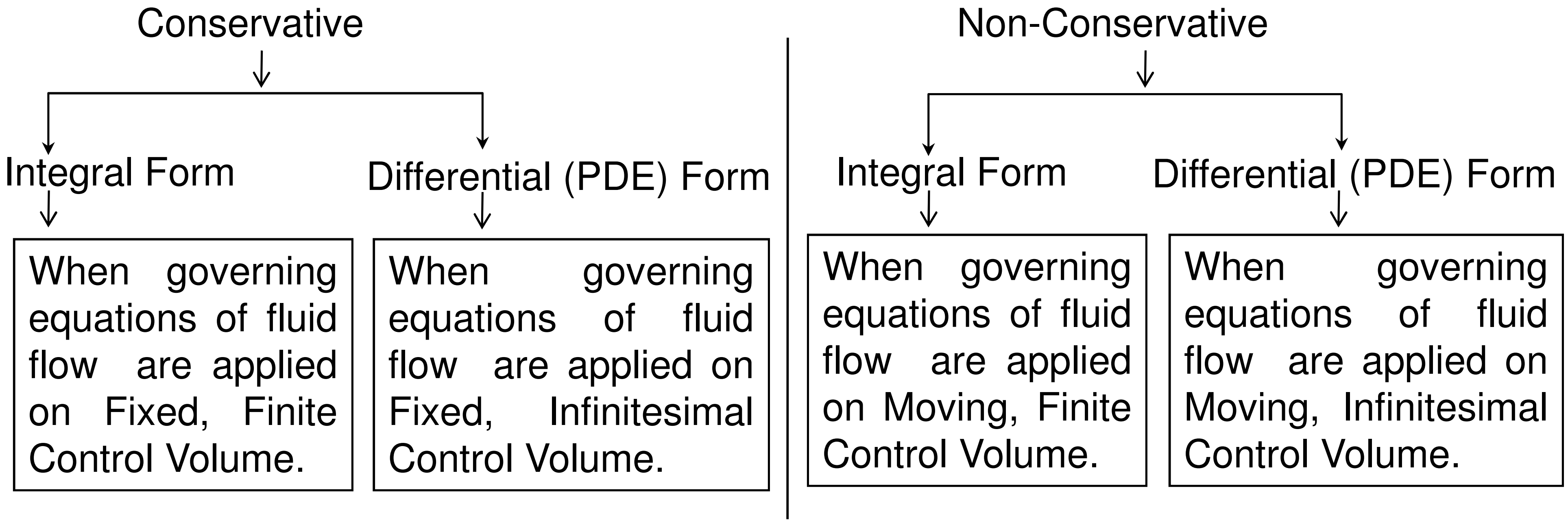
Navier-Stokes Equation

This equation is a manifestation of "mass conservation" and "Newton's 2nd Law" on fluid particles also called "Control Volume", C.V. Mass conservation usually known as "Continuity Equation" and Newton's 2nd Law designated as "Momentum Equation" are collectively called as "Governing Equation of Fluid Flow".

It is imperative that every CFD practitioner has good understanding, both mathematically and practically, of the two equations. Based on mathematical treatment, the momentum equation (Navier-Stokes equation) takes following 4 forms:



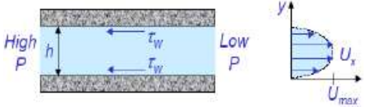
Conservative

- Integral Form
  - When governing equations of fluid flow are applied on Fixed, Finite Control Volume.
- Differential (PDE) Form
  - When governing equations of fluid flow are applied on Fixed, Infinitesimal Control Volume.

Non-Conservative

- Integral Form
  - When governing equations of fluid flow are applied on Moving, Finite Control Volume.
- Differential (PDE) Form
  - When governing equations of fluid flow are applied on Moving, Infinitesimal Control Volume.

