

On Generation of Synthetic Initial Flow Field for Simulating Turbulent Eddies

Dochan Kwak and Cetin Kiris
NASA Advanced Supercomputing Division
NASA Ames Research Center
M/S 258-5
Moffett Field CA 94035-1000
dochan.kwak@nasa.gov

NAS Technical Report NAS-2016-02
July 2016

ABSTRACT

To simulate turbulent eddies, initial and interface flow fields need to be synthesized consistent with the algorithm being employed. In this report, the turbulent kinetic energy conservation issue is discussed followed by a presentation of the detailed steps to synthesize an initial flow field for computing turbulent eddies.

1. Introduction

High-fidelity flow simulations involving complex geometry and flow conditions have become routine practices in aerospace sciences and engineering, especially for obtaining steady-state solutions. For unsteady flow simulations, often associated with separated flows, turbulence modeling has been a major challenge. As the flow analysis involving advanced flow devices and operating conditions become increasingly sophisticated, turbulent eddy simulations that include significant scales of motion, especially scales relevant to engineering interests, have become desirable.

For practical turbulent eddy simulations, even with the increased computer capability, the large eddy simulation (LES) is more realistic than the direct numerical simulation (DNS) of turbulence. The LES approach reduces the modeling requirements at the expense of increased computing cost compared to Reynolds-averaged Navier-Stokes (RANS) approach. The LES approach requires initial and boundary condition procedures consistent with LES algorithm being used which may differ from RANS approach.

Instead of creating new flow solver codes and software, a large number of legacy codes and commercial software are now available for applications. In general, it is a common practice to extend these legacy codes to the simulation of turbulent eddies and unsteady separated regions. Therefore, it would be prudent to assess the validity of this practice. In the current report, we intend to review the energy conservation issues in extending legacy codes to turbulent eddy simulations. And then we will discuss how to synthesize turbulent flow fields as initial

conditions. Similar procedures can be used to generate boundary and interface conditions in multi-zone simulations. This will be done using an incompressible Navier-Stokes formulation.

Even though some details of the incompressible Navier-Stokes (INS) formulation, solution algorithms and application procedures have been discussed in the monograph by Kwak and Kiris [1], some specifics of turbulent eddy simulations are not fully expanded. In the current report, we will discuss some issues not fully covered in our book and thus this article can be considered as a supplement to the monograph.

Various approaches in mathematical formulation and associated numerical methods for incompressible flows have been developed and applied to date. There are books and review articles covering many of these methods, for example, Kwak and Kiris [1], Drikakis [2], Hafez [3], Ferziger [4], Kwak et al. [5]. Readers are referred to these materials for the fundamentals of incompressible flow simulation procedures and applications.

2. Some computational aspects of turbulent eddy simulation

For time-dependent turbulent flows, LES offers the possibility of modeling only the small scale eddies, and thus minimizes tuning the turbulence models as required in the RANS approach. When solid boundaries are involved, the mesh requirement is very high for LES. Therefore, a wall modeled LES (WMLES) or a hybrid RANS-LES became a realistic modeling practice for wall-bounded flows.

One important issue in LES is whether the numerical scheme being used conserves turbulent kinetic energy. Even though the kinetic energy conservation issue is relevant to compressible flow simulations as well, an INS formulation is convenient to investigate this aspect since the density fluctuations are not present in the incompressible flow formulation. Thus, the INS equations are used in this report.

For LES we need to generate the initial flow field which exhibits physical turbulence statistics of the flow being simulated as much as possible. Similarly at the interface of the RANS-LES hybrid model, we need to synthesize the interface flow field to be compatible to both RANS and LES computations. An initial flow field can be started with arbitrary conditions and continue computing until a realistic turbulent flow field is developed numerically. However, this approach will be computationally expensive. Another possibility is to generate the flow field from a given energy spectrum, $E(K)$, such as the one generated from empirical data or a reasonable analytic expression derived from empirical data. Once the initial flow field is synthesized, a time advancing scheme should be implemented which conserves the turbulent kinetic energy.

Regarding current practices of extending legacy codes to the LES region, we need to determine whether the algorithm used in the legacy codes is adequate in the LES region. For example, upwinding schemes often used in legacy codes need to be compared to energy conserving central differencing schemes to confirm that the magnitude of kinetic energy dissipation due to upwinding does not deteriorate large-scale motions. The magnitude of inaccuracy stemming

from grid quality needs to be assessed relative to the error due to numerical schemes as well. Some specifics are discussed next.

