Second-order kinetic energy-conservative analysis of incompressible turbulence under a collocated grid system

Hiroki Suzuki¹,²,⁴, Yutaka Hasegawa², Masaya Watanabe³, Ushijima Tatsuo² and Shinsuke Mochizuki¹

¹Graduate School of Sciences and Technology for Innovation, Yamaguchi University, 2-16-1 Tokiwadai, Ube-shi, Yamaguchi 755-8611, Japan
²Department of Electrical and Mechanical Engineering, Nagoya Institute of Technology, Gokiso-cho, Showa-ku, Nagoya-shi, Aichi 466-8555, Japan
³Department of Engineering Physics, Electronics and Mechanics, Nagoya Institute of Technology, Gokiso-cho, Showa-ku, Nagoya-shi, Aichi 466-8555, Japan
⁴E-mail: h.suzuki@yamaguchi-u.ac.jp

Abstract. The study presented in this paper investigates the second-order conservation error of kinetic energy of an incompressible flow under a collocated grid. A collocated grid is often used to numerically analyse the flow around an aerofoil. The kinetic energy conservation error under a collocated grid is often greater than that under a staggered grid. The results of analysis based on coarse computational grids, such as LES analysis, are considerably affected by the kinetic energy conservation error. This study shows the results of numerical analysis in which the kinetic energy has a second-order conservation error under the collocated grid. This higher-order kinetic energy is strictly validated using inviscid flow. Furthermore, the results of conservation errors of higher-order statistics of kinetic energy are discussed in this study. The numerical analysis is then validated using a turbulent channel flow. The statistics obtained by this analysis agree well with those of previous studies in turbulent channel flows.

1. Introduction
The Navier-Stokes equation is one of the governing equations of incompressible turbulent flows, for which the derivation of analytical solutions is not possible. Therefore, to aid the research of incompressible turbulent flows, wind tunnel experiments and numerical analysis methods are used. Wind tunnel experiments are carried out cautiously, so as to reduce the uncertainty of experimental measurements, such as that due to hot-wire measurements, thereby accurately reproducing the targeted turbulent flow [1–4]. Similarly, in numerical analysis, error reduction has been a priority in previous studies. In the numerical analysis of incompressible turbulent flows, the conservation laws of the governing equations should be held in the discretised governing equations. Also, to increase the accuracy of the numerical analysis, it is necessary to reduce the conservation error of the kinetic energy, i.e. the secondary conservation error [5].

By improving the accuracy of the kinetic energy conservation law in addition to those held in the governing equations, secondary conservation, highly accurate numerical analysis is realised. The advantage of this technique is further found in large-eddy simulations, in which the coarser computational grid is used, as the source terms of the conservation error depend directly on the spatial accuracy [5]. A discretisation technique has been developed under the staggered grid to explicitly
achieve secondary conservation [5]. This discretisation technique has also been used to discretise governing equations based on cylindrical coordinate systems [6]. Also, there are examples for the numerical analysis of incompressible turbulent flow using the discretisation technique. Some of these works [7-8] performed a numerical analysis of grid-generated turbulence often used in experimental measurements by utilising the discretisation technique with secondary conservation.

A collocated grid has been used instead of a staggered grid to simulate the flow around an air foil with flow separation [9-10]. Previously conducted studies attempted to improve the analysis of incompressible turbulent flows under a collocated grid [11-14]. Some of these studies focused on the conservation error of kinetic energy under a collocated grid. Another study improved the order of accuracy of the kinetic energy conservation error under a collocated grid by introducing a straightforward method [15]. This work improved the accuracy of the conservation error of kinetic energy to the second order. In addition, a study conducted recently used a CFD solver [16]. In the simulation, it was shown that the conservation error of kinetic energy might not decrease when a CFD solver is used. On the other hand, few computing methods based on collocated grids were able to reduce the conservation error of kinetic energy.

The purpose of this study is to develop and validate a computational code, characterised by higher-order conservation of kinetic energy under a collocated grid, by using a numerical method proposed in a previously conducted study [15]. In this study, the conservation error of the kinetic energy is accurate to the second order with respect to time. Compared with the numerical analysis method, the use in this study of a second-order time integration scheme, such as the second-order Adams Bashforth scheme, makes the order of accuracy of the conservation error sufficiently high. In this study, a three-dimensional periodic inviscid flow is used to determine the conservation error of kinetic energy. The use of an inviscid flow enables the determination of the conservation error with greater accuracy. The conservation error of the intensity of fluctuation of kinetic energy, having a dimension of the fourth-order of velocity, is also computed. A numerical analysis is further performed, using a turbulent channel flow, to provide an example of the actual turbulent flow fields [17-19].