Understanding SIMPLE Algorithm through Analogy



Laminar Flow: 1-D

$$\frac{\partial}{\partial y}\left[\mu\ \frac{\partial u}{\partial y}\right] = \frac{\partial p}{\partial x}$$

∂p/∂x is assumed constant

Newton's Law of (Laminar) Viscosity

$$\tau_{xy} = \mu\ \frac{dU}{dy}$$

u = u(y, μ, ∂p/∂x)

∂p/∂x is not known

Solve Ordinary Partial Differential Equation & Apply B.C.

$u = 1/2\mu.\ (\partial p/\partial x).\ y^2 + C_1.y + C_2$

y =0, u =0 || y = w, u = 0 (NO SLIP) & Px = Constant

$u = 1/2\mu.\ (\partial p/\partial x)\ (y^2 - w.y)$

∂p/∂x is still not known

Invoke Continuity

$$\frac{\partial u_i}{\partial x_i} = 0 \qquad Q = W\int_0^b U_x dy$$

$Q = w\ h^3.\ \Delta p\ /\ 12\mu\ L$,
w = gap between plates

~~Solve Momentum Euqation~~

~~Invoke Continuity~~

Turbulent Flow: 3 D

$$\frac{\partial \overline{u}_i}{\partial t} + \overline{u}_j\frac{\partial \overline{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \upsilon\frac{\partial^2 \overline{u}_i}{\partial x_j \partial x_j} - \frac{\partial}{\partial x_j}\overline{u_i' u_j'}$$

Newton's Law of (Effective) Viscosity

$$\tau_{xy} = (\mu + \mu_T)\frac{dU}{dy}$$

u = u(x, y, z, μ, μ_t, ∂p/∂x)

∂p/∂x is not known

Discretize NS Equation → Generate Algebraic System of Equations

$$\left(\frac{\partial \Phi}{\partial x}\right)_i \approx \frac{\Phi_i - \Phi_{i-1}}{x_i - x_{i-1}} \qquad \phi = u, v, w, p$$

Guess Pressure Field (p*) & Solve following Set of Equations:

[A] . {u} = b   [A] . {v} = {b}   [A] . {w} = {b}

p(x,y,z) is still not known

Invoke Continuity

Get "Pressure Correction" Equation: p = p* + p` in order to satisfy mass conservation, p` is a Numerical artifice to get u,v,w satisfying continuity

Solve Momentum Euqation

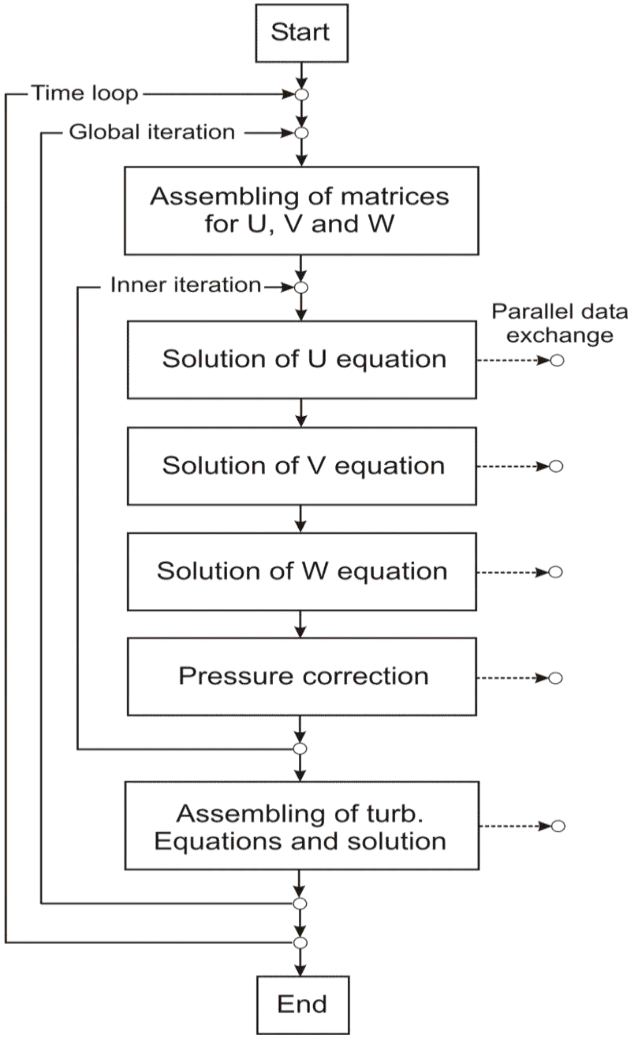
Update Velocity Vield u^{n+1}, v^{n+1}, w^{n+1}

Invoke Continuity

and so on till global mass imbalance is withing Desired Criteria.

