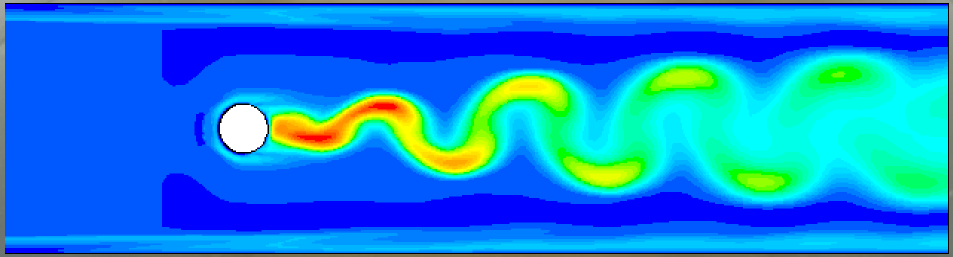
Modelling Turbulent Flows with FLUENT

Dr Aleksey Gerasimov
Senior CFD Engineer
Fluent Europe Ltd
October 2006



Is the Flow Turbulent?

External Flows

$\mathbf{Re}_x \geq 5 \times 10^5$ along a surface

$\mathbf{Re}_D \geq 20{,}000$ around an obstacle

Internal Flows

$\mathbf{Re}_{D_h} \geq 2{,}300$

where $\mathbf{Re}_L \equiv \dfrac{\rho UL}{\ }$

$L = x, D, D_h$, etc.

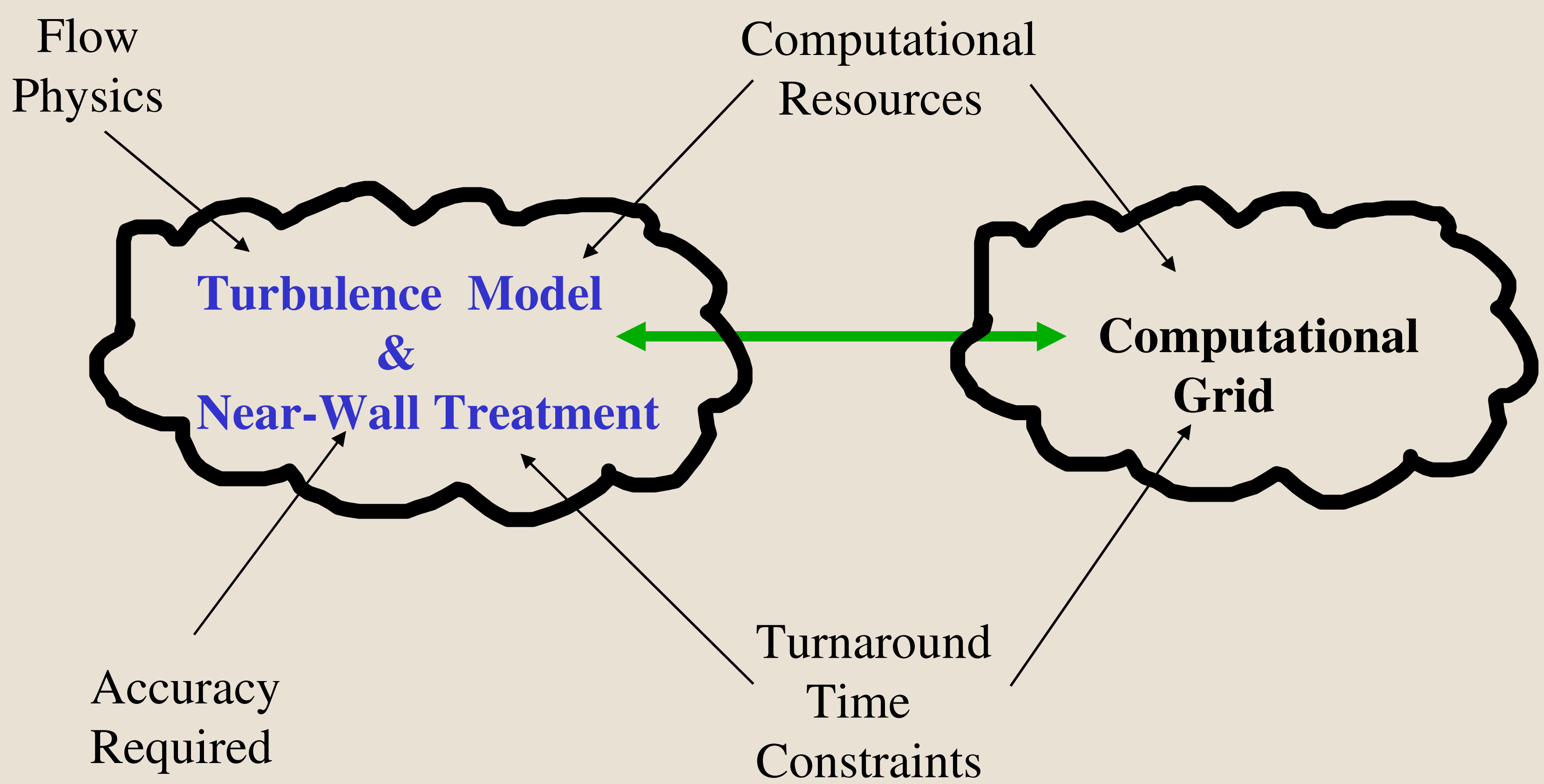
Other factors such as free-stream turbulence, surface conditions, and disturbances may cause earlier transition to turbulent flow

