12.8.3 SST k-ω RANS Model

The dissipation term of the turbulent kinetic energy (see Section 12.5.1: Modeling the Turbulence Dissipation) is modified for the DES turbulence model as described in Menter's work [238] such that

$$Y_k = \rho\beta^* k\omega f_{\beta^*} \tag{12.8-6}$$

where f_{β^*} is no longer a constant equal to 1 as in the SST k-ω model (see Section 12.5.1: Modeling the Turbulence Dissipation), but is now expressed as

$$f_{\beta^*} = max\left(\frac{L_t}{C_{\text{des}}\Delta}, 1\right) \tag{12.8-7}$$

where C_{des} is a calibration constant used in the DES model and has a value of 0.61, Δ is the maximum local grid spacing $(\Delta x, \Delta y, \Delta z)$ and f_{β^*} is defined in Equation 12.5-16.

The turbulent length scale is the parameter that defines this RANS model:

$$L_t = \frac{\sqrt{k}}{\beta^*\omega} \tag{12.8-8}$$

12.9 Large Eddy Simulation (LES) Model Theory

12.9.1 Overview

Turbulent flows are characterized by eddies with a wide range of length and time scales. The largest eddies are typically comparable in size to the characteristic length of the mean flow. The smallest scales are responsible for the dissipation of turbulence kinetic energy.

It is possible, in theory, to directly resolve the whole spectrum of turbulent scales using an approach known as direct numerical simulation (DNS). No modeling is required in DNS. However, DNS is not feasible for practical engineering problems involving high Reynolds number flows. The cost required for DNS to resolve the entire range of scales is proportional to Re_t^3, where Re_t is the turbulent Reynolds number. Clearly, for high Reynolds numbers, the cost becomes prohibitive.

In LES, large eddies are resolved directly, while small eddies are modeled. Large eddy simulation (LES) thus falls between DNS and RANS in terms of the fraction of the resolved scales. The rationale behind LES can be summarized as follows:

- Momentum, mass, energy, and other passive scalars are transported mostly by large eddies.
- Large eddies are more problem-dependent. They are dictated by the geometries and boundary conditions of the flow involved.
- Small eddies are less dependent on the geometry, tend to be more isotropic, and are consequently more universal.
- The chance of finding a universal turbulence model is much higher for small eddies.

Resolving only the large eddies allows one to use much coarser mesh and larger time-step sizes in LES than in DNS. However, LES still requires substantially finer meshes than those typically used for RANS calculations. In addition, LES has to be run for a sufficiently long flow-time to obtain stable statistics of the flow being modeled. As a result, the computational cost involved with LES is normally orders of magnitudes higher than that for steady RANS calculations in terms of memory (RAM) and CPU time. Therefore, high-performance computing (e.g., parallel computing) is a necessity for LES, especially for industrial applications.

The following sections give details of the governing equations for LES, the subgrid-scale turbulence models, and the boundary conditions.

12.9.2 Filtered Navier-Stokes Equations

The governing equations employed for LES are obtained by filtering the time-dependent Navier-Stokes equations in either Fourier (wave-number) space or configuration (physical) space. The filtering process effectively filters out the eddies whose scales are smaller than the filter width or grid spacing used in the computations. The resulting equations thus govern the dynamics of large eddies.

A filtered variable (denoted by an overbar) is defined by

$$\overline{\phi}(\mathbf{x}) = \int_{\mathcal{D}} \phi(\mathbf{x}')G(\mathbf{x}, \mathbf{x}')d\mathbf{x}' \tag{12.9-1}$$

where $\mathcal{D}$ is the fluid domain, and G is the filter function that determines the scale of the resolved eddies.

In FLUENT, the finite-volume discretization itself implicitly provides the filtering operation:

$$\overline{\phi}(\mathbf{x}) = \frac{1}{V}\int_{\mathcal{V}} \phi(\mathbf{x}')\, d\mathbf{x}', \quad \mathbf{x}' \in \mathcal{V} \tag{12.9-2}$$

where V is the volume of a computational cell. The filter function, $G(\mathbf{x}, \mathbf{x}')$, implied here is then

$$G(\mathbf{x}, \mathbf{x}') \begin{cases} 1/V, & \mathbf{x}' \in \mathcal{V} \\ 0, & \mathbf{x}' \text{ otherwise} \end{cases} \tag{12.9-3}$$

The LES capability in FLUENT is applicable to compressible flows. For the sake of concise notation, however, the theory is presented here for incompressible flows.