Solver Setting: Segregated

Segregated Solver



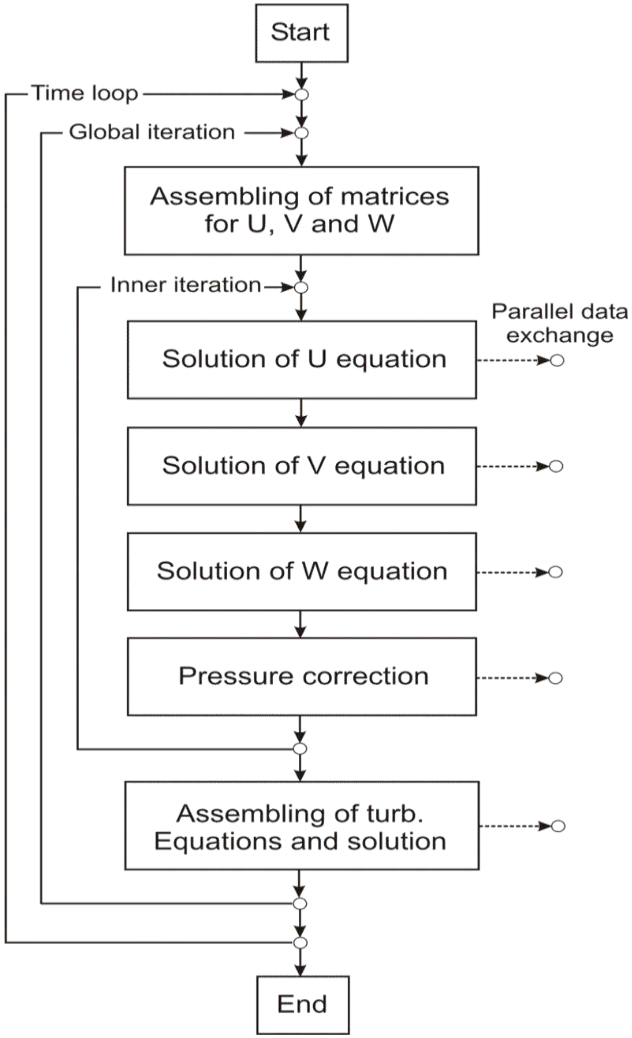
Excerpts from STAR-CCM+ User Guide:

The Segregated Flow model solves the flow equations (one for each component of velocity, and one for pressure) in a segregated, or uncoupled, manner. The linkage between the momentum and continuity equations is achieved with a predictor-corrector approach.

The complete formulation can be described as using a collocated variable arrangement (as opposed to staggered) and a Rhie-and-Chow-type pressure-velocity coupling combined with a SIMPLE-type algorithm.

This model has its roots in constant-density flows. Although it is capable of handling mildly compressible flows and low Raleigh number natural convection, it is not suitable for shock-capturing, high Mach number and high Raleigh-number applications.

Segregated Solver



Here governing conservation equations are solved sequentially (segregated from each other).

1. Physical properties such as Viscosity, Thermal Conductivity, etc are updated, based on the current iteration step. (For 1st iteration it is based on the initialized data.)
2. Momentum equations are solved using current values for pressure and face mass fluxes, to update velocity.
3. Since the velocities obtained in Step 2 may not satisfy the continuity equation locally, a “Poisson-type” equation for the pressure correction is derived from continuity and linearized momentum equations. Pressure correction equation is then solved to obtain the necessary corrections to the pressure and velocity fields & mass fluxes such that continuity is satisfied.
4. Other equations for scalars such as turbulence, energy are solved using the previously updated values of the other variables.

