MECH4480
Computational Fluid Dynamics

CFD

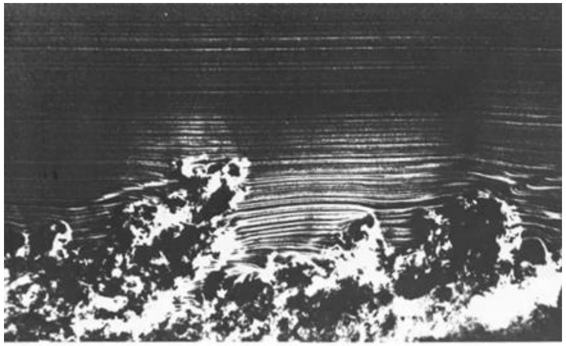
Finite-Volume Formulation

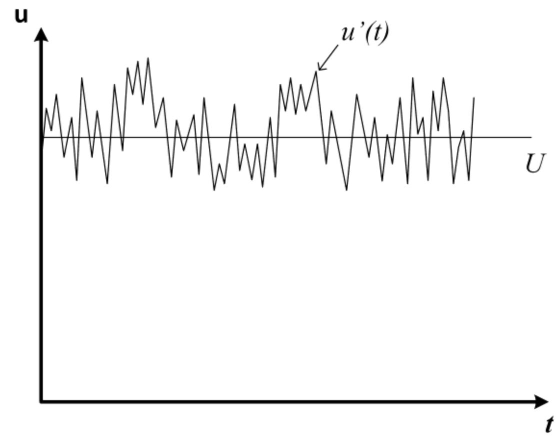
Content

<u>OpenFOAM</u>

1. OpenFOAM
    1. Setting up and running a first case
    2. Setting Boundary Conditions
    3. Adjusting the solver
    4. Getting good solutions
2. Turbulence modelling
    1. What is turbulence
    2. Modelling the effects of turbulence
3. Solving the Governing equations:
   The solution process in OF

What is turbulence





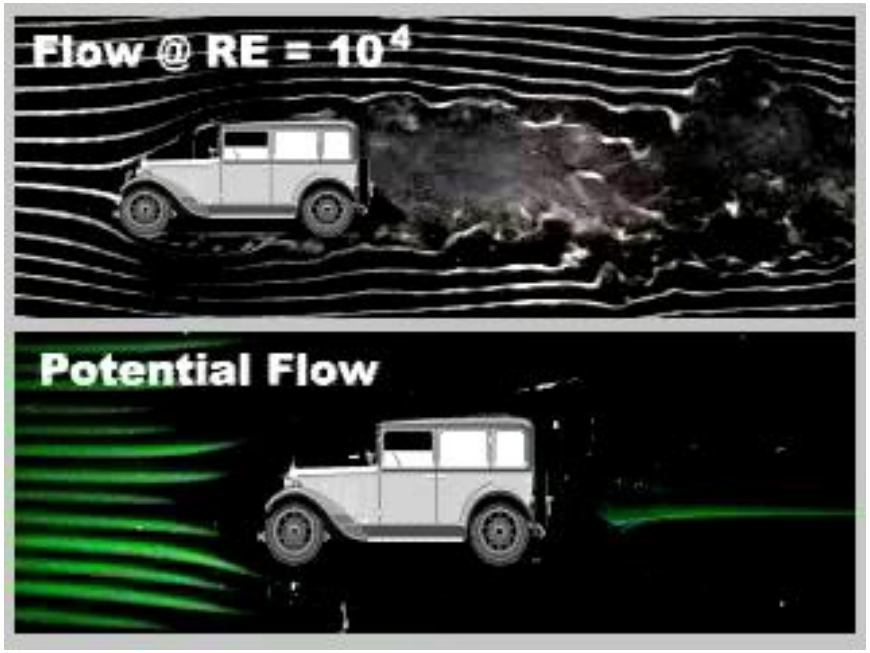
Video – Jets at increasing Reynolds Number

For flows above a critical Reynolds number, Re_{crit}, are turbulent. This means the flow becomes intrinsically unsteady. The **random** nature of the flow precludes an economical simulation of all flow particles. However the flow can be decomposed into a steady maen value, U, and a fluctuating term, u'.

$$u(t) = U + u'(t)$$

The same can be applied to other fluid properties: v, w, p, ... Note: Even in quasi 2-D flows, the turbulent flows have a 3-D nature Note: The turbulent eddies have a wide range of length scales.

Why modelling turbulence is hard…



- Turbulent structures span a wide range of length and time scales and become larger as Reynolds number increases.
- to resolve turbulence at smallest scales requires very fine meshes and time steps $\rightarrow$ prohibitively long simualtion times.