Filtering the Navier-Stokes equations, one obtains

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho \overline{u}_i) = 0 \tag{12.9-4}$$

and

$$\frac{\partial}{\partial t}(\rho \overline{u}_i) + \frac{\partial}{\partial x_j}(\rho \overline{u}_i \overline{u}_j) = \frac{\partial}{\partial x_j}\left(\mu \frac{\partial \sigma_{ij}}{\partial x_j}\right) - \frac{\partial \overline{p}}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} \tag{12.9-5}$$

where σ_{ij} is the stress tensor due to molecular viscosity defined by

$$\sigma_{ij} \equiv \left[\mu\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right)\right] - \frac{2}{3}\mu \frac{\partial \overline{u}_l}{\partial x_l}\delta_{ij} \tag{12.9-6}$$

and τ_{ij} is the subgrid-scale stress defined by

$$\tau_{ij} \equiv \rho\overline{u_i u_j} - \rho \overline{u}_i \overline{u}_j \tag{12.9-7}$$

12.9.3 Subgrid-Scale Models

The subgrid-scale stresses resulting from the filtering operation are unknown, and require modeling. The subgrid-scale turbulence models in FLUENT employ the Boussinesq hypothesis [142] as in the RANS models, computing subgrid-scale turbulent stresses from

$$\tau_{ij} - \frac{1}{3}\tau_{kk}\delta_{ij} = -2\mu_t \overline{S}_{ij} \tag{12.9-8}$$

where μ_t is the subgrid-scale turbulent viscosity. The isotropic part of the subgrid-scale stresses τ_{kk} is not modeled, but added to the filtered static pressure term. $\overline{S}_{ij}$ is the rate-of-strain tensor for the resolved scale defined by

$$\overline{S}_{ij} \equiv \frac{1}{2}\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right) \tag{12.9-9}$$

For compressible flows, it is convenient to introduce the density-weighted (or Favre) filtering operator:

$$\phi = \frac{\overline{\rho\phi}}{\overline{\rho}} \tag{12.9-10}$$

The Favre Filtered Navier-Stokes equation takes the same form as Equation 12.9-5. The compressible form of the subgrid stress tensor is defined as:

$$T_{ij} = -\rho u_i u_j + \overline{\rho} u_i u_j \tag{12.9-11}$$

This term is split into its isotropic and deviatoric parts

$$T_{ij} = \underbrace{T_{ij} - \frac{1}{3}T_{ll}\delta_{ij}}_{\text{deviatoric}} + \underbrace{\frac{1}{3}T_{ll}\delta_{ij}}_{\text{isotropic}} \tag{12.9-12}$$

The deviatoric part of the subgrid-scale stress tensor is modeled using the compressible form of the Smagorinsky model:

$$T_{ij} - \frac{1}{3}T_{ll}\delta_{ij} = 2\mu_t(\delta_{ij} - \frac{1}{3}\delta_{ii}\delta_{ij}) \tag{12.9-13}$$

As for incompressible flows, the term involving T_{ll} can be added to the filtered pressure or simply neglected [99]. Indeed, this term can be re-written as $T_{ll} = \gamma {M^2}_{sgs}\overline{p}$ where M_{sgs} is the subgrid Mach number. This subgrid Mach number can be expected to be small when the turbulent Mach number of the flow is small.

FLUENT offers four models for μ_t: the Smagorinsky-Lilly model, the dynamic Smagorinsky-Lilly model, the WALE model, and the dynamic kinetic energy subgrid-scale model.

Subgrid-scale turbulent flux of a scalar, ϕ, is modeled using s subgrid-scale turbulent Prandtl number by

$$q_j = -\frac{\mu_t}{\sigma_t}\frac{\partial \phi}{\partial x_j} \tag{12.9-14}$$

where q_j is the subgrid-scale flux.

In the dynamic models, the subgrid-scale turbulent Prandtl number or Schmidt number is obtained by applying the dynamic procedure originally proposed by Germano [116] to the subgrid-scale flux.

Smagorinsky-Lilly Model

This simple model was first proposed by Smagorinsky [337]. In the Smagorinsky-Lilly model, the eddy-viscosity is modeled by

$$\mu_t = \rho L_s^2 \left|\overline{S}\right| \tag{12.9-15}$$

where L_s is the mixing length for subgrid scales and $\left|\overline{S}\right| \equiv \sqrt{2\overline{S}_{ij}\overline{S}_{ij}}$. In FLUENT, L_s is computed using

$$L_s = \min\left(\kappa d, C_s V^{1/3}\right) \tag{12.9-16}$$

where κ is the von Kármán constant, d is the distance to the closest wall, C_s is the Smagorinsky constant, and V is the volume of the computational cell.

