracy of CFD prediction of the velocity distribution at pedestrian level. Thus, a working group for CFD prediction of the wind environment around a building was organized by the Architectural Institute of Japan (AIJ). This group consists of researchers from several universities and private companies [Note].

At the first stage of the project, the working group planned to carry out cross comparison of CFD results of flow around a high-rise building predicted by various numerical methods, in order to clarify the major factors affecting prediction accuracy. The first part of this paper compares results of CFD prediction of flow around a 2:1:1 shaped building model and a 4:4:1 shaped building model placed within the surface boundary layer and flow within a building complex in an actual urban area using different grid systems.

2 Outline of cross comparisons
2.1 Flowfields tested
1) Test Case A (2:1:1 shaped building model)
Test Case A is the flowfield around a high-rise building model with the scale ratio of 2:1:1 placed within a surface boundary layer (Fig.1a). For this flowfield, detailed measurement was reported by Ishihara & Hibi [6]. The Reynolds number based on H (building height) and U0 (inflow velocity at z=H) was 2.4×10^4.

2) Test Case B (4:4:1 shaped building model)
For Test Case B, the flowfield around a building model with the scale ratio of 4:4:1 (Fig.1b) was selected. A wind tunnel experiment was carried out by the present authors to obtain the experimental data for assessing the accuracy of CFD results. The Reynolds number based on H (building height=4b) and U0 (inflow velocity at z=H=4b) was 7.2×10^4.
3) Test Case C (a building complex in an actual urban area)

The target for Test Case C was the flowfield within a building complex in an actual urban area (Fig.1(c)). A wind tunnel experiment was carried out by the present authors.

In the experiments for cases A and B, the wind velocity was measured by a split fiber type anemometer that could monitor each component of an instantaneous velocity vector. On the other hand, the mean wind velocity was measured by non-directivity thermistor anemometers for case C.

2.2 Specified Conditions

In order to assess the performance of turbulence models, the results should be compared under the same computational conditions. Special attention was paid to this point in this study. The computational conditions, i.e., grid arrangements, boundary conditions, etc., were specified by the organizers of the cross comparison, and is summarized in the Appendix 1 and Table 4. The

| Affiliation | Software | Turbulence model | Scheme for convection terms | Computational method and time integral scheme | $x_a/b$ | $x_b/b$ | CASE |
|-------------|----------|------------------|-----------------------------|---------------------------------------------|--------|--------|------|
| A           | STREAM   | k-ε (standard)   | QUICK                       | SIMPLE, steady solution                      | \      | 2.54   | KE1  |
|             | ver.2.10 |                  |                             |                                             |        |        |      |
| B           | STREAM   | k-ε (standard)   | QUICK                       | SIMPLE, steady solution                      | \      | 1.66   | KE2  |
|             | ver.2.10 |                  | (1st-order upwind for k and ε) |                                             |        |        |      |
| C           | STREAM   | k-ε (standard)   | QUICK                       | SIMPLE, steady solution                      | \      | 2.00   | KE3  |
|             | ver.2.10 |                  |                             |                                             |        |        |      |
| D           | STREAM   | k-ε (standard)   | QUICK                       | SIMPLE, steady solution                      | 0.87   | 2.98   | KE4  |
|             | ver.2.10 |                  |                             |                                             |        |        |      |
| E           | STAR-LT  | k-ε (standard)   | QUICK                       | SIMPLE, steady solution                      | \      | 2.20   | KE5  |
|             | ver.2.0  |                  |                             |                                             |        |        |      |
| F           | Homemade | k-ε (MMK) 11     | QUICK                       | MAC, unsteady solution with implicit scheme  | 0.65   | 2.72   | MMK1 |
| G           | FLUENT   | k-ε (standard)   | Central                     | SIMPLE, steady solution                      | \      | 2.41   | KE6  |
|             | ver.5.0  |                  |                             |                                             |        |        |      |
|             |          | k-ε (RNG) 10     |                             |                                             | 0.58   | 3.34   | RNG2 |
|             |          | k-ε (standard)   |                             |                                             | 0.53   | 3.11   |      |
|             |          | k-ε (Lk) 5      |                             |                                             | 0.58   | 3.19   |      |
|             |          | k-ε (modified Lk) 12 | QUICK                       | HSMAC, unsteady solution with implicit scheme |        |        |      |
|             |          | k-ε (MMK) 11    |                             |                                             | 0.63   | 2.70   |      |
|             |          | k-ε (Durbin) 13 |                             |                                             | >1.0   | 4.22   | DSM  |
|             |          | DSM 2           |                             |                                             | \      |        |      |
|             |          | LES(without inflow turbulence) 9 | SECOND-ORDER CENTERED DIFFERENCE | HSMAC, Convection terms: Adams-Bashforth scheme Diffusion terms: Cranck-Nicolson scheme | 0.62   | 1.02   | LES1 |
|             |          | LES(with inflow turbulence) 9 |                             |                                             | 0.50   | 2.10   | LES2 |
| H           | Homemade | k-ε (standard)   | QUICK                       | HSMAC, unsteady solution with implicit scheme | \      | 1.98   | KE7  |
|             |          |                  |                             |                                             |        |        |      |
| i           | Homemade | DNS 3           | 3rd-order Upward Scheme     | Artificial compressibility method, explicit  | 0.92   | 2.05   | DNS  |