RANS – Reynolds Averaged Navier Stokes

The Reynolds decomposition $u(t) = U + u'(t)$ and the fact that the $\overline{u'(t)} = 0$ allows the RANS equations to be developed. (see Textbook for derivation)

$$\frac{d\bar{u_x}}{dx} + \frac{d\bar{u_y}}{dy} + \frac{d\bar{u_z}}{dz} = 0 \qquad \text{Continuity}$$

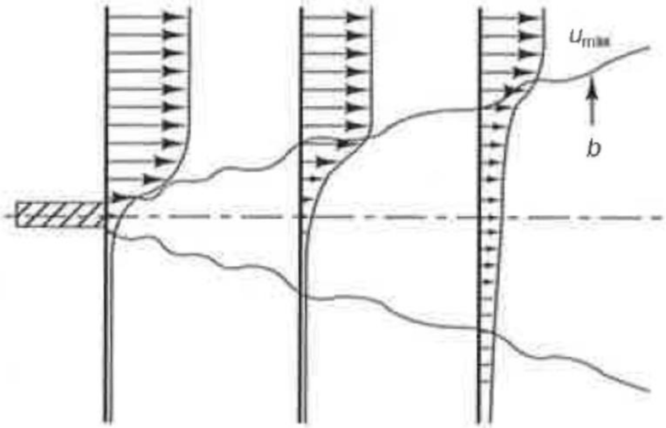
$$\rho\left(\frac{d\bar{u}_x}{dt} + \bar{u}_x\frac{d\bar{u}_x}{dx} + \bar{u}_y\frac{d\bar{u}_x}{dy}\right) = -\frac{d\bar{p}}{dx} + \underbrace{\mu\left(\frac{d^2\bar{u_x}}{dx^2} + \frac{d^2\bar{u_x}}{dy^2}\right)}_{\text{viscous stress}}$$

$$+\underbrace{\rho\left(\frac{d\overline{u'_x u'_x}}{dx} + \frac{\overline{u'_x u'_y}}{dy}\right)}_{\text{Reynolds stress}} + \rho S_x \qquad \text{x-momentum in 2-D}$$

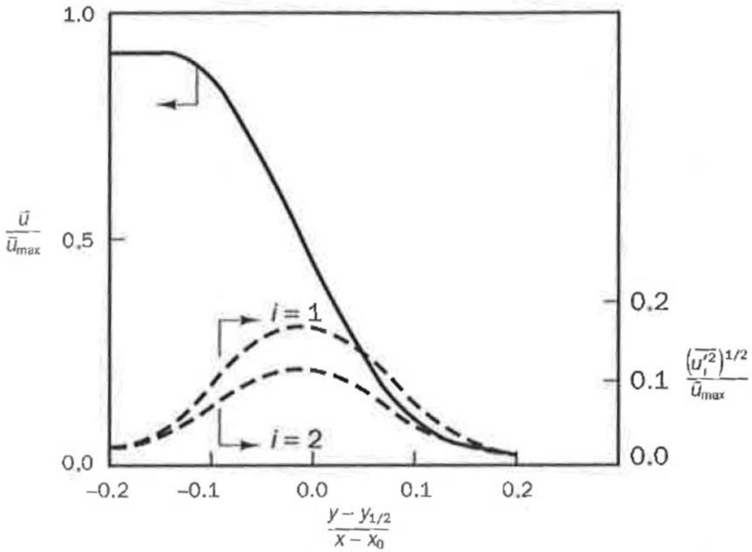
$$\rho\frac{D\bar{u}_i}{Dt} = -\frac{d\bar{p}}{dx_i} + \underbrace{\mu\,\Delta\bar{u}_i}_{\text{viscous stress }=\tau} - \underbrace{\rho\left(\frac{d\overline{u'_i u'_j}}{dx_j}\right)}_{\text{Reynolds stress }=\tau_t} + \rho S_i \qquad \text{Momentum in tensor form}$$

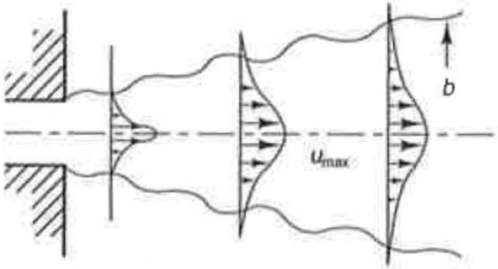
- From RANS equations:
  - Continuity equation is based on mean properties
  - Momentum equation requires correction for *Reynolds stress*
- For Turbulence modelling:
  - Create a mathematical function for *Reynolds stress* based on flow properties and / or energy dissipation.
  - Resolve large eddies
  - "Model" small eddies