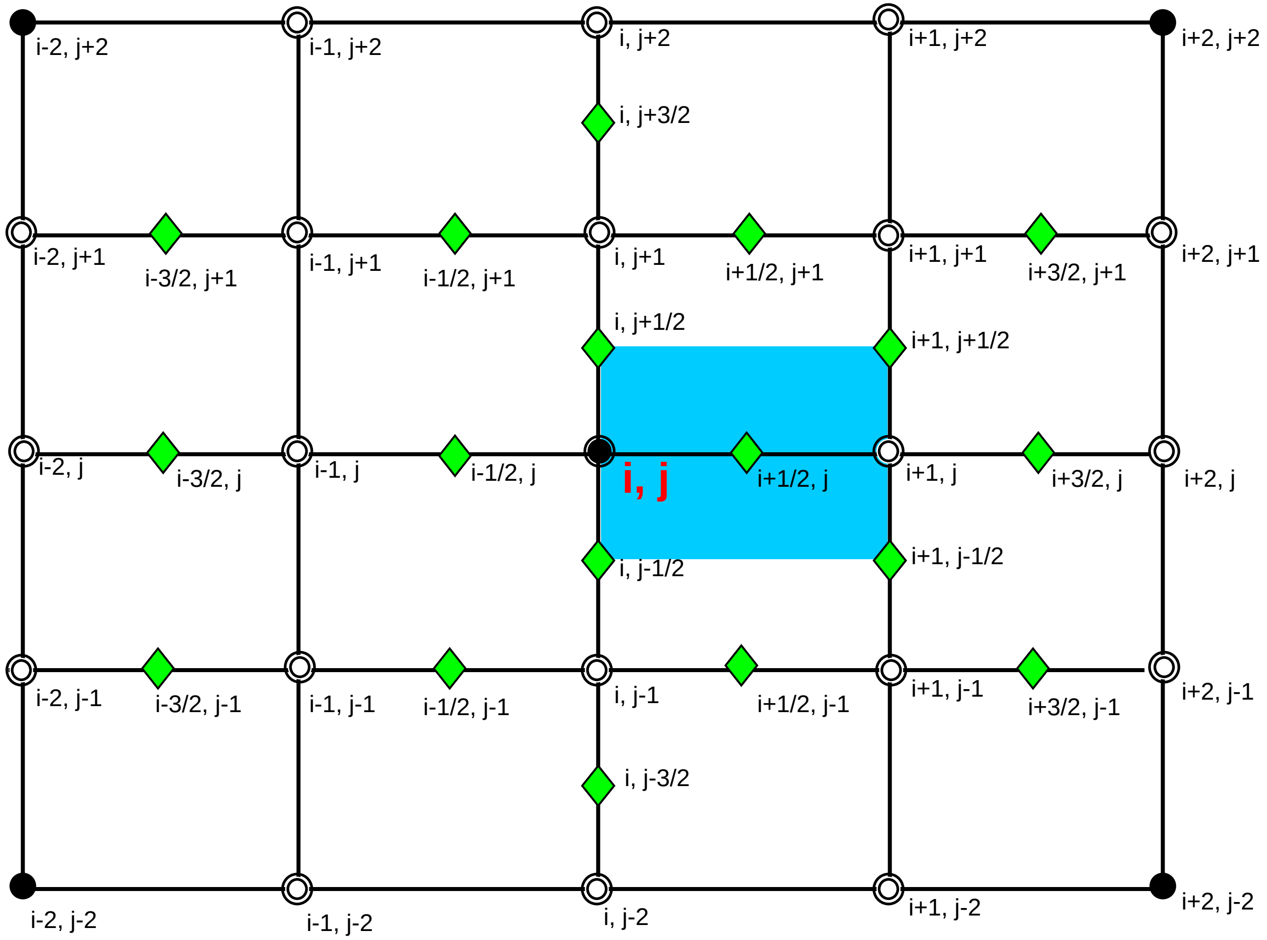
Segregated Solver: When to use?