Experiment [6] 0.52 1.42
3. Results and discussion

3.1 Test Case A (2:1:1 shaped building model)

The computed cases are outlined in Table 1. Nine groups have submitted a total of eighteen datasets of results. The performance of the standard k-ε and five types of revised k-ε models was examined. Furthermore, Differential Stress Model (DSM) [7] and Direct Numerical Simulation (DNS) with third-order upwind scheme [8] and Large Eddy Simulation (LES) using the Smagorinsky subgrid-scale model [9] were also included for comparison. The computational conditions in this test case are described in Appendix 1 and Table 4.

1) Reattachment lengths

The predicted reattachment lengths on the roof, \( X_R \), and that behind the building, \( X_F \), are given for all cases in Table 1. As shown by the results of the standard k-ε (KE1–8), the reverse flow on the roof, which is clearly observed in the experiment, is not reproduced. This was pointed out in previous researches by the present authors [1,2]. On the other hand, the reverse flow on the roof appears in the results for all revised k-ε models (LK1, RNG1, MMK1, RNG1, LK2, LK3, MMK2, DBN), although it becomes a little larger than that in the experiment. In the DSM result, the predicted separated flow from a windward corner is too large, and does not reattach to the roof. The result of LES without inflow turbulence (LES1) can reproduce the reattachment on the roof, but \( X_R \) is somewhat overestimated in this case. On the other hand, the result of LES with inflow turbulence (LES2) shows close agreement with the experiment.

The evaluated reattachment length behind the building, \( X_F \), is larger than in the experiment in all cases compared here except for LES1. It is surprising to see that there are significant differences between the \( X_F \) values of the standard k-ε model. As is already noted, the grid arrangements and boundary conditions were set to be identical in all cases, and QUICK scheme was used for convection terms in many cases. The reason for the difference in \( X_F \) values predicted by the standard k-ε models is not clear, but it may be partly due to differences in some details of the numerical conditions, e.g. the convergence condition, etc. The results of the revised k-ε models except for the Durbin’s model are in the tendency to evaluate \( X_F \) larger than the standard k-ε model. This discrepancy is improved in the LES and DNS computations. On the other hand, DSM greatly overestimates \( X_F \). The overestimation of reattachment length behind a three-dimensional obstacle was also reported by Lakehal and Rodi [5]. In ref. [5], predicted results of flow around a surface mounted cube obtained by five types of k-ε models, i.e. the standard k-ε model, Kato-Lauder model, Two-layer k^{1/2} velocity-scale-based model, Two-layer k^{1/2} velocity-scale-based model with Kato-lauder modification and Two-layer (v’^2)^{1/2} velocity-scale-based model, were compared. The all models compared in ref. [5] including three types of the