Synthesizing the initial flow field, inflow and outflow conditions are all realistic concerns in simulating turbulent eddies. We will present a step-by-step procedure for synthesizing a flow field which exhibits realistic turbulent statistics. Even though this procedure may have been well known to seasoned CFD researchers, we intend to give a convenient guide to those not quite familiar with the turbulent eddy simulation.

2.1 Issues related to extending legacy codes to LES region

In current practices, legacy CFD codes developed primarily for obtaining steady-state solutions are often extended to unsteady turbulent eddy simulation whether it be a hybrid RANS-LES, wall-modeled, or entirely LES simulation.

Where and why kinetic energy conservation could be important?

For turbulent eddy simulations using LES, grid resolution is high and numerical dissipation down to the resolvable scale should be minimized. However, for RANS computations, numerical dissipation is added (in the case of central differencing), or implicitly included in flux computations (in up-winding formulations) to achieve stability. In many legacy RANS codes for 3D applications, up-wind biased schemes are used for flux computations. In such cases, one needs to ask the following specific question:

Are current algorithms in the legacy codes compatible with the LES approach?

For example, up-winding schemes (or central + dissipation term) designed for obtaining primarily steady-state solution in RANS formulations are often highly dissipative. In LES, differencing schemes are designed to conserve kinetic energy and thus the dissipation is handled by a sub-grid scale model.

As illustrated in Figure 1, application of legacy codes in the LES region with a usual sub-grid scale model may result in excessive total dissipation due to a combined effect of numerical dissipation and the dissipation coming from the subgrid scale model.

When legacy codes are applied, especially to complex flow computations, we need to investigate whether kinetic energy conservation is an issue. Since up-winding (or central + dissipation) can resolve energy containing eddies, for those cases where fine grids are utilized to increase the resolution of flow dynamics, kinetic energy conservation may not be a significant concern. Therefore, for turbulent eddy simulation it would be prudent to examine spatial differencing schemes in legacy codes relative to kinetic energy conservation, or define the range of applicability where current numerical schemes are acceptable.

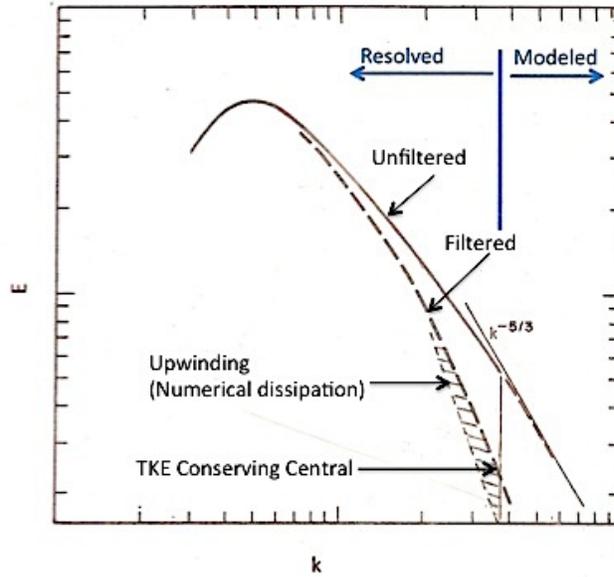


Figure 1. A schematic of kinetic energy spectra: eddies computed using 1) up-winding scheme vs. 2) central-differencing scheme: Shaded region represents the energy dissipated by upwinding compared to kinetic energy conserving central differencing scheme. In the figure, E = energy spectrum, k =wave number

Specifically, we need to:

- compare unwinding schemes vs. various central differencing schemes (staggered, generalized collocated schemes) relative to kinetic energy conservation,
- define the region where upwinding schemes (even high order) are deemed unsuitable for use in LES, (e.g. - Mittal, et al. [6] studied the suitability of upwinding schemes for LES.)
- determine where upwinding schemes may be suitable for producing acceptable results for lower order statistics i.e. Reynolds stress, velocities, forces and moments etc. This may be the situation when the level of energy dissipated due to upwinding is low relative to the total flow energy.