Natural Convection

$Ra \geq 10^8 - 10^{10}$

where $Ra \equiv \dfrac{g\beta\Delta TL^3\rho}{\mu\alpha}$

Choices to be Made



Turbulence Models in Fluent

Zero-Equation Models

One-Equation Models

- Spalart-Allmaras

Two-Equation Models

- Standard k-ε
- RNG k-ε
- Realizable k-ε
- Standard k-ω
- SST k-ω

V2F Model

Reynolds-Stress Model

Detached Eddy Simulation

Large-Eddy Simulation

Direct Numerical Simulation

RANS-based models

Available in **FLUENT 6.2**

Increase in Computational Cost Per Iteration

RANS Turbulence Models

Model	Description:
Spalart-Allmaras	A single transport equation model solving directly for a modified turbulent viscosity. Designed specifically for aerospace applications involving wall-bounded flows on a fine, near-wall mesh. Fluent's implementation allows use of coarser meshes. •Option to include strain rate in *k* production term improves predictions of vortical flows.
Standard *k*-ε	The baseline two transport equation model solving for *k* and ε. This is the default *k*-ε model. Coefficients are empirically derived; valid for fully turbulent flows only. •Options to account for viscous heating, buoyancy, and compressibility are shared with other *k*-ε models.
RNG *k*-ε	A variant of the standard *k*-ε model. Equations and coefficients are analytically derived. Significant changes in the ε equation improves the ability to model highly strained flows. •Additional options aid in predicting swirling and low Re flows.
Realizable *k*-ε	A variant of the standard *k*-ε model. Its 'realizability' stems from changes that allow certain mathematical constraints to be obeyed which ultimately improves the performance of this model.
Standard *k*-ω	A two transport equation model solving for *k* and ω, the specific dissipation rate (ε/*k*) based on Wilcox (1998). This is the default *k* -ω model. Demonstrates superior performance for wall bounded and low-Re flows. Shows potential for predicting transition. •Options account for transitional, free shear, and compressible flows.
SST *k*-ω	A variant of the standard *k*-ω model. Combines the original Wilcox model (1988) for use near walls and standard *k*-ε model away from walls using a blending function. Also limits turbulent viscosity to guarantee that $\tau_t \sim k$. •The transition and shearing options borrowed from SKO. No compressibility option.
RSM	Reynolds stresses are solved directly with transport equations avoiding isotropic viscosity assumption of other models. Use for highly swirling flows. •Quadratic pressure-strain option improves performance for many basic shear flows.