| Affiliation | Software | Turbulence model | Scheme for convection terms | Computational method and time integral scheme | \( X_F/b \) | CASE |
|-------------|----------|-----------------|-----------------------------|-----------------------------------------------|-------|------|
| B           | FLUENT 6.0 | \( k-\varepsilon(standard) \) | QUICK | SIMPLE, steady solution | 10.8 | KE1 |
|             |          | \( k-\varepsilon(RNG) \) 12 | | | 13.6 | RNG1 |
|             |          | \( k-\varepsilon(Realizable) \) 14 | | | 12.4 | REA |
| C           | STREAM 4.0 | \( k-\varepsilon(standard) \) | QUICK | SIMPLE, steady solution | 9.2 | KE2 |
|             |          | \( k-\varepsilon(LK) \) 9 | | | 13.2 | LK |
|             |          | \( k-\varepsilon(RNG) \) 10 | | | 13.4 | RNG2 |
| G           | STAR-CD 3.15 | \( k-\varepsilon(standard) \) | QUICK | SIMPLE, steady solution | 11.6 | KE3 |
|             |          | \( k-\varepsilon(Shih) \) 15 | | | 13.2 | SHI |
|             | Homemade | \( k-\varepsilon(standard) \) | QUICK | HSMAC, solution with implicit scheme | 10.0 | KE4 |
|             |          | \( k-\varepsilon(Durbin) \) 13 | | | 9.2 | DBN |
| I           | Homemade | \( k-\varepsilon(standard) \) | QUICK | Artificial compressibility method, explicit | 11.6 | KE5 |
|             |          | \( k-\varepsilon(Mixed time scale) \) 16 | | | 11.8 | MIX |
|             | DNS with 3rd-order Upwind Scheme 17 | | | | 7.6 | DNS |

Experiment | 7.6 |
two-layer models overpredicted the reattachment length behind the obstacle as well as in this study. In the two-layer models, viscous-affected near-wall region is resolved by a low Reynolds type one-equation model, while the outer region is simulated by the k-ε model. In the one-equation model, the eddy viscosity is made proportional to a velocity scale and a length scale.

The size of the recirculation region behind the building is strongly affected by the momentum transfer mechanism in the wake region, where vortex shedding plays an important role. Thus, the reproduction of vortex shedding is significantly important for accurately predicting the X_F value. However, none of the k-ε models compared here could reproduce vortex shedding. This resulted in underestimation of the mixing effect in the lateral direction causing too large a recirculation region behind the building.

2) Lateral distributions of <u> near ground surface (z=1/16H)

Fig. 2 shows the lateral distributions of the streamwise mean velocity component, <u>, near the ground surface in the area affected by the separation at the front corner in the selected cases. The peak in the measured velocity distribution appears at y/b=-0.9. The standard k-ε (KE8) and the modified LK model (LK3) underestimate the velocity around this point. For the Durbin’s model (DBN), the position and the peak value in the velocity distribution are well reproduced. In DSM, the evaluated velocities are generally larger in the region of y/b<-1.5 than those with other computations.

3.2 Test Case B (4:4:1 shaped building model)

Outlines of computed cases are listed in Table 2. Five groups have submitted a total of twelve datasets of results. The performance of the standard k-ε and six types of revised k-ε models was examined. Furthermore, DNS with a third-order upwind scheme [15] was also included for comparison. Computational conditions in this test case are described in Appendix 1 and Table 4.