2. Inviscid flow analysis

2.1. Numerical methods

In this study, the analysis of an incompressible turbulent flow is undertaken. The governing equations are the continuity equation and the Navier-Stokes equation. They are given as follows:

\[ \frac{\partial u_i}{\partial x_i} = 0 \]

\[ \frac{\partial u_i}{\partial t} = - \frac{\partial u_i u_j}{\partial x_j} - \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}, \text{ where } \tau_{ij} = \frac{1}{Re} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right). \]  

(1)

Here, \( t \) is non-dimensional time, \( x_i \) is the physical space for the \( i^{th} \) component, \( u_i \) is the normalised instantaneous velocity of the \( i^{th} \) component, \( p \) is the normalised pressure, and \( Re \) is the Reynolds number. Since an inviscid flow is analysed in this section, the viscous terms in Equation (1) are not used. Equation (1) for the inviscid flow gives the following equation under the collocated grid system:

\[ \delta F_i/\delta x_i = 0 \text{ and } \delta u_i/\delta t = - \delta_i (F_j \bar{u}_j)/(\delta x_j - \delta p/\delta x_i) = 0. \]  

(2)

Here, \( F_i \) is the velocity component defined on the surface of the computational cell for the \( i^{th} \) component. The velocity component \( u_i \) for the \( i^{th} \) component and pressure \( p \) are defined at the centre of the computational cell. The discretisation and approximation forms used in the above equation are given by the second-order difference scheme as follows:

\[ \delta f/\delta x_1 = (f(x_1 + h/2, x_2, x_3) - f(x_1 - h/2, x_2, x_3)) / h, \]

\[ \delta f/\delta x_2 = (f(x_1, x_2 + h, x_3) - f(x_1, x_2 - h, x_3)) / 2h, \]

(3)

and

\[ \delta f/\delta x_3 = (f(x_1, x_2, x_3 + h) - f(x_1, x_2, x_3 - h)) / 2, \]

where the same forms are used for the other components.
Equation (2), which is discretised under the collocated grid, is solved using the time integration method. Here, the second-order Adams-Bashforth scheme is used for conducting the analysis. Using this scheme, the governing equations are solved for the temporal direction as follows:

\[
\tilde{u}_i^{n+1} = u_i^n - (3/2) \Delta t N_i^n + (1/2) \Delta t N_i^{n-1},
\]

\[
\delta_i \left( \frac{\delta (p^{n+1})}{\delta x_i} / \delta x_i \right) = (1 / 2 \Delta t) \delta_i \left( \overline{u_i^{n+1/2}} \right) / \delta x_i
\]

\[
u_i^{n+1} = \tilde{u}_i^{n+1} - 2 \Delta t \delta_i \frac{p^{n+1}}{\delta x_i},
\]

\[
F_i^{n+1} = \overline{u_i^{n+1/2}} - 2 \Delta t \delta_i \frac{p^{n+1}}{\delta x_i},
\]

where \( N_i^n = \delta_i \left( \overline{F_i^{n+1/2}} \right) / \delta x_i \).

In the discretised governing equations for incompressible flow under the staggered grid, the error in conservation of kinetic energy is given as a power function in the time integration method, where the order of accuracy is consistent with the value of power. Thus, by implementing a higher-order time integration method such as the fourth-order Runge-Kutta method, the error in conservation of kinetic energy can be reduced to a negligible value. However, in a collocated grid system, even if an accurate method discretises the convection term, the characteristics of the error in conservation of kinetic energy are different from those of the staggered grid. Specifically, the error in conservation of kinetic energy is proportional to the first power of the time increment. This effect is caused by the discretisation of pressure terms in the kinetic energy equation for the collocated grid. In a numerical simulation of incompressible flow, a high-order time-integration method is generally used. Therefore, for the analysis under a collocated grid, the error in conservation of kinetic energy is more significant than that in the staggered grid.