RANS Turbulence Models Behavior and Usage

Model	Behavior and Usage
Spalart-Allmaras	Economical for large meshes. Performs poorly for 3D flows, free shear flows, flows with strong separation. Suitable for mildly complex (quasi-2D) external/internal flows and b.l. flows under pressure gradient (e.g. airfoils, wings, airplane fuselage, missiles, ship hulls).
Standard *k*-ε	Robust. Widely used despite the known limitations of the model. Performs poorly for complex flows involving severe ∇p, separation, strong stream line curvature. Suitable for initial iterations, initial screening of alternative designs, and parametric studies.
RNG *k*-ε	Suitable for complex shear flows involving rapid strain, moderate swirl, vortices, and locally transitional flows (e.g., b.l. separation, massive separation and vortex-shedding behind bluff bodies, stall in wide-angle diffusers, room ventilation)
Realizable *k*-ε	Offers largely the same benefits and has similar applications as RNG. Possibly more accurate and easier to converge than RNG.
Standard *k*-ω	Superior performance for wall-bounded b.l., free shear, and low Re flows. Suitable for complex boundary layer flows under adverse pressure gradient and separation (external aerodynamics and turbomachinery). Can be used for transitional flows (though tends to predict early transition). Separation is typically predicted to be excessive and early.
SST *k*-ω	Similar benefits as SKO. Dependency on wall distance makes this less suitable for free shear flows.
RSM	Physically the most sound RANS model. Avoids isotropic eddy viscosity assumption. More CPU time and memory required. Tougher to converge due to close coupling of equations. Suitable for complex 3D flows with strong streamline curvature, strong swirl/rotation (e.g. curved duct, rotating flow passages, swirl combustors with very large inlet swirl, cyclones).

Near-Wall Modelling
Recommended Strategy

- ***For most high Re industrial applications (Re > 10^6) for which you cannot afford to resolve the viscous sublayer, use StWF or NEWF***
  - *There is a lot of evidence showing that there is little gain from resolving the viscous sublayer (choice of core turbulence model is more important)*
- ***You may consider using EWT if:***
  - *The same or similar cases ran successfully previously with the two-layer zonal model (in Fluent v5)*
  - *The physics and near-wall mesh of the case is such that y^+ is likely to vary in a wide range in a significant portion of the wall region*
  - *Try to make the mesh either coarse or fine enough, and avoid putting the wall-adjacent cells in the buffer layer ($y^+ = 5 \sim 30$)*

Estimating Placement of First Near-Wall Grid Point

- ***Ability for near-wall treatments to accurately predict near-wall flows depends on placement of wall adjacent cell centroids (cell size)***
  - *For StWF and NEWF, centroid should be located in log-layer:* $y_p^+ \approx 30 - 300$
  - *For best results using EWT, centroid should be located in laminar sublayer:* $y_p^+ \approx 1$
    - *This near-wall treatment can accommodate cells placed in the log-layer*
- ***To determine actual size of wall adjacent cells, recall that:***
  - $y_p^+ \equiv y_p u_\tau / \nu \;\Rightarrow\; \boxed{y_p \equiv y_p^+ \nu / u_\tau}$
  - $u_\tau \equiv \sqrt{\tau_w / \rho} = U_e \sqrt{\bar{c}_f / 2}$
  - *The skin friction coefficient can be estimated from empirics:*
    - *Flat Plate -* $\bar{c}_f / 2 \approx 0.037 \, \mathrm{Re}_L^{-0.2}$
    - *Pipe Flow -* $\bar{c}_f / 2 \approx 0.039 \, \mathrm{Re}_D^{-0.2}$
- ***Use post-processing to confirm near-wall mesh resolution***