Why use Segregated Solver?

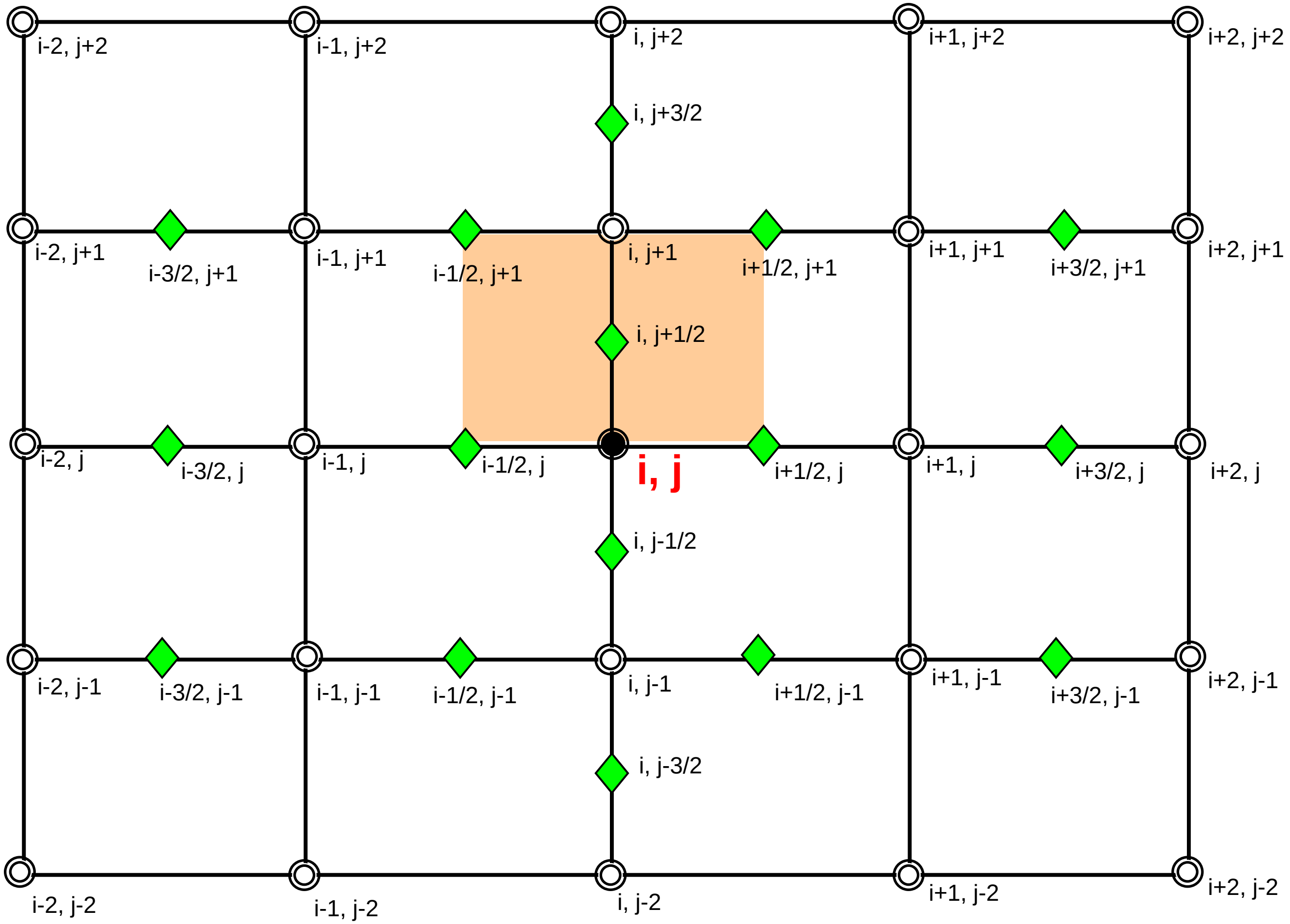
The Segregated Flow model requires less memory and have a faster convergence rate for well-posed problems. Hence, if you do not have large machines to make complex runs, this approach is the only option.