A previous study focused on the discretisation of pressure terms and reduced the conservation error under the collocated grid. This study proposed to reduce the conservation error using a method that analyses pressure increments instead of the pressure, which is similar to the SMAC method. The time evolution procedure of this method is as follows:

\[
\tilde{u}_i^{n+1} = u_i^n - (3/2) \Delta t \left( N_i^n + \delta_x p^n / \delta x_i \right) + (1/2) \Delta t \left( N_i^{n-1} + \delta_x p^{n-1}/ \delta x_i \right),
\]

\[
\delta_i \left( \frac{\delta (\Delta p^{n+1})}{\delta x_i} / \delta x_i \right) = (1 / 2 \Delta t) \delta_i \left( \overline{u_i^{n+1/2}} \right) / \delta x_i
\]

\[
u_i^{n+1} = \tilde{u}_i^{n+1} - 2 \Delta t \delta_i \frac{\Delta p^{n+1}}{\delta x_i},
\]

\[
F_i^{n+1} = \overline{u_i^{n+1/2}} - 2 \Delta t \delta_i \frac{\Delta p^{n+1}}{\delta x_i},
\]

\[
p^{n+1} = p^n + \Delta p^{n+1}.
\]

In this method, the kinetic energy conservation error is proportional to the square of the time increment. In this study, the error in conservation of kinetic energy is reduced using this method.

### 2.2. Results of the inviscid flow analysis

In this study, in order to validate the characteristics of conservation of kinetic energy for the numerical analysis, an inviscid flow with periodic boundary conditions is used. Inviscid flow has been used to validate the characteristics of conservation of kinetic energy in earlier studies. In this study, a three-dimensional inviscid flow is used. The governing equations are the continuity equations and the Euler equations. The second-order central difference method was used to discretise the spatial derivative. A second-order Adams-Bashforth method was used as a time integration scheme. The Poisson equation was solved through the BiCGStab scheme. The initial flow field, which satisfied the continuity equation, was calculated from the divergence of the random vector potential. The initial velocity field is normalised and its kinetic energy is unity. Deviation from the constant energy is shown as a function of time increment. The present study calculates the absolute difference between the values of mean kinetic energy at \( t = 0 \) and \( t = 10 \). Computational domain size of the 3D periodic box is \( 2 \pi \times 2 \pi \times 2 \pi \). The number of computational grids are \( 16^3 \). Cases 1 and 2 denote numerical cases using Equation (4) and Equation (5), respectively.
This study focuses on the order of accuracy in the conservation error of kinetic energy with respect to the time increment. Figure 1(a) shows the absolute error in the conservation of the kinetic energy between the values of mean kinetic energy at $t = 0$ and $t = 10$, as a function of time increment. Here, analytically, the fluctuating kinetic energy does not change between these two times. As shown in the figure, the conservation error decreases as the time increment decreases. For Case 1, where the MAC method is used, the conservation error is proportional to the time increment itself. On the other hand, for Case 2, where the SMAC method is used, the conservation error is proportional to the square of the time increment, which is also used in the present study. As shown in the figure, it is validated that the conservation error of the present analysis has second-order accuracy of time increment. The conservation error of Case 2 is much smaller than that of Case 1 with a smaller time increment.

2.3. Discussion

The kinetic energy has the dimension of the square of velocity. Turbulence statistics with dimension of the fourth power of velocity are also used in various studies. For example, the computation of the kurtosis factor of the velocity field requires the mean of the square of velocity as well as the mean of the fourth power of velocity. In this study, the intensity of the fluctuating kinetic energy is used as a flow statistic, which has the dimension of the fourth power of velocity. Figure 1(b) shows temporal variations of the kinetic energy and the intensity of the energy in the inviscid flow field. In the figure, $\Delta t$ is set to 0.1, to increase the magnitude of the conservation error. The conservation error of the fourth power of velocity increases with time more rapidly than the conservation error of the kinetic energy. Figure 1(b) suggests that the present analysis with second-order accuracy of the conservation error is useful for investigating higher-order turbulence statistics.