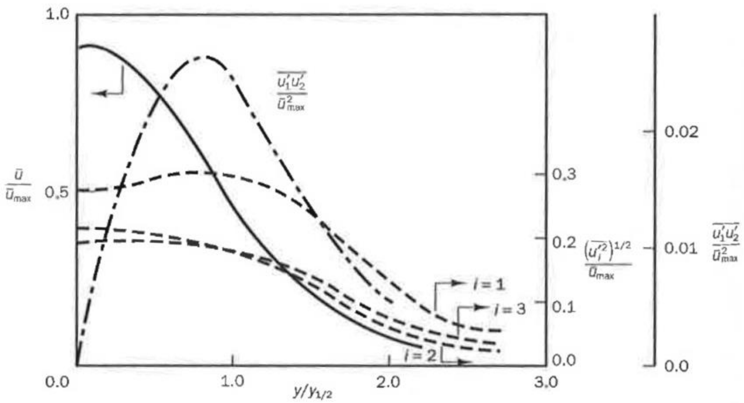
Some Turbulent Flow Examples



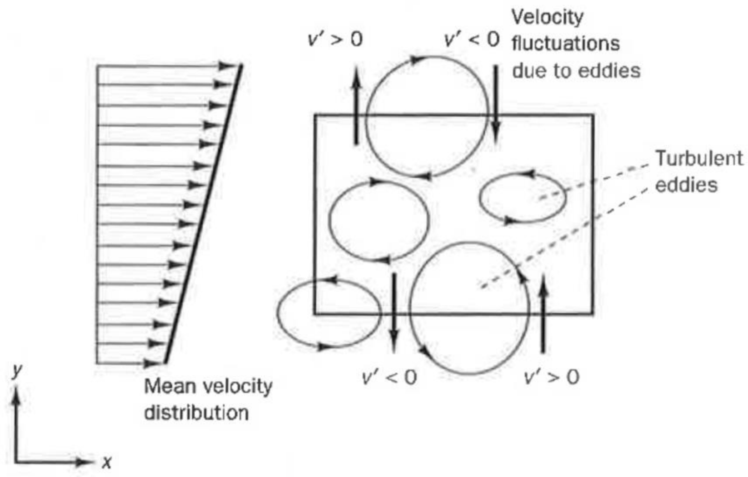
Mixing Layer







The Effects of Turbulence

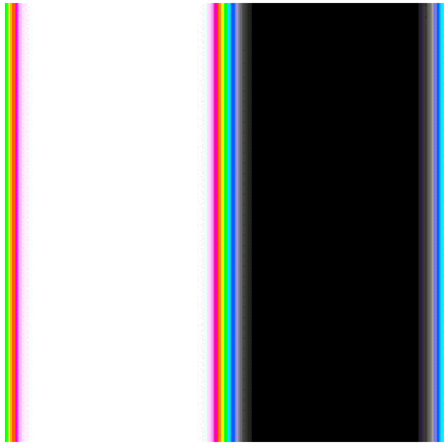


Control Volume showing 3-D turbulent Flow.

- The eddies transport fluid across the boundaries. There is no net mass transfer, but they carry energy and momentum in and out of the control volume.
- Due to mean velocity gradient "fast " (high momentum) fluid particles are transported downwards and "slow" ones are transported upwards.
- Through this process turbulence enhance diffusion (mixing) in regions with gradients.

The 3 Turbulence interaction mechanisms:

1. Momentum Exchange by Convection
2. Turbulence due to Reynolds stresses
   - Slowing and accelerating of eddies as they pass in and out of Control Volume creates extra stresses
3. For non uniform temperature or concentration cases, the eddies generate extra fluxes.



At the same time, they turn work done by viscous shear into internal energy (heat)

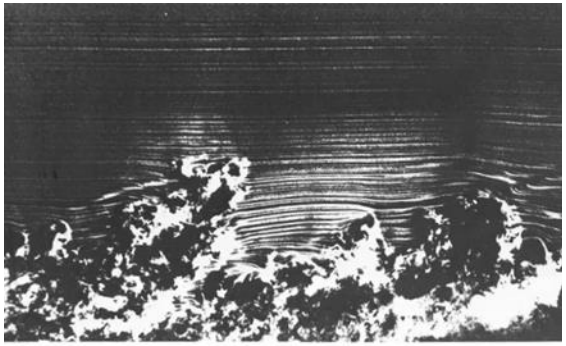
Turbulence Scales

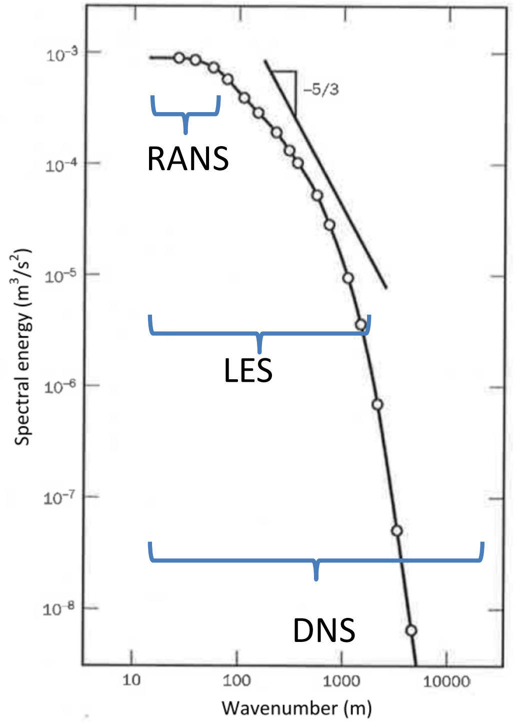
Dimensionless analysis yields the following relationships for ratios:

$$\underbrace{\frac{\eta}{l_T} \approx Re_{l_T}^{-\frac{3}{4}}}_{\text{Ratio of Length}} \; ; \quad \underbrace{\frac{t_\eta}{\frac{l_T}{v}} \approx Re_{l_T}^{-\frac{1}{2}}}_{\text{Ratio of Time}} ; \quad \underbrace{\frac{u_\eta}{v} \approx Re_{l_T}^{-\frac{1}{4}}}_{\text{Ratio of Velocity}}$$