Segregated solvers are not unconditionally stable, that is, the convergence is not always guaranteed. Hence, sometimes it is not possible to reach a good result with this algorithm.

Staggered Grid – X-Momentum Equation



Staggered Grid – Y-Momentum Equation



Solver Setting: Coupled Solver

Coupled Solver

The coupled solvers solve the governing equations of continuity, momentum, and (when ON) energy and species transport simultaneously as a set, or vector, of equations. Governing equations for additional scalars (such as turbulence parameters) will be solved sequentially (i.e. segregated from one another & from coupled set).

Excerpts from STAR-CCM+ User Guide:

The Coupled Flow and Energy model solves the conservation equations for mass, momentum and energy simultaneously using a time- (or pseudo-time-) marching approach.

One advantage of this formulation is its robustness for solving compressible flows or those with dominant source terms (such as rotation or buoyancy). Another advantage of the coupled solver is that CPU time scales linearly with cell count; in other words, the convergence rate does not deteriorate as the mesh is refined. Furthermore, due to the preconditioned form of the governing equations used by the coupled flow and energy model, convergence rate is effectively independent of Mach number, ranging from incompressible through to supersonic regimes.

Comparative Summary of Turbulence Models

Sr. No.	Turbulence Model	Typical Application	Speciality of the Model	Limitations with Reasons	Category
1	Spalart Allmaras Model	*Suitable for mildly complex (quasi-2D) external/internal flows and B.L. flows under pressure gradient *Turbine Blades, specifically developed for wall-bounded aerodynamic flows with adverse pressure gradients like airfoils, airplane fuselage, missiles, ship hulls	*Low Re Model directly applicable throughout B.L. Good compromise between accuracy & simplicity *Solves a transport equation for turbulent eddy viscosity itself, instead of specifying it with a characteristic velocity and length scales. *Transported variables have fewer gradients than other modes, thus doesn't demand too fine grids near wall.	* Y^+ value should be of the oder of 5 *Performs poorly in adverse pressure gradient flows, Point of zero skin friction that is separation and reattachement points *Production term is function of vorticity and the later vanishes at separation point	1-Equation Model
2	Baldwin-Lomax Model	*Thin BL-flow that is favourable pressure gradient flows, Turbine Blades	0-Equation Model	*Adverse pressure gradient flows, typical of any 0-Equation model	**0-Equation Model:** Velocity scale V and length scale L are related by algebraic equations to the local properties of the flow.
3	Standard k-ε Model	*Attached flow, generally applicable to high Re number homogeneous flow and in which production and dissipation turbulence are in local equilibrium *Most useful in parametric studies & intial screening of various design options *Typically all types of flow since solution time and convergence issues are MINIMUM.	*The most VERSATILE & SIMPLE turbulence model *Transport equation contains classical convection and diffusion term & 'modelled' dissipation and production terms.	*Flow with boundary layer separation, basic assumption of isotropic turbulence thru' Eddy Viscosity m_t *Flow with sudden change in strain rate *Poor prediction of recirculating flows which involves streamline curvature and rotational strains, not suitable for unsteady flow	2-Equation Model: The standard k-ε model employs a single time scale t = k/ε to characterize dynamic processes occurring in turbulent flows. Accordingly, the source, sink & transport terms are held to proceed at rates μ to k/ε. Turbulence, however, comprises fluctuating motions with a spectrum of time scales, and a single-scale approach is unlikely to be adequate under all circumstances because different turbulence interactions are associated with different parts of the spectrum.
4	Realizable k-ε Model	*Applies certain mathematical constraints on Reynold stresses consistent with turbulent flows *Flows involving rotation, boundary layers under adverse pressure gradients, separation & recirculation.	*Ensures turbulent normal stress 'Realizabel' i.e. > 0 (Schwarz Inequality) *Easier to converge than RNG *Variable Cm instead of constant	*Difficult to converge	
