Best Practices for Large Eddy Simulations (LES) in ANSYS FLUENT

Note: This document provides best practice guidelines that highlight various methods that can be used within ANSYS products. These guidelines are meant to be used in conjunction with the information provided in the ANSYS, Inc. product documentation. For the latest version, please visit the ANSYS Customer Portal.

Introduction

The Large-Eddy Simulation (LES) technique directly resolves the large scale of turbulence (scales which contains most of the turbulent kinetic energy and are responsible for most of the momentum and scalar transport) and models the small scales. The strength of LES is that the large scales (which depend strongly on the geometry of the problem) are fully resolved and only the more universal small scales are modeled. This is in strong contrast with RANS models, where all turbulent scales are modeled with the same hypothesis. Thus LES has the potential of being more accurate than RANS simulations. However, LES is also more demanding than RANS simulations (mesh requirements, solver setup, boundary conditions and knowledge of the physics of the flow, etc).

From the theory, the large and small scales can be separated by a spatial filtering operation. In practice, the filtering operation is implicit and is based on the grid resolution: the LES model filters out the turbulent scales smaller than the grid size. The effects (mainly dissipation and backscatter) of the small scales on the resolved large scales are taken into account by using so-called sub-grid turbulence models.

Key Applications/Physics

LES is valid for all applications and all fluids. Key applications are combustion, mixing, meteorology, etc.

Subgrid models

All the models in ANSYS FLUENT are eddy viscosity models (Boussinesq approximation).

Smagorinsky–Lilly

This model assumes sub-grid scales local equilibrium (i.e. local equilibrium between subgrid scales turbulent kinetic energy production and dissipation and no transport of subgrid turbulent energy).

- Pros: computationally cheap and simple.
- Cons: In practice, the model constants can be different for each flow. The wall behavior is not accurately predicted (no damping effect in the wall region). Transition from laminar to turbulent flows cannot be predicted. The model does not allow energy transfer from the subgrid scales to the resolved scales (a.k.a. backscatter).

WALE

This model uses the same hypothesis as the Smagorinsky-Lilly model. However, the local rotation rate is taken into account to compute the turbulent viscosity. Because small (dissipative) scales are characterized by a high rotation rate, this model is more accurate than the Smagorinsky-Lilly model.

- Pros: the model is computationally cheap, simple, and predicts accurately the flow near-wall behavior (implicit damping effect). The model also predicts transition from laminar to turbulent flows in wall bounded simulations.
- Cons: the model does not allow for non-equilibrium or transport effects of turbulence at the subgrid scales. The model constants can depend on the flow conditions.

Dynamic Smagorinsky

This model assumes subgrid scales turbulent energy local equilibrium. This model also assumes scale similarity between the smallest resolved scales and the subgrid scales are used (i.e. the Kolmogorov spectrum hypothesis is not used at the subgrid scales)

- Pros: This model is more universal because constants are computed dynamically. The model can accurately predict the wall behavior and allow for energy backscatter (i.e. turbulent kinetic energy transfer from small scales to large scales). This model was demonstrated to successfully predict a wide range of flows, including transitional flows.
- Cons: The computation of dynamic constant requires additional computational expense. Wide fluctuations in dynamically computed constants can cause stability issues.

Dynamic Kinetic Energy Transport

This model assumes subgrid scales turbulent energy non-equilibrium and scale similarity.

- Pros: this model addresses the limitations of both the Smagorinsky-Lilly and the Dynamic Smagorinsky models (subgrid turbulent energy local equilibrium). An additional transport equation for subgrid scale kinetic energy (k_{sgs}) allows for the history of k_{sgs} and non-equilibrium effects to be taken into account. The model also accurately predicts energy backscatter.
- Cons: the model requires one additional transport equation (k_{sgs}) and additional explicit filtering operations are performed.

Detached Eddy Simulation (DES)

In the DES approach, the unsteady RANS models are employed in the boundary layer, while the LES treatment is applied to the separated regions. The LES region is normally associated with the core turbulent region where large unsteady turbulence scales play a dominant role. In this region, the DES models recover LES-like subgrid models. In the near-wall region, the respective RANS models are recovered.

DES models have been specifically designed to address high Reynolds number wall bounded flows, where the cost of a near-wall resolving Large Eddy Simulation would be prohibitive. The difference with the LES model is that it relies only on the required resolution in the boundary layers. The application of DES, however, may still require significant CPU resources and therefore, as a general guideline, it is recommended that the conventional turbulence models employing the Reynolds-averaged approach be used for practical calculations.

The DES models, often referred to as the hybrid LES/RANS models combine RANS modeling with LES for applications such as high-Re external aerodynamics simulations. In **ANSYS FLUENT**, three variations of the DES models are available. The DES model can be based on the one-equation Spalart-Allmaras model, or the realizable k-ε model, or the SST k-w model (in the order of complexity, and computational cost). The computational cost of the DES model is less than LES computational cost, but greater than RANS.