This study then compiles the results shown in Figure 1(b) in a mathematical form, for which the following relations are used:

$$\tilde{k} = k + k$$ and $$\langle k^2 \rangle = \langle (\tilde{k})^2 \rangle - K^2.$$  \hspace{1cm} (6)

Here, $\tilde{k}$, $K$, and $k$ are instantaneous kinetic energy, mean kinetic energy (defined as $K = \langle \tilde{k} \rangle$) and fluctuation of kinetic energy. Another relation is given as follows:

$$\langle k^2 \rangle = \langle (\tilde{k})^2 \rangle - K^2 = (F - 1) K^2,$$ \hspace{1cm} (7)
where $F$ is flatness factor. The above equation gives the following simplified equation:

$$
\frac{\langle k^2 \rangle}{\langle k^2 \rangle|_{t=0}} = \frac{[(F - 1)/(F - 1)]}{(K^2 / K|_{t=0})^2} = \frac{K^2}{K|_{t=0}}^2.
$$

(8)

Therefore, using conservation error $\varepsilon(t)$, conservation error of mean kinetic energy $K$ and the intensity of kinetic energy are given as follows:

$$
K / K|_{t=0} = 1 + \varepsilon(t) \text{ and } \frac{\langle k^2 \rangle}{\langle k^2 \rangle|_{t=0}} = (1 + \varepsilon(t))^2 = 1 + 2 \varepsilon(t) + \ldots.
$$

(9)

Therefore, the linear gradient of conservation error in the intensity of kinetic energy has twice the magnitude as that of the mean kinetic energy itself.

3. Analysis using a turbulent channel flow

3.1. Numerical methods for simulating the turbulent channel flow

Based on the results of the validation of the conservation error using an inviscid flow, this study further validates the present analysis using an actual turbulent flow, as shown in Figure 2. Turbulent channel flows have been widely used in previous studies (e.g. [19]) to validate numerical analyses. The turbulent channel flow in this study is driven by a constant pressure gradient. Here, the friction Reynolds number $Re_\tau(=u_\tau h/\nu)$ is set to 180, where $u_\tau$, $h$ and $\nu$ are the friction velocity, the half width of the channel, and kinematic viscosity, respectively. This value of the friction Reynolds number is the same as that in a previous study [17], to which the present result is compared. The mean velocity and the Reynolds stress are used in this validation method. The root mean square (RMS) value of the velocity fluctuation is also calculated in the validation process, because RMS components of the velocity fluctuation are not included in the Reynolds equation of the mean flow.

The present analysis code of Case 2 simulates the turbulent channel flow. Here, the spatial derivative is discretised using the second-order central difference scheme. The convection and pressure terms are temporally integrated by the third-order Runge-Kutta method, while the viscous terms are temporally integrated using the Crank Nicholson scheme. The Poisson equation at the fractional step is directly solved using Fourier transform and the TDMA scheme. The number of grid points $N_x \times N_y \times N_z$ are, $N_x \times N_y \times N_z = 256 \times 128 \times 128$. Here $N_x$, $N_y$, and $N_z$ are the number of grid points for the streamwise, transverse, and spanwise directions, respectively. For the computational domain size normalised by the half width, $L_x \times L_y \times L_z$ is set to $L_x \times L_y \times L_z = 5 \times 2 \times 8 \times 7$. The size of computational domain is sufficiently large, based on previous studies [19]. The grid widths normalized by the viscous length for streamwise and spanwise directions, $\Delta x^+$ and $\Delta z^+$, are 11.0 and 5.0, respectively.

**Figure 2.** Schematics of turbulent channel flow, computational domain and boundary condition: Here, boundary conditions for directions parallel to the wall are set to a periodic condition.
Figure 3. Validation results using the turbulent channel flow: Results of streamwise mean velocity $u^+$ shown in (a); Results of shear stress are shown in (b).

Figure 4. Validation results using rms values of velocity fluctuation; RMS values of streamwise velocity fluctuation, transverse velocity fluctuation, and spanwise velocity fluctuation, used to validate the present simulation, are shown in (a), (b), and (c), respectively.
3.2. Validation results and discussion

Figure 3(a) shows the transverse profiles of the streamwise mean velocity. As shown in the figure, the result obtained by the present analysis is similar to that of the previous study, which is also indicated by the solid line in the figure. This similarity is found over the transverse computational region. The similarity of the present streamwise mean velocity around the free stream shows that the present value of the wall friction coefficient is also similar to that of the previous study.