- The difference between the largest and smallest scales increases as Reynolds number increases.
- Based these observations Kolmogorov created a hypotheses. This can be summarised as:
  - Large eddies are highly **anisotropic**. Their fluctuations are different in different directions and they are strongly affected by problem boundary conditions.
  - Structure and energy of smallest eddies only depends on rate of energy dissipation.
  - The structure of the smallest eddies is **isotropic**. Their fluctuations are the same in all directions and only depend on rate of energy flow down the energy cascaded.

→ To simulate turbulent flow, one needs to resolve (simulate) the large eddies, but can model the smallest eddies as an isentropic property that is convected.





Approaches to Simulation

A range of simulation approaches exist that can resolve different portions of the turbuelnce energy spectrum.

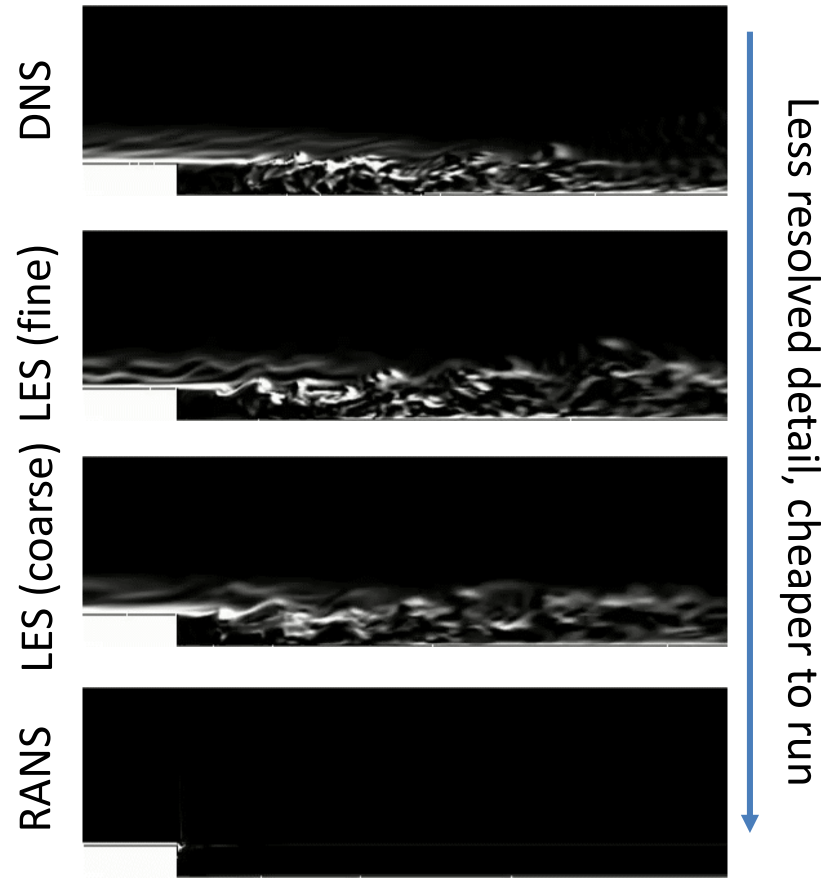
DNS -Direct Numerical Simulation: All turbulent length scales and timescales are resolved. Requires very fine meshes and time-steps. Typically limited to canonical test cases. Used for scientific research on turbulence and to develop better models

LES -Large Eddie Simulations: Only large eddies are resolved and turbulence at sub-grid scale is modelled. Sub-grid scale models, based on $-\frac{5}{3}$ relationship are employed.

RANS -Reynolds Averaged Navier Stokes Equations: Only largest and typically steady flow structures are resolved. All turbulence is modelled.

Currently RANS is the most common approach in engineering as it is a good balance between computational speed, accuracy, and required level of detail.

Video Comparison of Simulation Approaches



Different RANS Turbulence models

List of turbulence models in order of increasing complexity:

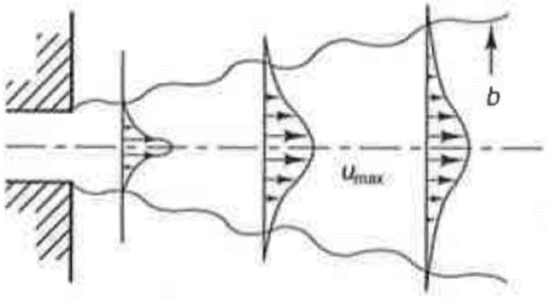
- Mixing length (zero equation)
  Only suitable for very simple flows
- Spalart-Allmaras [1] (1-equation)
  Economical computation of boundary layers in external flow
- $k - \epsilon$ [2] (2-equation)
  Good free-stream performance
- $k - oemga$ [3] (2-equation)
  Good for near wall flows
- sst models
  Blended model using $k - \epsilon$ far from wall and $k - \omega$ near wall
- LES - Large Eddy Simulation
- DNS Direct Numerical Simulation

k – turbulence energy
ε – dissipation
ω – turbulence frequency

[1] Spallart and Allmaras, (1992), *One-Equation Turbulence Model for Aerodynamic Flows*
[2] Bradshaw et al, (1981), *Engineering Calculation Methods for Turbulent Flows*
[3] Wilcox, (1988, 1993a,b, 1994), *Re-assessments of the Scale-determined Equation for Advanced Turbulence Models; Comparison of Two-Equation Turbulence Models for Boundary Layers with Pressure Gradients; Turbulence Modelling for CFD; Simulating Transition with a Two-Equation Turbulence Model*

Mixing Length Model

The Mixing length model is a "zero" equation model, meaning that no additional transport equations need to be solved.



- Based on assumption that turbulent viscosity is a function of turbulent velocity scale v and turbulent length scale l only.

$$\nu_t = C_1 \, v \, l \quad (\; C_1 \text{ dimensionless constant })$$

- Attempt to link mean flow properties to eddie velocity scale. Under assumption that only single velocity gradient $\frac{dU}{dy}$ exits

$$v = C_2 \, l \left| \frac{dU}{dy} \right| \quad (\; C_2 \text{ dimensionless constant })$$

- combinig the above equations and defining a new length scale l_m yields

$$\nu_t = l_m^2 \left| \frac{dU}{dy} \right|$$

- If $\frac{dU}{dy}$ is the only significant mean velocity gradient

$$\boxed{\tau_{t,\,xy} = \tau_{t,\,yx} = -\rho \, \overline{u'v'} = \rho \, l_m^2 \left| \frac{dU}{dy} \right| \frac{dU}{dy}}$$

Mixing Length Turbulence Model

$$\rho \frac{D\bar{u}_i}{Dt} = -\frac{d\bar{p}}{dx_i} + \underbrace{\mu \, \Delta \bar{u}_i}_{\text{viscous stress}} - \underbrace{\rho \, \rho \, l_m^2 \left| \frac{dU}{dy} \right| \frac{dU}{dy}}_{\text{Reynolds stress } = \tau_t} + \rho S_i$$

New momentum equation

Using the mixing length model

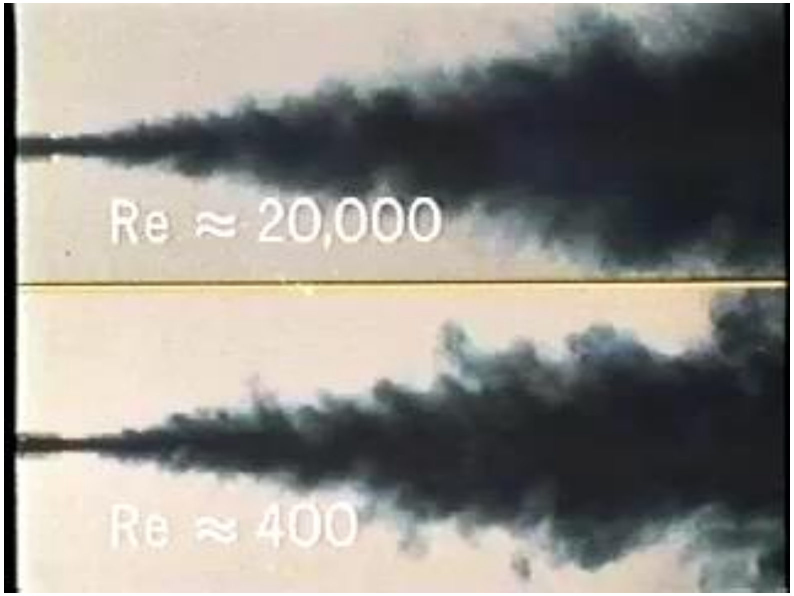
$$\tau_{t,\,xy} = \tau_{t,\,yx} = -\rho\,\overline{u'v'} = \rho\,l_m^2 \left|\frac{dU}{dy}\right| \frac{dU}{dy}$$

Can be easily implemented through addition of equation to correctly calculate turbulent shear stress.
l_m needs to be selected specific to problem (e.g. Mixing layer $l_m = 0.07\times$Layer width; Jet $l_m = 0.09\times$jet half width)
Advantages:

- Easy to implement and cheap with respect to computational resources
- Good predictions for thin shear layers, and boundary layers
- Well established

Disadvantages:

- Completely incapable of describing flows with separation and recirculation
- Only calculates mean flow properties and turbulent shear stress



Model exploits fact that major flow structure features remain constant for different Reynolds number.

Model generally only works in 2-D flows with single major velocity gradient.

l_m is a *tuning parameter* that can be adjusted until simulation and experiment match.