In addition, we need to consider the following potential issues:

- For separated and/or unsteady flows, often a hybrid approach is adopted where RANS is used for wall region, switching over to LES away from the wall. Then, numerical schemes for the wall region and the separated region may not be compatible.
- If RANS codes are used in eddy simulations, small scale eddies can be dissipated too much and the results may be significantly different from those obtained based on LES (see the sketch in Figure 1). In this case, the kinetic energy conservation issue needs to be assessed.

2.2 Kinetic Energy Conservation Issues

In this section, we will review kinetic energy conservation issues related to spatial differencing schemes. The kinetic energy conservation issue has been extensively discussed in meteorological flow simulation (i.e. weather modeling), and a large volume of research has been published in the past. For example, Arakawa [7], Lilly [8] and many others published fundamental methods and turbulent flow simulation related to atmospheric flow. The discussion

in this section stems from these studies specifically extracted for our applications. Our discussion will then be followed by presentation of an example of generating a velocity field starting from a given energy spectrum, $E(K)$. For example, for homogeneous isotropic turbulence the measured spectrum by Comte-Bellot and Corrsin [9] can be used. For free shear flow, a similar procedure can be applied starting from a given energy spectrum (e.g. one derived from von Karman spectrum [10]).

Consider, inviscid, zero pressure gradient, constant density, divergence free flow:

$$\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j} (u_i u_j) = 0 \quad (2.1)$$

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (2.2)$$

Kinetic energy equation: $u_i \cdot (2.1)$

$$u_i \frac{\partial u_i}{\partial t} + u_j \frac{\partial}{\partial x_j} (u_i u_j) = 0 \quad (2.3)$$

Now we will look at the difference form of equations (2.1), (2.2) and (2.3). For convenience the following notations are defined first:

$$\begin{aligned} f_x &= \frac{1}{\Delta} \left[f \left(x + \frac{\Delta}{2} \right) - f \left(x - \frac{\Delta}{2} \right) \right] \\ \bar{f}^x &= \frac{1}{2} \left[f \left(x + \frac{\Delta}{2} \right) + f \left(x - \frac{\Delta}{2} \right) \right] \\ \bar{f}^{2x} &= \frac{1}{2} [f(x + \Delta) + f(x - \Delta)] \\ \bar{f}^{xx} &= \overline{(\bar{f}^x)^x} = \frac{1}{2} \left[\overline{f \left(x + \frac{\Delta}{2} \right) + f \left(x - \frac{\Delta}{2} \right)} \right]^x \\ &= \frac{1}{4} [f(x + \Delta) + 2f(x) + f(x - \Delta)] = f + \frac{\Delta^2}{4} f_{xx} \end{aligned}$$

and the following identities hold

$$\begin{aligned} (fg)_x &\equiv \bar{f}^x g_x + \bar{g}^x f_x \\ \overline{(fg)^x} &\equiv \bar{f}^{2x} \bar{g}^x + \bar{g}^{2x} \bar{f}^x \\ \overline{(\bar{f}^x \bar{g}^x)}_x &\equiv \bar{f}^{xx} \bar{g}^x + \bar{g}^{xx} \bar{f}^x \\ &\equiv \frac{1}{2} \left[\overline{(fg)^x} + f \bar{g}^x + g \bar{f}^x \right] \end{aligned}$$

Now look at the energy conserving difference form of equation (2.1), (2.2), and (2.3).

Note that the second order central differencing scheme is

$$\frac{\partial f}{\partial x} = \overline{f}_x^x + O(\Delta^2) = \frac{f_{i+1} - f_{i-1}}{2\Delta} + O(\Delta^2)$$

and the 4th order scheme can be obtained by Richardson extrapolation

$$\begin{aligned} \frac{\partial f}{\partial x} &= \frac{4}{3}\overline{f}_x^x - \frac{1}{3}\overline{f}_{2x}^{2x} + O(\Delta^4) \\ &= \frac{1}{12\Delta}(f_{i-2} - 8f_{i-1} + 8f_{i+1} - f_{i+2}) + O(\Delta^4) \end{aligned}$$

Then the kinetic energy conserving difference form of (2.1) can be written as

$$\frac{\delta u_i}{\delta t} + \frac{\delta}{\delta x_j} \left(\overline{u_i}^{x_j} \overline{u_j}^{x_j} \right) = 0 \quad (2.4)$$

Take the difference form of kinetic energy equation (2.3), and then summing over all mesh points,

$$\sum u_l \frac{\delta u_l}{\delta t} = -\sum u_l \left(\overline{u_l}^{x-x} \overline{u}^{x-x} \right)_x + v\text{-component} + w\text{-component} \quad (2.5)$$

where u_l is either u , v or w

To show that this conserves the kinetic energy, we need to show that the RHS of equation (2.5) becomes zero for periodic boundary conditions.