- Pros: this model allows the engineer to simulate high Reynolds, massively separated, wall bounded flows at a smaller computational expense than LES.
- Cons: the constants of the model depend on the flow and the type of grid used.

<u>Recommendations</u>

Using the WALE model or Dynamic Smagorinsky model is a good starting point. Consider using DES for high Reynolds number flows and wall bounded flows with “massive” separation. Consider using wall functions for high Reynolds internal flows.

Numerics

Temporal Discretization

- Non-Iterative Time Advancement (NITA) / Fractional Step Method
    - Robust for simple geometry only
    - Use for incompressible or weakly compressible flows
    - Reduce residual tolerance for all equations to 0.0001
    - Use for flow with density changes < 1.2 (i.e. not valid for combustion applications)
- Iterative Time Advancement (ITA) method can be used for cases where NITA is not valid (e.g.: combustion simulations)
    - Second order implicit time advancement
    - SIMPLEC pressure-velocity coupling with large Under-Relaxation factors for all equations

Spatial Discretization

- Flow variables: Bounded Central Difference. Avoid upwind schemes whenever possible because they are too diffusive.
- Scalars variables: Use of upwind schemes (MUSCL, QUICK, or Second-Order) is preferable to avoid local overshoots. Central or Bounded Central Difference scheme should not be used because scalar gradients are often sharper than velocity gradients.
- Use Least Square Node based discretization.

Mesh Requirements

Below are some recommendations about mesh type and mesh resolution:

- Hexahedral mesh elements are recommended for higher accuracy and computational efficiency
- For complex geometry use prism layers on solid surfaces.
- Grid stretching should be lower than 5%. All the subgrid models are grid based models (i.e. based on the local grid size). Thus, any large grid discontinuities will be accompanied by turbulent viscosity jump which can artificially dissipate small turbulent structures and induce numerical instabilities.
- The integral length scale, which measures the size of energy containing eddies, should be resolved with several grid points (i.e. 40 to 50). This integral length scale can be estimated from correlation or obtained by post-processing a k-ε RANS simulation. The integral length (L) scale is easily computed by the following formula: $L = {k^{3/2}}/{\varepsilon}$.
- Recirculation regions must be well resolved.

Boundary Conditions

<u>Inlets</u>

By definition LES simulations require unsteady inlet boundary conditions. Depending on the case, the impact of these boundary conditions can be either critical or negligible. LES boundary conditions greatly influence internal flows. We recommend using the vortex method or spectral synthesizer methods.

- The Vortex Method is more accurate because coherent structures are generated but also requires realistic inlet conditions (U, k, and ε profiles). Otherwise the boundary conditions can force the flow in an unphysical manner.
- The spectral synthesizer is more flexible but also more random in nature (i.e. the turbulence generated at the inflows is less coherent and can rapidly dissipate).

<u>Outflows</u>

If possible, the outflow boundary condition is a better choice than the constant pressure outlet boundary condition because exiting vortices do not satisfy the constant pressure assumption. For certain acoustics calculations like jet noise, it is recommended to use the non-reflecting boundary conditions available for non reacting flows in the Density-Based solver.

<u>Walls</u>

A wall treatment must be chose. 3 options are available:

- LES: Near Wall Resolving technique (NWR).
- LES: Near Wall Modeling technique (NWM).
- DES for external flows with massive separation.

Near Wall Resolving:

- All the large scales turbulent structures are explicitly resolved. The laminar sublayer is resolved. Hence, the first grid point must be at y+ <=1.
- In this approach, it is recommended to use moderate grid stretching in the stream-wise and span-wise directions because it is critical to resolve the energy containing eddies in the near wall region. Because the turbulent structures in the wall region are elongated in the stream-wise direction , the grid resolution parallel to walls needs to be such that $\Delta x^{+} \sim 50$ and $\Delta z^{+} \sim 15$, where x is the stream-wise direction and z the span-wise direction.
- This technique is appropriate for low-Reynolds flows, but too expensive for large Reynolds flows.

Near Wall Modeling

- In this technique, the near wall turbulence is explicitly computed inside the boundary layer, but not to the laminar sub-layer. The first grid point must lie inside the logarithmic layer. The wall shear stress is modeled using to wall functions. The recommendations are:
    - Wall function: Werner-Wengle. To access this wall function, use the following Text-User Interface command: `/models/viscous/ near-wall-treatment/werner-wengle-wall-fn?`
    - Wall resolution: y+ ~20-150.
    - Grid stretching: $\Delta x^{+} \sim 100 - 600$ and $\Delta z^{+} \sim 100 - 300$
- This technique is appropriate for high Reynolds number flows and massively separated flows. This model usually fails to predict flow separation induced by adverse pressure gradient.

Detached-Eddy Simulation

- The near wall turbulence is not explicitly computed, but fully modeled. Only the mean velocity profile is resolved with a RANS approach.
- Ideally, the first grid point should be at y+=1. Benefits of using DES instead of NWM when y+>>1 is questionable.
- It is possible to use more stretching in the stream-wise and span-wise directions than with LES/NWR or LES/NWM because it is not necessary to resolve eddies located in the wall region.