1) Reattachment length

The predicted reattachment lengths behind the building, X_F, are given for all cases in Table 2. The result of the DNS with a third-order upwind scheme shows very close agreement with the experiment. The evaluated X_F value is larger than the experimental value in all computed results based on the standard and revised k-ε models for this test case, as well as in the results for Test Case A presented in 3.1. The results of the revised k-ε models except for Durbin’s model predict a larger X_F value than the result of the standard k-ε. This tendency is also similar to the results for Test Case A.

2) Lateral distributions of each component of velocity vector near ground surface (z=1/16H)

Fig. 3(a) shows the lateral distributions of scalar velocity and each component of mean velocity vector near the ground surface in the area affected by the separation at the frontal corner. These values are normalized by the velocity value at the same height at the inflow boundary. The peak measured scalar velocity distribution appears at y/b=-0.9. The standard k-ε and the revised k-ε models overestimate the velocity around this point. As shown in Fig. 3(b), in this area, the streamwise component, <u>, of the mean velocity vector decreases as the distance from the side-wall decreases in the experimental result. On the other hand, the measured <u> values decrease in the area and increase in the area 4<y/b as distance from the wall increases. The results of the standard k-ε model do not reproduce this tendency at all, while the result of the revised k-ε models show better agreement with the measured distribution. Between the results of the two revised k-ε models compared here, the distribution of <u> obtained from the LK model shows much better agreement with the experiment than do the RNG models.

As shown in Fig. 3(d), the peak in the measured <w>
distribution appears at y/b ≈ 2.5. For the standard k-ε, the peak value is hardly reproduced. However, the result of the RNG and LK models show generally close agreement with the experiment.

3.3 Test Case C
(building complex in actual urban area)

Finally, prediction accuracy for wind environment within an actual building complex, located in Niigata City, Niigata Prefecture, Japan, is examined. Fig. 6 illustrates three-target buildings (A~C). Building A is 60m high, and buildings B and C are both 18m high. The surrounding area is mostly covered with low-rise residential houses. The wind rose of Niigata Local Meteorological Observatory is shown in Fig. 4.

Here, we compare the results predicted with three different codes: a homemade CFD code and two commercial CFD codes. The computational conditions are described in Appendix 1 and Table 4. Data from an identical CAD file is used to reproduce the geometries of the surrounding building blocks. This CAD file is produced from a drawing of the experimental model. Specifications of the CFD codes are compared in Table 3. Fig. 5 illustrates an enlarged view of the computational grid around the high-rise building model in all cases.

Although CFD simulations were performed for sixteen different wind directions, only the wind distributions for wind directions NNE and W are shown here due to the limitation of available space. These are the prevailing wind directions in Niigata City. Since no clear differences were observed between the horizontal distributions of scalar velocity near the ground surface (z=2m) predicted by the three CFD codes, the results from Code T are shown in Fig. 6. This figure illustrates the horizontal distributions of scalar velocity near the ground surface (z=2m). The values in Fig. 6 are normalized by the velocity at the same height at the inflow boundary. It can be seen that high velocity regions appear in the area around the corner of the north and east sides of building A and strong wind blows into the space between buildings with the wind direction from NNE. On the other hand, a high velocity region is observed in the area around the corner of the south side of building A with the wind direction from W. The velocities in the street for the NNE wind direction are smaller than those for the W wind direction.

Fig. 7 shows the correlation between the normalized velocities obtained for each code and those of the wind tunnel experiment. The black circle indicates the