Spalart Allmaras Turbulence Model

The Spalart-Allmaras model is based on a single transported parameter, called the *kinematic eddy viscosity*, $\tilde{\nu}$ (or `nuTilda`). By calculating the growth, decay and transport of this parameter, regions and intensity of turbulence can be modelled. As it is a transport equation, the convection of turbulence is considered also.
Turbulent viscosity is related to $\tilde{\nu}$ by

$$\mu_t = \rho\, \tilde{\nu}\, f_{v1}$$

Here $f_{v1} = f_{v1}\left(\frac{\tilde{\nu}}{\nu}\right)$ is the wall damping function, which equals to zero near the wall and unity for high Reynolds numbers.
The Reynolds stresses are computed with

$$\tau_{ij} = -\rho\, \overline{u_i'\, u_j'} = 2\, \mu_t\, S_{ij} = \rho\, \tilde{\nu}\, f_{v1} \left(\frac{dU_i}{dx_j} + \frac{dU_j}{dx_i} \right)$$

and the transport equation for $\tilde{\nu}$ is:

$$\underbrace{\frac{d\tilde{\nu}}{dt}}_{\text{Variation with Time}} + \underbrace{u_j \frac{d\tilde{\nu}}{x_j}}_{\text{Convection}} = \underbrace{C_{b1}\left[1 - f_{t2}\right] \tilde{S}\, \tilde{\nu} + \frac{1}{\sigma} \left\{ \nabla \cdot \left[(\nu + \tilde{\nu})\, \nabla \tilde{\nu} \right] + C_{b2}\, |\nabla \nu|^2 \right\} - \left[C_{w1}\, f_w - \frac{C_{b1}}{\kappa^2} f_{t2} \right] \left(\frac{\tilde{\nu}}{d} \right)^2 + f_{t1}\, \Delta U^2}_{\text{Localised growth and decay, based on flow and geometry parameters}}$$

A number of extra equations is used to calculate local vorticity $\tilde{S}$, wall distance d, wall damping functions f_{t1}, f_{t2}, etc... And finally there are a number of experimentally tuned constants:
$\sigma = \frac{2}{3}$; $C_{b1} = 0.1355$; $C_{b2} = 0.622$; $\kappa = 0.41$; $C_{w1} = \frac{C_{b1}}{\kappa^2} + \frac{1 + C_{b2}}{\sigma}$;
$C_{w2} = 0.3$; $C_{w3} = 2$; $C_{v1} = 7.1$; $C_{t1} = 1$; $C_{t2} = 2$; $C_{t3} = 1.1$; $C_{t4} = 2$

All the Spallart Allmaras Equations

The turbulent viscosity is calculated as follows:
(`http://www.cfd-online.com/Wiki/Spalart-Allmaras_model`)

$$\nu_t = \tilde{\nu} \, f_{\nu 1}; \quad f_{\nu 1} = \frac{\chi^3}{\chi^3 + C_{\nu 1}^2}; \quad \chi = \frac{\tilde{\nu}}{\nu}$$

$$\frac{d\tilde{\nu}}{dt} + u_j \frac{d\tilde{\nu}}{x_j} = C_{b1} \left[1 - f_{t2}\right] \tilde{S} \, \tilde{\nu} + \frac{1}{\sigma} \left\{ \nabla \cdot \left[(\nu + \tilde{\nu}) \nabla \tilde{\nu} \right] + C_{b2} \left| \nabla \nu \right|^2 \right\} - \left[C_{w1} \, f_w - \frac{C_{b1}}{\kappa^2} f_{t2} \right] \left(\frac{\tilde{\nu}}{d} \right)^2 + f_{t1} \, \Delta U^2$$

$$\tilde{S} = S + \frac{\tilde{\nu}}{\kappa^2 \, d^2} f_{\nu 2}; \quad f_{\nu 2} = 1 - \frac{\chi}{1 + \chi \, f_{\nu 1}}$$

where

$$S = \sqrt{2 \, \Omega_{ij} \, \Omega_{ij}}; \quad \Omega_{ij} = \frac{1}{2} \left(\frac{du_i}{dx_j} - \frac{du_j}{dx_i} \right)$$

$$f_w = g \left(\frac{1 + C_{w3}^6}{g^6 + C_{w3}^6} \right)^{\frac{1}{6}}; \quad g = r + C_{w2} \left(r^6 - r \right); \quad r = \frac{\tilde{\nu}}{\tilde{S} \, \kappa^2 \, d^2}$$

$$f_{t1} = C_{t1} \, g_t \, \exp\left(-C_{t2} \frac{w_t^2}{\Delta U^2} \left[d^2 + g_t^2 \, d_t^2 \right] \right); \quad f_{t2} = C_{t3} \, \exp\left(-C_{t4} \, \chi^2 \right)$$

and d is the distance to the nearest surface.