- It is critical to analyze the location of the LES/RANS interface (post processing). The goal is to be in RANS mode in the wall regions and in LES mode in the free flow. This information can be accessed by post-processing the Relative Length Scale (Display -> Contours -> Turbulence -> Relative Length Scale). The cells were the Relative Length Scale is positive belong to the LES region. The cells were the Relative Length Scale is negative belong to the RANS region.
- DES cannot be used for internal flows because turbulent structures would not be computed accurately (and would be fully damped if y+>1).
- The DES method is less accurate than the NWM technique.
- Because near wall turbulence is not explicitly computed, DES cannot be used for accurate dipole acoustic source computation.

Simulation and Post-Processing

General set-up

- If the pressure signal is of interest, make sure that the reference pressure location in Operating Conditions panel is at a constant pressure level location.
- First perform steady RANS calculation (to obtain the LES initial conditions). Then, use the spectral synthesizer (Text-User Interface: solve/initialize/initialize-instantaneous-velocity). Last, switch to LES and select subgrid model.
- If the acoustic of the problem is not of interest, the goal is to have, in regions of interest, a maximum local CLF (Courant Friedrich Levi) number of 1, based on the convection speed. The CFL number can be accessed, in the Density Based Solver, under Display -> Contour -> Velocity -> Cell Courant Number. The CFL number based on the convection speed can be approximated by $CFL = {(U * dt)}/{\Delta}$, where U, dt and Δ are the local velocity magnitude, the LES time step and the local grid size, respectively.
- If the goal is to resolve the acoustic of the problem, the maximum local CFL number (based on the speed of sound) must be smaller than 1. The CFL number based on the speed of sound can be approximated by $CFL = {(c * dt)}/{\Delta}$, where c, dt and Δ are the local speed of sound, the LES time step and the local grid size, respectively.

Monitoring Statistics - Initial Phase

Because typical LES simulation can be very computationally expensive, it is necessary to carefully follow the initial phase of the simulation in order to detect possible defects ASAP. The objective of this initial phase is to develop the turbulent structures starting from initial (i.e. RANS) conditions and to reach a statistically steady state. The duration of this phase depends on the flow characteristics. The simulation must be run for a total time large enough to allow all turbulent instabilities which develop during this phase to be convected across the region of interest. However the convecting velocities of the turbulent structures and the regions of interest are not always known a priori. This is why it is recommended to run the simulation for at least one flow through time. A flow through time is defined as the ratio of the system dimension over the bulk velocity. During this phase:

- Create and observe monitors at different points of interest (velocity or temperature monitors, for example). When the signal repeats itself, the statistically stable regime is reached.
- For internal flows, it is recommended to track the wall-shear stresses.
- Visualize typical flow structures.

Monitoring Statistics - Statistics Gathering Phase

When the statistical steady state is reached:

- If statistics are required, activate Data sampling in Iterate panel to allow calculation of mean and rms quantities.

- Ideally take statistics for 5-25 flow through time. Mean statistics will converge after 5 flow through time, rms quantities can require 10-20 flow through time.
- If a spectral analysis is required, store time varying signals at given location (for example, the pressure signal) and post-process the signal with the FFT tool.

Animations
For animations, export CFDPost data and create the animation using CFDPost.

Final comments
- If the turbulence decays too fast (Kelvin-Helmoltz structures, streaks, etc.), it is an indication that the mesh resolution is too coarse.
- Mean quantities can be reasonably accurate without having the correct fluctuations (or rms) level. In other words, always post-process both mean and rms quantities.

Summary: Recommended Procedure

1. Compute the mean flow with steady RANS with k-ε model. The RANS solution doesn't have to be fully converged
2. Superimpose the synthesized turbulence on the mean flow. For this, use the TUI command: /solve/initialize/init-instantaneous-velocity
3. Switch to LES, select the subgrid turbulence model
4. Select the solver algorithm (e.g., ITA/NITA, FSM/PISO/SIMPLEC) and the discretization schemes
5. Set the time-step (Dt) and adjust the solver parameters if needed (e.g., URF's, convergence criteria)
6. Set the monitors for relevant global (e.g., forces/moments) and local quantities (e.g., velocity, pressure) of your choice:
    - Solve/Monitors/Force…
    - Solve/Monitor/Surface…
    - Solve/Monitor/Volume…
7. Set the autosave of the data files (e.g., every a few hundred time-steps)
8. Start the transient run and continue until a statistically stationary state is reached by monitoring integral and/or local quantities
9. Save the viewgraphs of your choice for animation (contours of pressure, iso-surfaces of vorticity, second-invariant, etc.)
10. Start sampling the data to compute mean and r.m.s. values
11. Continue sampling for a sufficiently long period of time (several flow-through times)
12. Post-process the results

www.ansys.com

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

Toll Free U.S.A./Canada:
1.866.267.9724
Toll Free Mexico:
001.866.267.9724
Europe:
44.870.010.4456
eu.sales@ansys.com