| Affiliation | Software | Turbulence model | Scheme for convection terms | Computational method and time integral scheme | computation time | Grid System |
|-------------|----------|------------------|----------------------------|---------------------------------------------|-----------------|-------------|
| C           | Commercial code (Code M) | k-ε (standard) | QUICK | SIMPLE, steady solution | 6 hours/wind direction a Xeon 2GHz | Structured Grid |
| D           | Commercial code (Code T) | k-ε (standard) | MUSCL (2nd-order) | SIMPLE, steady solution | 2 hours/wind direction a Pentium4 2.8GHz | Unstructured Grid |
| I           | Homemade (Code O) | k-ε (standard) | 3rd-order Upwind Scheme | Artificial compressibility method, explicit | 2 hours/wind direction a Pentium4 2.1GHz | Overlapping Structured Grid |
velocities at the measuring points in the wake region. A similar tendency is observed for all results in Figs. 7(1) and (2). It is found that the scalar velocity predicted by all CFD codes tested here tends to be smaller in the wake region compared to the experimental value, as well as in the results for Test Cases A and B. Except for the velocities in the wake region, the CFD analyses agree closely with the experimental results. The difference between the scalar velocities in the wake region from the CFD and the experimental results is partly because the definition of the mean scalar velocity measured by the non-directivity thermistor anemometers is different from that of CFD (cf. Appendix 3). This point will be examined in more detail in the next stage of this project.

Fig. 8 compares the normalized velocities at each measuring point. It is confirmed that all three CFD codes compared here can predict the distribution of scalar velocity in reasonable agreement with the measurements except for the wake region and the region far from the target buildings. The prediction error in the far region is mainly caused by the insufficient grid resolution in this region, which is obviously not fine enough.

4 Conclusions

1) In the first part of this paper, the flowfields around two types of a high-rise building model, i.e. a 2:1:1 shaped model and a 4:4:1 shaped model placed within the surface boundary layer, were predicted using the standard k-ε model, the revised k-ε models, DSM, LES
and DNS with a 3rd order upwind scheme. Results of these predictions were compared with experimental data.

2) The standard k-ε model could not reproduce the reverse flow on the roof in Test Case A. This drawback was corrected by all revised k-ε models tested here. However, the revised k-ε models except for the Durbin’s model overestimated the reattachment length behind the building in comparison with the standard k-ε model in Test Cases A and B.

3) The LK and RNG models provided more accurate results than did the standard k-ε model in the area around the side face of the building near the ground surface in Test Case B.

4) In the latter part, the flowfield within a building complex in an actual urban area (Test Case C) was predicted by three different CFD codes based on different grid systems. Results of these predictions were compared with experimental data. No clear differences were observed between the CFD results given from these three codes for this test case under the computational conditions specified by the organizer.

5) The CFD codes compared here can predict the distribution of the scalar velocity at pedestrian level within the actual building complex in reasonable agreement with the measurements except for the wake region and the region far from the target buildings where the grid resolution is obviously not fine enough.

Acknowledgements
The authors would like to express their gratitude to the members of working group for CFD prediction of the wind environment around a building [cf. Note].

Note
The working group members are: A. Mochida (Chair, Tohoku Univ.), Y. Tominaga (Secretary, Niigata Inst. of Tech.), Y. Ishida (Kajima Corp.), T. Ishihara (Univ. of Tokyo), K. Uehara (National Inst. of Environ. Studies), R. Ooka (I.I.S., Univ. of Tokyo), H. Kataoka (Obayashi Corp.), T. Kurabuchi (Tokyo Univ. of Sci.), N. Kobayashi (Tokyo Inst. Polytechnics), N. Tuchiya (Takenaka Corp.), Y. Nonomura (Fujita Corp.), T. Nozu (Shimizu Corp.), K. Harimoto (Taisei Corp.), K. Hibi (Shimizu Corp.), S. Murakami (Keio Univ.), R. Yoshie (Maeda Corp.)

Appendix 1 Outline of computational conditions specified by the organizer

1) Computational domain:
The computational domain covers the specified sizes, which corresponds to the size of the wind tunnel in the experiment. The computational domain was divided into a specified number of grids. The size of the computational domain, grid discretization and the minimum grid interval are summarized in Table 4.

