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

General Tips

1. Mesh
   a. Hex mesh is an ideal choice for accuracy
   b. Avoid abrupt change in grid spacing (cell size) or too aggressive stretching in eddy-congested regions (e.g., boundary layer, near-wake)
   c. Take advantage of hybrid mesh and local mesh refinement capability (e.g. wall boundary layer, near-wake)
2. Spatial discretization
   a. The “node-based gradient” option is preferred for unstructured (e.g., tet) meshes
   b. Stick with central differencing or bounded central differencing for momentum equations
   c. When undershoot/overshoot of the solution fields becomes a critical issue, consider using high-order upwind schemes (QUICK, MUSCL, SOU) for scalars
3. Time-discretization
   a. Use the second-order scheme (default)
   b. Use the NITA/fractional-step method for incompressible flows
   c. With the iterative scheme, SIMPLEC has been found best choice
      i. When the mesh quality is not bad, jack up the URF (0.9 – 1.0) for pressure and momentum equations
4. SGS modeling
   a. Dynamic Smagorinsky’s model is recommended
   b. Dynamic TKE model can potentially benefit highly non-equilibrium flows and reacting flows

Recommended Procedure

1. Compute the mean flow with steady RANS
   k-ε model is sufficient
   The RANS solution doesn’t have to be fully converged
2. Superimpose the synthesized turbulence on the mean flow
   Use TUI (“/solve/initialize/init-instantaneous-vel”)
3. Switch to LES, select the SGS turbulence model of your choice
   GUI (Define/Model/Viscous…)
4. Select the solver algorithm (e.g., ITA/NITA, FSM/PISO/SIMPLEC) and the discretization schemes

Define/Model/Solver…
Solve/Controls/Solution…

5. Set the time-step (Δt) and adjust the solver parameters if needed (e.g., URF's, convergence criteria)
   Solve/Iterate…
   Solve/Controls/Solution…
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 hundreds time-steps)
   File/Write/Autosave…
8. Start the transient run and continue until a statistically stationary state is reached
   Monitoring integral and/or local quantities will help you judge this
9. Save the viewgraphs of your choice for animation (contours of pressure, iso-surfaces of vorticity, second-invariant, etc.)
   Solve/Execute Commands…
10. Start sampling the data (to compute mean and r.m.s. values)
    Solve/Initialize/Reset Statistics
    Solve/Iterate (Click on the "Data Sampling for Time Statistics" button)
11. Continue sampling for a sufficiently long period of time
    Several flow-through times (L/U_0)
    Until the mean fields recover any homogeneity (e.g., axisymmetry, 2-D)
12. Post-process the results
    Display/Contours…(Vectors…)/Unsteady Statistics
    Plot/XY Plot…
    Plot/FFT…