Noting that

$$\left(\overline{f}^{x-x} \overline{g}^{x-x} \right)_x = \frac{1}{2} \left[\left(\overline{fg} \right)_x^x + \overline{f} \overline{g}_x^x + \overline{g} \overline{f}_x^x \right] = \frac{1}{2} \left[\overline{f}^{2x} \overline{g}^{2x} + \overline{g}^{2x} \overline{f}^{2x} + \overline{f} \overline{g}_x^x + \overline{g} \overline{f}_x^x \right]$$

RHS terms of equation (2.5) can be expanded as

$$-\sum u_l \left(\overline{u_l}^{x-x} \overline{u}^{x-x} \right)_x = -\sum \frac{u_l}{2} \left\{ \left[\left(\overline{u_l u} \right)_x^x + u \left(\overline{u_l} \right)_x^x \right] + \left[u_l \overline{u_x}^{x-x} \right] \right\}$$

The first bracketed term on the RHS becomes zero for periodic boundary, and the second term becomes zero for incompressible flow.

The above energy conserving scheme or a similar one can be compared to the up-winding scheme typically used in legacy RANS codes. The decaying box turbulence case sketched in Figure 1 can be used as a benchmark problem where periodic boundary conditions can be imposed. To perform decaying process, it is first required to generate initial turbulent flow field

that matches with the experimental energy spectrum. Some details of this synthetic flow field generation procedure is explained next.

2.3 Generation of synthetic initial turbulent flow field

For simplicity a uniform Cartesian grid will be used here. Since experimental energy spectrum data are available for box turbulence and an analytical expression for free shear flow is available derived from empirical data, we will start with a given energy spectrum.

First, the correlation tensor for Reynolds stress is written as

$$\bar{R}_{ij}(\vec{r}) = \overline{u_i(\vec{x}, t) u_j(\vec{x} + \vec{r}, t)}$$

The Fourier transform of \bar{R}_{ij} produces the spectrum tensor ϕ_{ij} :

$$\phi_{ij}(\vec{K}) = \frac{1}{(2\pi)^3} \int_{-\infty}^{\infty} \exp(-i\vec{K} \cdot \vec{r}) R_{ij}(\vec{r}) d\vec{r} \quad (2.6)$$

$$R_{ij}(\vec{r}) = \int_{-\infty}^{\infty} \exp(i\vec{K} \cdot \vec{r}) \phi_{ij}(\vec{K}) d\vec{K}$$

Note that

$$R_{ii}(0) = \overline{u_i u_i} = 3u^2 = \int_{-\infty}^{\infty} \phi_{ii}(\vec{K}) d\vec{K}$$

$$E(K) = \frac{1}{2} \oint \phi_{ii}(\vec{K}) d\sigma$$

where $d\sigma$ is the surface element of the spherical shell of radius K .

The spectrum function ϕ_{ij} of the filtered field is

$$\phi_{ij}(k_n) \equiv \langle \widehat{u}_i(k_n) \widehat{u}_j^*(k_n) \rangle$$

where \widehat{u}_i is discrete Fourier Transform of \bar{u}_i and superscript * denotes complex conjugate.

The filtered three-dimensional energy spectrum, $\bar{E}(k)$, can be written as

$$\bar{E}(k) = 2\pi k^2 \phi_{ii}(k) \quad (2.7)$$

$$\bar{E}(k_n) \equiv 2\pi k_n^2 \langle \widehat{u}_i(k_n) \widehat{u}_j^*(k_n) \rangle \quad (2.8)$$

For a given energy spectrum, LHS of Eq. (2.8), discrete Fourier Transformed velocity field can be obtained. Here, \widehat{u}_i needs to be chosen to satisfy incompressibility in grid space.