2) Inflow boundary:
At the inflow boundary, the interpolated values of $u$ and k obtained from the experimental results are imposed. The vertical profile of mean velocity $u(z)$ approximately obeyed the power law expressed as $u(z) \propto z^\alpha$ in the experiment. The value of $\alpha$ is obtained from the relation $P_{k=0}$. The value of $\alpha$ for each test case is shown in Table 4.

3) Ground surface boundary [18]:
In these cross comparisons, the wall function based on the logarithmic law of the form containing the roughness length $z_0$ is employed. This is mainly because the velocity profile should be maintained in the area apart from the building. $z_0$ values for each test case are shown in Table 4. The friction velocity $u^*$ is obtained from the relation $P_{\kappa=0}$. The value of $\kappa$ for each test case is shown in Table 4.

4) Lateral and upper surfaces of computational...
Table 4 Computational conditions

| Test Case | Size of computational domain | Grid discretization | Minimum grid interval | $\alpha$ | $z_0$ | Remarks |
|-----------|------------------------------|--------------------|----------------------|---------|-------|---------|
| A         | (x=13.75,y=11.25)           | 0.075 [-]          | 0.27                 | 1.125×10^{-1} [H] |
| B         | (x=33,y=32)                 | 0.1 [-]            | 0.25                 | 4.8×10^{-1} [H] * |
| C         | About 500(x=500,y=300)      | Not specified**    | Not specified**      | 0.25    | 9.6×10^{-1} [m] |

*In some cases, only the half domain is reproduced imposing the symmetry flow conditions along the centerline.

**The distance between the ground surface and definition point of tangential velocity components in the 3rd grid was set to 2.0m.

domain:
In Test Case A, the wall functions based on a logarithmic law for a smooth wall are used.
In Test Cases B and C, the normal velocity components defined at the boundaries and the normal gradients of the tangential velocity components, $k$, $\varepsilon$ across the boundaries, were set to zero.

5) Building surface boundary:
The wall functions based on logarithmic law for a smooth wall are used.

6) Downstream boundary:
Zero gradient condition is used for all velocity components, $k$ and $\varepsilon$.

Appendix 2 Grid arrangements employed in Test Case C

Code M: A structured grid system was employed. The whole computational domain was divided into 150×140×38 grids. The target buildings were surrounded by 2m grids.

Code N: An unstructured grid system with prismatic cells over the ground and building surface was used. The whole computational domain was divided into 800,000 using Tetra, Pyramid and Prism cells. The distance from solid surfaces of ground and building to the first interior grid point was set to about 0.7m.

Code O: An overlapping structured grid system was employed. The whole computational domain was divided into 250,000. The grid interval was 5m in the horizontal directions. The sub-computational domain was divided into 250,000. The grid interval in the horizontal directions was 2m. The distance between the ground surface and the first interior grid point was set to about 0.7m.

Appendix 3
The mean scalar velocity measured in the wind tunnel using a non-directivity thermistor anemometer ($S_{exp}$) is regarded as the time averaged instantaneous scalar velocity, which can be expressed as:

$$S_{exp} = \left( \frac{1}{3} u^2 + \frac{1}{3} v^2 + \frac{1}{3} w^2 \right)^{1/2}$$

On the other hand, the mean scalar velocity given from the $k$-$\varepsilon$ model ($S_{k-\varepsilon}$) is calculated from the time averaged velocities vector, namely,

$$S_{k-\varepsilon} = \left( \frac{1}{3} \langle u^2 \rangle + \frac{1}{3} \langle v^2 \rangle + \frac{1}{3} \langle w^2 \rangle \right)^{1/2}$$

Thus, the output of the thermistor anemometer is larger than that given from the $k$-$\varepsilon$ model.

$$S_{exp} = \left( \frac{1}{3} \langle u^2 \rangle + \frac{1}{3} \langle v^2 \rangle + \frac{1}{3} \langle w^2 \rangle \right)^{1/2}$$

Here, $u$, $v$, $w$: three components of instantaneous velocity vector, $\langle f \rangle$: time-averaged value of $f$, $f' = f - \langle f \rangle$.