Lilly derived a value of 0.17 for C_s for homogeneous isotropic turbulence in the inertial subrange. However, this value was found to cause excessive damping of large-scale fluctuations in the presence of mean shear and in transitional flows as near solid boundary, and has to be reduced in such regions. In short, C_s is not an universal constant, which is the most serious shortcoming of this simple model. Nonetheless, C_s value of around 0.1 has been found to yield the best results for a wide range of flows, and is the default value in FLUENT.

Dynamic Smagorinsky-Lilly Model

Germano et al. [116] and subsequently Lilly [211] conceived a procedure in which the Smagorinsky model constant, C_s, is dynamically computed based on the information provided by the resolved scales of motion. The dynamic procedure thus obviates the need for users to specify the model constant C_s in advance. The details of the model implementation in FLUENT and its validation can be found in [181].

The C_s obtained using the dynamic Smagorinsky-Lilly model varies in time and space over a fairly wide range. To avoid numerical instability, in FLUENT, C_s is clipped at zero and 0.23 by default.

Wall-Adapting Local Eddy-Viscosity (WALE) Model

In the WALE model [262], the eddy viscosity is modeled by

$$\mu_t = \rho L_s^2 \frac{(S_{ij}^d S_{ij}^d)^{3/2}}{(\overline{S}_{ij}\overline{S}_{ij})^{5/2} + (S_{ij}^d S_{ij}^d)^{5/4}} \tag{12.9-17}$$

where L_s and S_{ij}^d in the WALE model are defined, respectively, as

$$L_s = \min\left(\kappa d, C_w V^{1/3}\right) \tag{12.9-18}$$

$$S_{ij}^d = \frac{1}{2}\left(\overline{g}_{ij}^2 + \overline{g}_{ji}^2\right) - \frac{1}{3}\delta_{ij}\overline{g}_{kk}^2 \,,\; \overline{g}_{ij} = \frac{\partial \overline{u_i}}{\partial x_j} \tag{12.9-19}$$

In FLUENT, the default value of the WALE constant, C_w, is 0.325 and has been found to yield satisfactory results for a wide range of flow. The rest of the notation is the same as for the Smagorinsky-Lilly model. With this spatial operator, the WALE model is designed to return the correct wall asymptotic (y^3) behavior for wall bounded flows.

Dynamic Kinetic Energy Subgrid-Scale Model

The original and dynamic Smagorinsky-Lilly models, discussed previously, are essentially algebraic models in which subgrid-scale stresses are parameterized using the resolved velocity scales. The underlying assumption is the local equilibrium between the transferred energy through the grid-filter scale and the dissipation of kinetic energy at small subgrid scales. The subgrid-scale turbulence can be better modeled by accounting for the transport of the subgrid-scale turbulence kinetic energy.

The dynamic subgrid-scale kinetic energy model in FLUENT replicates the model proposed by Kim and Menon [184].

The subgrid-scale kinetic energy is defined as

$$k_{\text{sgs}} = \frac{1}{2}\left(\overline{u_k^2} - \overline{u}_k^2\right) \tag{12.9-20}$$

which is obtained by contracting the subgrid-scale stress in Equation 12.9-7.

The subgrid-scale eddy viscosity, μ_t, is computed using k_{sgs} as

$$\mu_t = C_k k_{\text{sgs}}^{1/2} \Delta_f \tag{12.9-21}$$

where Δ_f is the filter-size computed from $\Delta_f \equiv V^{1/3}$.

The subgrid-scale stress can then be written as

$$\tau_{ij} - \frac{2}{3} k_{\text{sgs}} \delta_{ij} = -2 C_k k_{\text{sgs}}^{1/2} \Delta_f \overline{S}_{ij} \tag{12.9-22}$$

k_{sgs} is obtained by solving its transport equation

$$\frac{\partial \overline{k}_{\text{sgs}}}{\partial t} + \frac{\partial \overline{u}_j \overline{k}_{sgs}}{\partial x_j} = -\tau_{ij} \frac{\partial \overline{u}_i}{\partial x_j} - C_\varepsilon \frac{k_{\text{sgs}}^{3/2}}{\Delta_f} + \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_k} \frac{\partial k_{\text{sgs}}}{\partial x_j}\right) \tag{12.9-23}$$

In the above equations, the model constants, C_k and C_ε, are determined dynamically [184]. σ_k is hardwired to 1.0. The details of the implementation of this model in FLUENT and its validation is given by Kim [181].