5	Low-Re k-ε Model	When lift, drag and pressure drop estimation are of prime importance	*Does not utilize wall funtion *Very sensitive to height of the 1st cell next to the solid boundary	*Fine computational mesh in near wall region due to abandoning of use of wall functions, thereby requiring solution of the viscosity-affected sublayer close to the wall.	
6	RNG k-ε Model	*Flows with strong curvature, vortices, local transitional flows, complex shear flows with high strain rate *Transitional flows * Wall heat and mass transfer	*Additional dissipation term which accounts for the effect of mean flow distortion on ε * Analytical formula for turbulent Prandtl number * Differential formula for effective viscosity	*Suffers from inherent limitation of isotropic eddy viscosity model * Does not predict the spreading of a round jet correctly.	2-Equation Model: the application of a rigorous statistical technique (Renormalization Group Method) to the instantaneous Navier-Stokes equations.
7	SST	*To benefit from this model, BL has to be resolved with MIN 10 nodes *Better prediction of leading edge HTC *Free Jet, External Aerodynamic and turbomachinery	*A combination of 2 models, k-ε and k-ω, with specific blending function. k-ω is activated in BL and k-ε in rest of the domain *The SST model produces highly accurate prediction of flow separation & is suited for aerodynamic simulation.	*Dependency on wall distance makes it less suitable for free shear flow *High degreee of boundary layer resolution	
8	Standard k-ω Model	*Effect of transition, free surface turbulence, wall roughness *Free Jet, External Aerodynamic and turbomachinery	Does not utilize wall function and hence require very fine grid near the wall	*Sensitive to free-stream value of turbulence frequency outside the boundary layer	
9	Reynolds Stress Model (RSM)	*Free shear flows with strong anisotropy, like a strong swirl component. This includes swirling flows like Cyclons, Stirred Tank *Sudden changes in the mean strain rate. *Complex strain fields , reproduces the anisotropic nature of turbulence itself.	*High degree of tight coupling between the momentum equations and the turbulent stresses **All 6 components of Reynolds stress are directly computed instead of modeling done in standard EVMs (Eddy Viscosity Models), thus avoids isotropic eddy viscosity assumption	*Numerical scheme is more prone to stability and convergence difficulties as compared to k-e model *More CPU time and large memory requirements	2nd Order or Second Moment Closure Method
10	Large Eddy Simulation (LES)	*Primarily for research purpose *Gives details on the structure of turbulent flows, e.g. pressure fluctuations & Lighthill stresses, which can't be obtained from RANS *Flow is likely to be unstable, with large scale flapping of a shear layer or vortex shedding. *Other fluctuating information is required (e.g., fluctuating forces, gusts of winds).	*Experimental turbulence model *Solves for the large-scale fluctuating flows and uses “sub-grid” scale (SGS) turbulence models for the small-scale motion. *The flow is likely to be unsteady with coherent structures (cyclone, flasher). *The noise from the flow is to be calculated, and especially when the broadband contribution is significant. *Space-averaged model and not the time-averaged	*Single phase, Single component, Non-reacting Flow *Very high computation time (typically weeks for 8 to 16 processor systems) *Not recommended for wall-wounded flows due to high resolution requirements & consequently large computation time *Symmetry BC & 2D cannot be used as Turbulence a 3D phenomena.	
11	Detaached Eddy Simulation (DES)	Detached eddy simulation combines RANS in BL and LES in detached (separated) region	*Good prediction of separation and separtion line *Car wake region, car side view mirror *Flow around aerodynamic obstacles like buildings, bridges	*CPU intensive, execution time is at least an order of magnitude higher than RANS *Does not use SYMMETRY & PERIODICITY boundary conditions	
12	v^2-f Model	Accurate in predicting the peak of turbulent KE near the wall	*Non-equilibrium Flows like Tank Filling	High computational cost (~ 20% high as compared to 2-equation model)	4-Equation Model: Wall normal fluctuations are responsible for near wall transport which behaves different than other two fluctuation components.