The constants are:

$$\sigma = \frac{2}{3}; \quad C_{b1} = 0.1355; \quad C_{b2} = 0.622; \quad \kappa = 0.41; \quad C_{w1} = \frac{C_{b1}}{\kappa^2} + \frac{1 + C_{b2}}{\sigma};$$

$$C_{w2} = 0.3; \quad C_{w3} = 2; \quad C_{v1} = 7.1; \quad C_{t1} = 1; \quad C_{t2} = 2; \quad C_{t3} = 1.1; \quad C_{t4} = 2$$

Using the Spalart Allmaras model

Boundary conditions, both `nu` and `nuTilda` must be set at boundaries.

- Inlet: ideally $\tilde{\nu} = 0$, but some solvers have problems with this.
  In this case set $\tilde{\nu} = \frac{\nu}{10}$
- Outlet / Symmetry: convective (`zeroGradient`)
- Walls: $\tilde{\nu} = 0$ and $\nu = 0$ (or wall function)
- InternalField: Again starting problems may be encountered if field is initialised with $\tilde{\nu} = 0$.
  In this case start with $\tilde{\nu} < \frac{\nu}{5}$. This starts the model with turbulence everywhere.

Extra info `http://www.cfd-online.com/Wiki/Turbulence_free-stream_boundary_conditions`

Advantages:

- Computationally cheap. Only one equation
- Works well for external flows
- Good following in turbo machinery community
- Good performance in boundary layers with adverse pressure gradients

Disadvantages:

- Problems with free vortex flows (e.g. jet)
- Lacks sensitivity to transport processes in rapidly changing flows

RANS + turbulence models

For RANS modelling:

- Governing equations use *averaged* properties, $u(t) = U + u'(t)$
- Momentum equations get extra term for *Reynolds stresses (turbulent viscosity)*,

$$\underbrace{\rho\left(\frac{d\overline{u'_x u'_x}}{dx} + \frac{\overline{u'_x u'_y}}{dy}\right)}_{\text{Reynolds Stresses}} = \mu_t \left(\frac{d^2\bar{u_x}}{dx^2} + \frac{d^2\bar{u_x}}{dy^2}\right)$$

- Turbulent model provides a mathematical approximation to calculate turbulent viscosity, μ_t

Lets look at turbulence model implementations in OF.

1. Select suitable `RASModel` in `/case/constant/turbulenceProperties`
2. Create appropriate initial conditions and boundary conditions on `/0`

SpalartAllmaras : Requires convected property `nuTilda`, and calculated `nut`

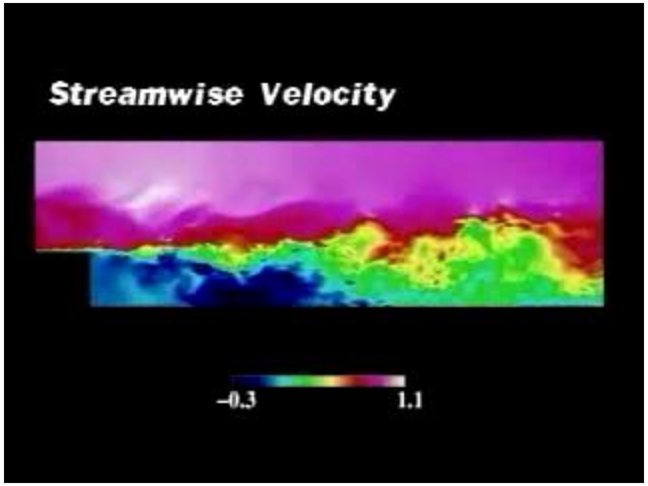
kEpsilon : Requires turbulent kinetic energy `k`, turbulent dissipation `epsilon`, and calculated `nut`

kOmega : Requires turbulent kinetic energy `k`, specific rate of dissipation `omega`, and calculated `nut`

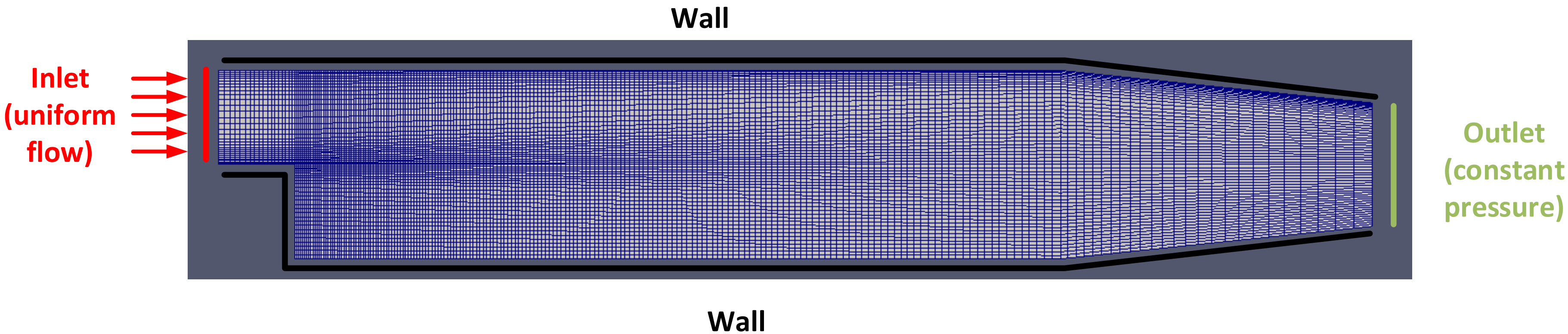
kOmegaSST : Requires turbulent kinetic energy `k`, specific rate of dissipation `omega`, and calculated `nut`