Figure 3(b) shows results on the transverse profile of shear stresses. The total shear stress is derived as the sum of Reynolds stress and viscous stress. The viscous stress is determined with the help of the transverse derivative of a profile of streamwise mean velocity. The profile of the Reynolds stress in the present analysis is in good agreement with that of the previous study. As also shown in the figure, the total stress, expressed as the sum of Reynolds stress and viscous stress, agrees with the linear function, which is derived as an analytical result. This agreement indicates that the turbulent channel flow with a constant pressure gradient is accurately simulated in the present analysis. The streamwise mean velocity and Reynolds stress are included in the Reynolds equation representing the mean flow. The results in Figure 3 show that the mean flow obtained by the present analysis agrees well with that of previous studies. Therefore, the results shown in Figure 3 validate the present analysis from the viewpoint of the mean flow.

Figure 4 shows the calculated RMS values of velocity fluctuation for the streamwise direction, transverse direction and spanwise direction, respectively. The three intensities of the velocity fluctuations are not included in the Reynolds equation to describe the mean flow. Therefore, three RMS values of velocity fluctuation are needed to validate the turbulent flow field obtained in this analysis sufficiently. As shown in this figure, the three transverse profiles obtained in this study are in good agreement with those of a previous study. These agreements indicate that the present numerical simulation is able to analyse the velocity field of turbulent channel flow with sufficient accuracy.

3.3. Analysis using turbulent channel flow under coarse grid conditions

If conservation error of the kinetic energy is sufficiently small, low-order turbulence statistics may be obtained with relatively good accuracy, even if a coarse grid is used. The following computational conditions are used to examine the dependence of the present numerical simulation on computational grid: \( N_x \times N_y \times N_z = 16 \times 128 \times 16 \), \( N_x \times N_y \times N_z = 32 \times 128 \times 32 \), and \( N_x \times N_y \times N_z = 64 \times 128 \times 64 \).
where the size of the computational domain is the same for all the grid conditions. Here, the grid resolution for the transverse direction is the same for different computational grid conditions.

Figure 5(a) shows the dependence of the streamwise mean velocity on the grid system. As shown in the figure, the streamwise mean velocity obtained under the computational conditions of \( N_x \times N_y \times N_z = 16 \times 128 \times 16 \) and \( N_x \times N_y \times N_z = 32 \times 128 \times 32 \) deviates from those of the previous study. On the other hand, the mean velocity under the computational conditions of \( N_x \times N_y \times N_z = 64 \times 128 \times 64 \) is relatively similar to that of the previous study. These results are qualitatively consistent with those of a previous study. Figure 5(b) shows the transverse profiles of shear stresses for the computational condition of \( N_x \times N_y \times N_z = 64 \times 128 \times 64 \). As shown in the figure, Reynolds stress can be calculated with sufficient accuracy even when the coarser grid is used. As shown in the previous study, a sufficiently fine computational grid is required when other turbulence statistics are to be investigated.

4. Conclusions
A numerical analysis method under a collocated grid system has significant needs to simulate incompressible turbulent flow, such as the flow around an air foil with flow separation. The purpose of this study is to develop a numerical analysis method for an incompressible flow, with higher-order kinetic energy conservation characteristics, under a collocated grid, and to validate it. As opposed to the numerical analysis under a staggered grid, there is a limit to the improvement of the characteristics of conservation of kinetic energy under a collocated grid. By focusing on a time integration scheme, an accuracy of the second-order for characteristics of conservation of kinetic energy, with respect to time increment, is realised in this numerical analysis. In this paper, a method for improving the characteristics of conservation of kinetic energy to the second-order accuracy was outlined in the section of numerical analysis. The latter was further validated using an inviscid fluid analysis and a turbulent channel flow. The analysis using an inviscid flow was required in this study because the kinetic energy is analytically preserved in the inviscid flow.

In this study, conservation characteristics of kinetic energy were validated in a three-dimensional periodic inviscid flow. The validation indicated that the conservation error of kinetic energy has a second-order accuracy with respect to time increments. This result was discussed using higher-order statistics, such as the intensity of the kinetic energy fluctuation. The analysis was further validated using a turbulent channel flow. The results of the simulation were in good agreement with those of a previous study in terms of streamwise mean velocity, shear stresses, and RMS value of the components of velocity fluctuation. Furthermore, the present analysis was validated under computational conditions of coarser grid systems, where the influence of the conservation error of the kinetic energy was more significant. The streamwise mean velocity and shear stresses could be obtained with relatively sufficient accuracy, even using a coarser grid system. For future works, the present numerical analysis should be applied to simulate flow around an air foil with flow separation.