Continuity in grid space:

$$\bar{u} = \sum_n \hat{u}_i(\underline{k}) \exp(i\underline{k} \cdot \underline{x})$$

To illustrate the continuity in grid space, use a second order central differencing scheme

$$\frac{\delta \bar{u}}{\delta x} = \frac{1}{2\Delta} (\bar{u}_{k+1} - \bar{u}_{k-1})$$

Fourier transform of the above is

$$\frac{\delta \hat{u}}{\delta x} = \frac{1}{\Delta} \sin(\Delta k_1) \hat{u} \equiv k_1'' \hat{u} \quad (2.9)$$

Then $D\bar{u} = 0$ in Fourier space can be expressed as

$$k_1'' \hat{u}_i = 0 \quad (2.10)$$

For any \underline{k} , \underline{k}'' can be obtained by equation (2.9). Then \hat{u} can be selected on a perpendicular plane to \underline{k}'' in \underline{k} -space as indicated in Eq (2.10).

How to achieve statistical isotropy:

Once the magnitude of $\hat{u}_j(k_n)$ is obtained from Eq (2.8), a divergence free velocity field can be obtained by satisfying Eq (2.10). A procedure to satisfy statistical isotropy can be as follows;

- the real and imaginary part of \hat{u}_j must be chosen randomly
- first choose a unit vector A on a plane perpendicular to \underline{k}'' by turning a random angle from a reference frame
- choose another unit vector B in a same way
- select a random angle, θ , and define
 $a = \cos(\theta)$, $b = \sin(\theta)$
- then the real and imaginary part of \hat{u}_j can be selected as below

$$\hat{u}_j(k_n) = |\hat{u}_i(\underline{k}_n) \hat{u}_i^*(\underline{k}_n)|^{1/2} (aA_j + ibB_j) \quad (2.11)$$

3. An illustration of synthetic turbulent flow field generation procedure

In this section detailed steps for synthesizing turbulent flow field commensurate with a given energy spectrum are illustrated.

Select an energy spectrum:

First, choose an energy spectrum either from an experiment or a mathematical approximation such as the von Karman spectrum [9]. In the current example, one spectrum from the classical experiment by Comte-Bellot and Corrsin [9] can be considered as an initial energy spectrum.

For numerical computation, we need to generate filtered field, $\bar{E}(k)$, from the empirical spectrum, $E(k)$. Let's define the filtered field, \bar{f} , as below:

$$\bar{f}(\underline{x}) = \int G(\underline{x} - \underline{x}') f(\underline{x}') d\underline{x}'$$

This filtering process filters out small-scale motion while preserving large scale energy containing motion. Generally, the following Gaussian filter is preferred to “box” filter.

$$G(\underline{x} - \underline{x}') = \left\{ \sqrt{\frac{\gamma}{\pi}} \frac{1}{\Delta_A} \right\}^3 \exp\left\{ -\gamma(\underline{x} - \underline{x}')^2 / \Delta_A^2 \right\}$$

where $\gamma = \text{const}$: defines filter shape
 Δ_A : filter length scale

Then the filtered flow field becomes

$$\bar{u}(\underline{x}) = \left\{ \sqrt{\frac{\gamma}{\pi}} \frac{1}{\Delta_A} \right\}^3 \int_{-\infty}^{\infty} \underline{u}(\underline{x}') \exp\left\{ -\gamma(\underline{x} - \underline{x}')^2 / \Delta_A^2 \right\} d\underline{x}'$$

The Fourier transform of this is

$$\hat{\bar{u}}(\underline{x}) = \hat{u}(k) \exp\left(-\frac{\Delta_A^2}{4\gamma} k^2 \right)$$

We can then derive the following filtered energy spectrum

$$\bar{E}(k) = E(k) \exp\left(-\frac{\Delta_A^2}{2\gamma} k^2 \right) \quad (3.1)$$

Calculate amplitude of velocity, $\hat{u}_i(k_n)$:

Once we select $E(k)$, then we can calculate the filtered energy spectrum from Eq. (3.1).

First we need to choose the shape parameter, γ , then Δ_A .

If we set $\gamma = 6$, the second order term for $\overline{u_i u_j}$ using Gaussian filter results in the same second order term using a box filter (sometimes called as “Top Hat” filter).