Setting Boundary Conditions

- ***When turbulent flow enters a domain at inlets or outlets (potential backflow), boundary values for:***
  - *k, ε, ω and/or $\overline{u_i u_j}$ must be specified*
- ***Four methods for directly or indirectly specifying turbulence parameters:***
  - *Explicitly input k, ε, ω, or $\overline{u_i u_j}$*
    - *This is the only method that allows for profile definition.*
  - *Turbulence intensity and length scale*
    - *Length scale is related to size of large eddies that contain most of energy.*
      - *For boundary layer flows: $l \approx 0.4\delta_{99}$*
      - *For flows downstream of grid: $l \approx$ opening size*
  - *Turbulence intensity and hydraulic diameter*
    - *Ideally suited for duct and pipe flows*
  - *Turbulence intensity and turbulent viscosity ratio*
    - *For external flows: $1 < \mu_t/\mu < 10$*
- ***Turbulence intensity depends on upstream conditions:***

$$u/U \approx \sqrt{2k/3}/U < 20$$

GUI for Turbulence Models

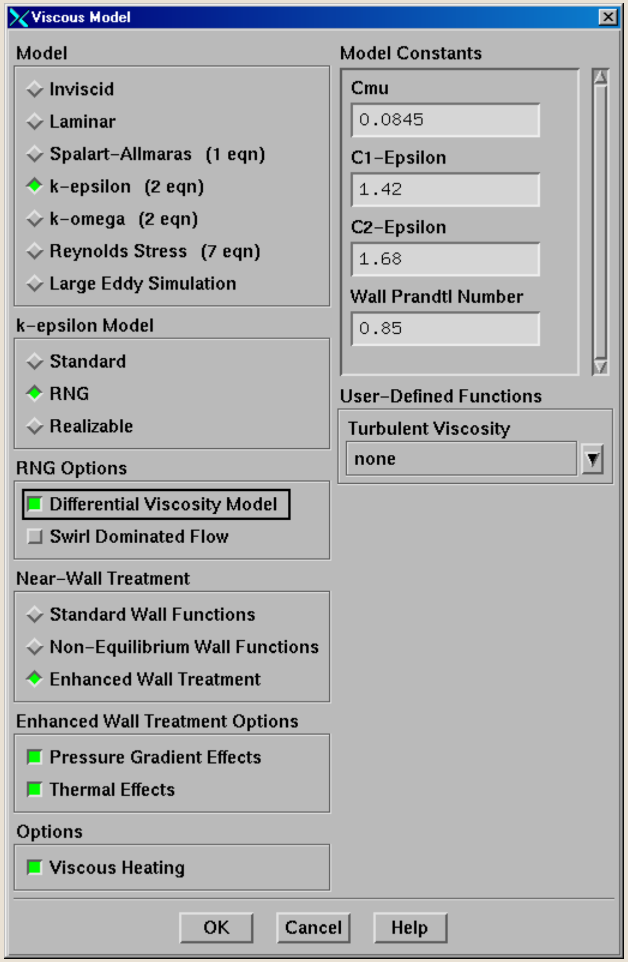
Define → Models → Viscous...

Inviscid, Laminar, or Turbulent

Turbulence Model options

Near Wall Treatments

Additional Turbulence options



Summary:
Turbulence Modelling Guidelines

- ***Successful turbulence modelling requires engineering judgement of:***
  - *Flow physics*
  - *Computer resources available*
  - *Project requirements*
    - *Accuracy*
    - *Turnaround time*
  - *Turbulence models & near-wall treatments that are available*
- ***Modelling Procedure***
  - *Calculate characteristic Re and determine if flow is turbulent*
  - *Estimate wall-adjacent cell centroid y^+ first before generating mesh*
  - *Begin with SKE (standard k-ε) and change to RNG, RKE, SKO, or SST if needed*
  - *Use RSM for highly swirling flows*
  - *Use wall functions unless low-Re flow and/or complex near-wall physics are present*