Acknowledgements
The authors acknowledge Professor Y. Morinishi (Nagoya Institute of Technology) for his valuable comments on this study. This work was supported in part by the Japanese Ministry of Education, Culture, Sports, Science and Technology through Grants-in-Aid (Nos. 17K06160, 18K03932, and 18H01369).

References
[1] Suzuki H, Mochizuki S and Hasegawa Y 2016 Validation scheme for small effect of wind tunnel blockage on decaying grid-generated turbulence J. Fluid Sci. Tech. 11 JFST0012
[2] Suzuki H, Hattori S, Mochizuki S and Hasegawa Y 2016 Basis study on the output-voltage variation of constant-temperature anemometry at freestream velocity Adv. Exp. Mech. 1 36-41
[3] Suzuki H, Fujita K and Mochizuki S 2017 Validation scheme for effects of wind-tunnel blockage for spatially varying acceleration parameter Adv. Exp. Mech. 2 53-58
[4] Suzuki H, Mochizuki S and Hasegawa Y 2018 Numerical-based theoretical analysis on effects of weak fluid acceleration of free-stream due to wind-tunnel blockage on grid-generated turbulence Flow Meas. Instrum. 62 1-8

[5] Morinishi Y, Lund T S, Vasilyev O V and Moin P 1998 Fully conservative higher order finite difference schemes for incompressible flow J. Comput. Phys. 143 90-124

[6] Morinishi Y, Vasilyev O V and Ogi T 2004 Fully conservative finite difference scheme in cylindrical coordinates for incompressible flow simulations J. Comput. Phys. 197 686-710

[7] Suzuki H, Nagata K, Sakai Y and Hayase T 2010 Direct numerical simulation of turbulent mixing in regular and fractal grid turbulence Phys. Scr. T142 014065

[8] Suzuki H, Nagata K, Sakai Y, Hayase T, Hasegawa Y and Ushijima T 2013 Direct numerical simulation of fractal-generated turbulence Fluid Dyn. Res. 45 061409

[9] Rhie C and Chow W 1983 Numerical study of the turbulent flow past an airfoil with trailing edge separation AIAA J. 21 1525-32

[10] Mary I and Sagaut P 2002 Large eddy simulation of flow around an airfoil near stall AIAA J. 40 1139-45

[11] Kajishima T, Ohta T, Okazaki K and Miyake Y 1998 High-order finite-difference method for incompressible flows using collocated grid system JSME Int. J. Ser. B 41 830-839

[12] Meier H F, Alves J J N and Mori M 1999 Comparison between staggered and collocated grids in the finite-volume method performance for single and multi-phase flows Comput. Chem. Eng. 23 247-262

[13] Ushijima S, Takemura M and Nezu I 2002 Investigation on computational scheme for mac methods with collocated grid system (in Japanese) Doboku Gakkai Ronbunshu 719 11-19

[14] Felten F N and Lund T S 2006 Kinetic energy conservation issues associated with the collocated mesh scheme for incompressible flow J. Comput. Phys. 215 465-484

[15] Morinishi Y 1999 An Improvement of collocated finite difference scheme with regard to kinetic energy conservation (in Japanese) Trans. JSME Int. Ser. B 65 505-512

[16] Komen E, Shams A, Camilo L and Koren B 2014 Quasi-DNS capabilities of OpenFOAM for different mesh types Comput. Fluids 96 87-104

[17] Moser RD, Kim J and Mansour NN 1999 Direct numerical simulation of turbulent channel flow up to Reτ = 590 Phys. Fluids 11 943-945

[18] Morinishi Y, Ichikawa A, Okamura T and Nakabayashi K 2000 Effects of the order of accuracy of finite-difference method and grid resolution on LES of wall turbulence (in Japanese) Trans. JSME Ser. B. 66 1750-57

[19] Suzuki H, Nagata K, Sakai Y, Hayase T, Hasegawa Y and Ushijima T 2013 An attempt to improve accuracy of higher-order statistics and spectra in direct numerical simulation of incompressible wall turbulence by using the compact schemes for viscous terms Int. J. Numer. Meth. Fluids 73 509-522