Then the filtered energy spectrum, $\bar{E}(k)$, becomes

$$\bar{E}(k) = E(k) \exp\left(-\frac{\Delta^2}{12} k^2\right) \quad (3.1a)$$

Next we need to choose, Δ_A .

For a mesh system with a width, Δ , the smallest scale wave (2Δ wave) has a wave number, π/Δ , and the wave number for the largest scale motion is $\frac{2\pi}{N\Delta}$.

Now we need to choose the mesh number, N , and the size, Δ , such that the grid system chosen captures as much of the turbulent energy as possible. At the same time it should capture the “inertial sub-range.”

For the mesh system, (n_1, n_2, n_3) ,

$$k = \frac{2\pi}{N\Delta} \sqrt{n_1^2 + n_2^2 + n_3^2}$$

For N^3 mesh the integer, n_i , ranges, from $\left(\frac{1}{2N}\right)$ to $\left(\frac{1}{2N}-1\right)$.

Then choose $\hat{u}_i(k_n)$ following the procedure described in Section 2.3.

To obtain the real value for \bar{u}_i

$$\hat{u}(-k_n) = \hat{u}^*(k_n)$$

The imaginary part of k_n cancels the imaginary contribution of $-k_n$ component.

Calculate velocity, $\bar{u}(\underline{x})$:

Once we selected the velocity field in Fourier space, the velocity field in physical space can be obtained by inverse Fourier transform. To utilize fast Fourier transform (FFT), the maximum number of grid points, N , has to be 2^m . Therefore, in selecting \hat{u} in spectral space, we can set

$$n_i = -\left(\frac{N}{2}-1\right) \sim \left(\frac{N}{2}-1\right)$$

This means that we are losing the value at $-\left(\frac{N}{2}\right)$.

The initial flow field generated as described above will have slightly lower kinetic energy at high wave number than the energy spectrum we started with. However, this may be acceptable for practical purposes. Thus we obtain the following velocity field:

$$\bar{u}(\underline{x}) = \sum_{-\left(\frac{N}{2}-1\right)}^{\left(\frac{N}{2}-1\right)} \hat{u}(k_n) \exp(i\mathbf{k}_n \cdot \underline{x})$$

We need to choose a mesh system with sufficient resolution for the desired small scale eddies.

Time-step Size Estimate:

The time step size can then be selected which makes spatial and temporal truncation errors in comparable magnitudes. A specific example is described next as an illustration on how time step can be determined for computing evolution or decay of turbulent eddies.

When a 4th order spatial differencing is used, the magnitude of the truncated term can be estimated as

$$\text{Order(Spatial Truncated Term)} \sim \frac{\Delta^4}{120} \frac{\partial^5 u^2}{\partial x^5} \sim \frac{\Delta^4}{120} \frac{U^2}{\lambda^5} \quad (3.2)$$

If a 2nd order time scheme is used, then the truncation term would be

$$\text{Order(Time Truncated Term)} \sim \frac{\Delta t^2}{6} \frac{\partial^3 u^2}{\partial t^3} \sim \frac{\Delta t^2}{6} \frac{U}{\tau^3} \quad (3.3)$$

Here, λ is the Taylor microscale defined by

$$\overline{\left(\frac{\partial u_1}{\partial x_1}\right)^2} = \frac{\overline{u_1^2}}{\lambda^2} = \frac{U^2}{\lambda^2} \quad (3.4)$$

This estimate is based on isotropic turbulence idea since small scale eddies in large Reynolds number flows are isotropic. Even though the Taylor microscale does not represent any physical length scale, it is often used as a dividing scale between integral and viscous range. Now we want to establish U, λ and τ relationship for the time step Δt estimate.

In isotropic turbulence, the dissipation rate, ε , can be estimated to be

$$\varepsilon = 2\nu \overline{S_{ij} S_{ij}} = 15\nu \overline{\left(\frac{\partial u_1}{\partial x_1}\right)^2} = 15\nu \frac{U^2}{\lambda^2} \quad (3.5)$$

Following Kolmogorov, the dissipation time scale, τ , can be estimated as

$$\tau = \left(\frac{\nu}{\varepsilon}\right)^{1/2} \quad (3.6)$$

From the above equations, we can obtain the following relation:

$$\frac{U}{\lambda} = 0.26 \left(\frac{\nu}{\varepsilon}\right)^{1/2} = 0.26 \tau^{-1}$$