12.9.4 Inlet Boundary Conditions for the LES Model

This section describes the three algorithms available in FLUENT to model the fluctuating velocity at velocity inlet boundaries.

No Perturbations

The stochastic components of the flow at the velocity-specified inlet boundaries are neglected if the No Perturbations option is used. In such cases, individual instantaneous velocity components are simply set equal to their mean velocity counterparts. This option is suitable only when the level of turbulence at the inflow boundaries is negligible or does not play a major role in the accuracy of the overall solution.

Vortex Method

To generate a time-dependent inlet condition, a random 2D vortex method is considered. With this approach, a perturbation is added on a specified mean velocity profile via a fluctuating vorticity field (i.e. two-dimensional in the plane normal to the streamwise direction). The vortex method is based on the Lagrangian form of the 2D evolution equation of the vorticity and the Biot-Savart law. A particle discretization is used to solve this equation. These particles, or "vortex points" are convected randomly and carry information about the vorticity field. If N is the number of vortex points and A is the area of the inlet section, the amount of vorticity carried by a given particle i is represented by the circulation Γ_i and an assumed spatial distribution η:

$$\Gamma_i(x,y) = 4\sqrt{\frac{\pi A k(x,y)}{3N[2\ln(3)-3\ln(2)]}} \tag{12.9-24}$$

$$\eta(\vec{x}) = \frac{1}{2\pi\sigma^2}\left(2e^{-|x|^2/2\sigma^2}-1\right)2e^{-|x|^2/2\sigma^2} \tag{12.9-25}$$

where k is the turbulence kinetic energy. The parameter σ provides control over the size of a vortex particle. The resulting discretization for the velocity field is given by

$$\vec{u}(\vec{x}) = \frac{1}{2\pi}\sum_{i=1}^{N}\Gamma_i\frac{((\vec{x}_i-\vec{x})\times\vec{z})(1-e^{|\vec{x}-\vec{x}'|^2/2\sigma^2})}{|\vec{x}-\vec{x}_i'|^2} \tag{12.9-26}$$

Where $\vec{z}$ is the unit vector in the streamwise direction. Originally [327], the size of the vortex was fixed by an ad hoc value of σ. To make the vortex method generally applicable, a local vortex size is specified through a turbulent mixing length hypothesis. σ is calculated from a known profile of mean turbulence kinetic energy and mean dissipation rate at the inlet according to the following:

$$\sigma = \frac{ck^{3/2}}{2\epsilon} \tag{12.9-27}$$

where $c = 0.16$. To ensure that the vortex will always belong to resolved scales, the minimum value of σ in Equation 12.9-27 is bounded by the local grid size. The sign of the circulation of each vortex is changed randomly each characteristic time scale τ. In the general implementation of the vortex method, this time scale represents the time necessary for a 2D vortex convected by the bulk velocity in the boundary normal direction to travel along n times its mean characteristic 2D size (σ_m), where n is fixed equal to 100 from numerical testing. The vortex method considers only velocity fluctuations in the plane normal to the streamwise direction.

In FLUENT however, a simplified linear kinematic model (LKM) for the streamwise velocity fluctuations is used [231]. It is derived from a linear model that mimics the influence of the two-dimensional vortex in the streamwise mean velocity field. If the mean streamwise velocity U is considered as a passive scalar, the fluctuation u' resulting from the transport of U by the planar fluctuating velocity field v' is modeled by

$$u' = -\vec{v}' \cdot \vec{g} \tag{12.9-28}$$

where $\vec{g}$ is the unit vector aligned with the mean velocity gradient $\nabla\vec{U}$. When this mean velocity gradient is equal to zero, a random perturbation can be considered instead.

i Since the vortex method theory is based on the modification of the velocity field normal to the streamwise direction, it is imperative that the user creates an inlet plane normal (or as close as possible) to the streamwise velocity direction.

Spectral Synthesizer

The spectral synthesizer provides an alternative method of generating fluctuating velocity components. It is based on the random flow generation technique originally proposed by Kraichnan [186] and modified by Smirnov et al. [338]. In this method, fluctuating velocity components are computed by synthesizing a divergence-free velocity-vector field from the summation of Fourier harmonics. In the implementation in FLUENT, the number of Fourier harmonics is fixed to 100.