References
1) Murakami, S., Mochida, A. and Hayashi, Y. (1990), "Examining the $k$-$\varepsilon$ model by means of a wind tunnel test and large eddy simulation of turbulence structure around a cube", J. Wind Eng. Ind. Aerodyn., 35, 87-100
2) Murakami, S. (1993), "Comparison of various turbulence models applied to a bluff body", J. Wind Eng. Ind. Aerodyn., 46&47, 21-36
3) Kato, M. and Launder, B.E. (1993), "The modeling of turbulent flow around stationary and vibrating square cylinders", Prep. of 9th Symp. on Turbulent shear flow, 10-4-1-6
4) T.Tominaga, H. Kawai, S. Kawamoto et al (1997), "Numerical prediction of wind loading on buildings and structure - AIJ cooperative project on CFD", J. of Wind Eng. and Ind. Aerodyn 67&68, 671-685
5) D. Lakehal, W. Rodi (1997), "Calculation of the flow past a surface-mounted cube with two-layer turbulence models", J. Wind Eng. Ind. Aerodyn., 67&68 (1997) 65-78
6) Ishihara, T. and Hibi, K. (1998), "Turbulent measurements of the flow field around a high-rise building", J. of Wind Eng., Japan, No.76, 55-64 (in Japanese)
7) Murakami, S., Mochida, A. and Ooka, R. (1993), "Numerical simulation of flowfield over surface-mounted cube with various second-moment closure models", 9th Symp. on Turbulent shear Flow, 13-5
8) Kataoka, H. and Mizuno, M. (2002), "Numerical flow computation around aerelastic 3D square cylinder using inflow turbulence", Wind and Structures, Vol. 5, No. 2-4, pp.379-392
9) Tominaga, Y., Mochida, A. and Murakami, S. (2003) "Large Eddy Simulation Flowfield around a High-rise Building", 11th ICWE,B10.5
10) Yakhout, V. and Orszag, S.A (1986), "Renormalization group analysis of turbulence", J. Sci. Comput. 1, 3
11) Tsuchiya, M., Murakami, S., Mochida, A., Kondo, K. and Ishida, Y. (1997), "Development of a new $k$-$\varepsilon$ model for flow and pressure fields around bluff body", J. of Wind Eng. and Ind. Aerodyn. 67/68, 169-182
12) Tominaga, Y. and Mochida, A. (1999), "CFD prediction of flowfield and snowdrift around building complex in snowy region", J. Wind Eng. Ind. Aerodyn. 81, 273-282
13) Durbin, P.A. (1996), "On the $k$-$\varepsilon$ stagnation point anomaly", Int. J. Heat and Fluid Flow, 17, 89-90
14) T.H. Shih, W. W. Liou, A. Shabbir, Z. Yang and J. Zhu (1995), "A New $k$-$\varepsilon$ Eddy Viscosity Model for High Reynolds Number Turbulent Flows" Computers Fluids Vol. 24 No.3 pp.227-238
15) T.H. Shih, J. Zhu, J.L. Lumley (1993), "A realizable Reynolds stress algebraic equation model", NASA TM-105993
16) Nagano, Y. and Hattori, H. (2003) "A new low-Reynolds number turbulence model with hybrid time-scale of meanflow and turbulence for complex wall flow", Proc. 4th Int. Symp. On Turbulence, Heat and Mass Transfer(Eds. K. Hanjalic, Y. Nagano and F. Arinc), Antalya, Turkey, October 12-17
17) Kataoka, H. (2003) "Large Eddy Simulation of building", Summaries of Technical Papers of Annual Meeting, Environ. Engg. II, AIJ (in Japanese)
18) Yoshiie, R. (1999), "CFD analysis of flow field around a high-rise building", Summaries of Technical Papers of Annual Meeting, Environ. Engg. II, AIJ (in Japanese)