Equating spatial and time truncated terms in Eq. (3.2) and (3.3),

$$\frac{\Delta^4}{120} \frac{U^2}{\lambda^5} = \frac{\Delta t^2}{6} \frac{U}{\tau^3} = \frac{\Delta t^2}{6} \frac{\partial^3 u^2}{\partial t^3} \left(\frac{U}{0.26\lambda}\right) U$$

Then we get the following relation between Δt and Δ :

$$(\Delta t)^2 \cong \frac{(0.26)^3}{20} \frac{\Delta^4}{(\lambda U)^2}$$

We can get an estimate of time step once we have selected a mesh size and an energy spectrum for generating the initial flow field. This time step may be used to carry on a few steps of computation to make the flow field adjust itself once synthetically generated.

4. Remarks on Pacing Issues for General Applications of LES

In this report we revisited the relevance of kinetic energy conserving schemes to turbulent eddy simulation especially for general engineering-level flow analysis, and presented detailed steps for generating an initial LES flow field synthetically for turbulent eddy simulation.

There are a number of pacing issues in applying LES in general flow analysis. As we discussed, the adequacy of extending legacy codes in LES region needs to be verified. In particular, in real-world applications, the magnitude of energy loss due to dissipation terms in legacy codes has to be insignificantly lower than the dissipation due to sub-grid scale models. This may be algorithm and grid dependent.

The primary difficulty yet to be resolved in LES is related to wall-bounded flows. Since the number of mesh points required for resolving wall region is huge (the estimate of mesh requirements varies among estimators), wall-modeled LES (WMLES) has been utilized by numerous researchers to make LES practically usable. Essentially, the accuracy of WMLES is dictated by the accuracy of the wall model.

To alleviate the complications, implicit LES (ILES) has been advocated by some researchers, where wall modeling is replaced by numerical dissipation which depends on the mesh. This is similar to direct numerical simulation idea where all significant scales are assumed to be computed requiring no modeling. The mesh used in ILES has to be constructed such that the mesh related dissipation does not contaminate large-scale motion significantly. Naturally construction of the right mesh system, especially near the walls, becomes important. This approach needs further investigation or some best practice guidelines need to be established.

In summary, LES offers conceptually a good possibility for simulating turbulent eddies without statistical modeling. However, there are a number of issues in practice requiring further development. Synthetic initial flow field generation we presented in this report can be a part of the general recipe for turbulent eddy simulation.

References

1. Kwak, D., and Kiris, C., *Computation of Viscous Incompressible Flows*, Springer, November 2010.
2. Drikakis, D., Rider, W.: *High-Resolution Methods for Incompressible and Low-Speed Flows*. (Springer, 2004)

3. Hafez, M.: *Numerical Simulations of Incompressible Flows*, ed. (World Scientific, 2002)
4. Ferziger, J.H.: Incompressible turbulent flows. *J. Comp. Phys.*, **69**, 1-48 (1987)
5. Kwak, D., Kiris, C., and Housman, J., "Implicit Methods for Viscous Incompressible Flows," *Computers & Fluids* 41, Issue 1, 51-64, February 2011.
6. Mittal, R. and Moin, P., "Suitability of Upwind-Biased Finite Difference Schemes for Large-Eddy Simulation of Turbulent Flows," *AIAA J.*, vol. 35, No 8, p 1415-1417 (1997).
7. Arakawa, A., "Computational Design of Long-Term Numerical Integration of the Equations of Fluid Motions: I. Two-dimensional Incompressible Flow," *J. Comp. Phys.* vol. 1, 1, 1966.
8. Lilly, D.K., "On the Computational Stability of Numerical Solutions of Time-dependent Non-linear Geophysical Fluid Dynamics Problems," *Mon. Weather Rev.*, vol. 93, 1, p11, 1965.
9. Comte-Bellot, G., and Corrsin, G., "Simple Eulerian time correlation of full-and narrow-band velocity signals in grid-generated, 'isotropic' turbulence." *Journal of Fluid Mechanics* 48.02 273-337: 1971.
10. von Kármán, Theodore, "*Progress in the Statistical Theory of Turbulence*," *Proceedings of the National Academy of Sciences* **34** (11): 530-539, 1948