See OF example `/incompressible/simpleFoam/pitzDaily` or `/airfoil2D` for examples and appropriate set-up files

pitzDaily Backward Step

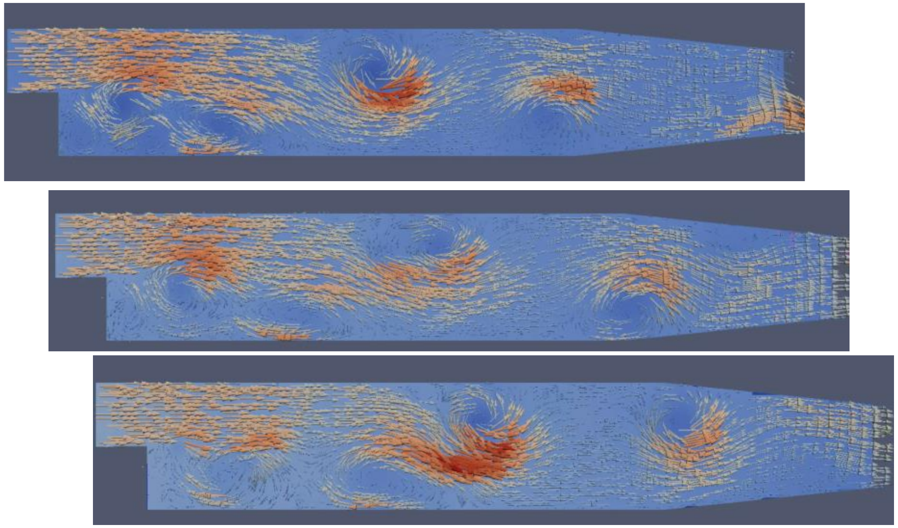


The backward facing step is a good example to test different turbulence models. Size and length of separation region is strongly dependent on turbulence modelling parameters.
The sudden expansion forces flow separation and the creation of a turbulent separation bubble.
To explore different turbulence models get a copy from the OpenFOAM tutorials and try out different settings by changing
`RASModel` in `case\constant\turbulenceProperties`
(To get case: `$ cp -r $FOAM_TUTORIALS/incompressible/simpleFoam/pitzDaily .`)



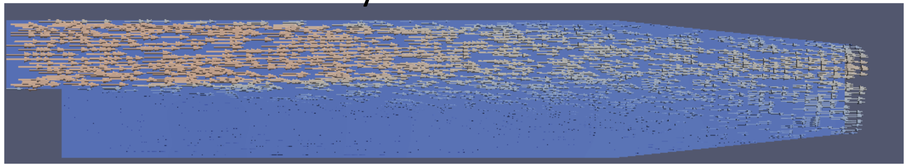
pitzDaily

The following animation shows the unsteady flow from a transient Large-Eddy-Simualtion (LES). This is illustartes what the real unsteady flow looks like. The RANS simualtions show the time-averaged equivalent.
Ideally the time average properties and effects acting on boundaries (pressure, shear, temperature, ...) are the same.

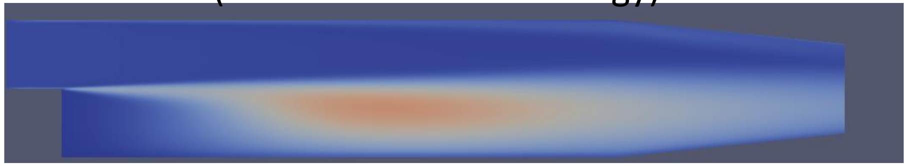


Unsteady and Transient LES

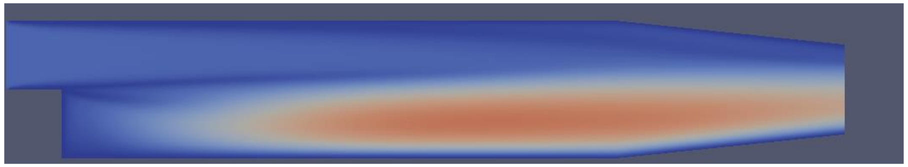
Velocity and Pressure



k (turbulence kinetic energy)



nut



Time-averaged RANS simulations.
unsteady effects are captured by *steady* distribution of `k`, which is used to calculate turbulent viscosity `nut`.

Turbulence model exercises:

Have a look at the cases in `/Desktop/Examples/Turbulence`. This will give you three folders with laminar and different RANS simualtions.
Note: All cases are solved with `simpleFoam`*, but different turbulence models are specified in* `/constant/turbulenceProperties`.

laminar : Laminar simulation case. This case results in an unsteady solution, which cannot be solved correctly using `simpleFoam`.

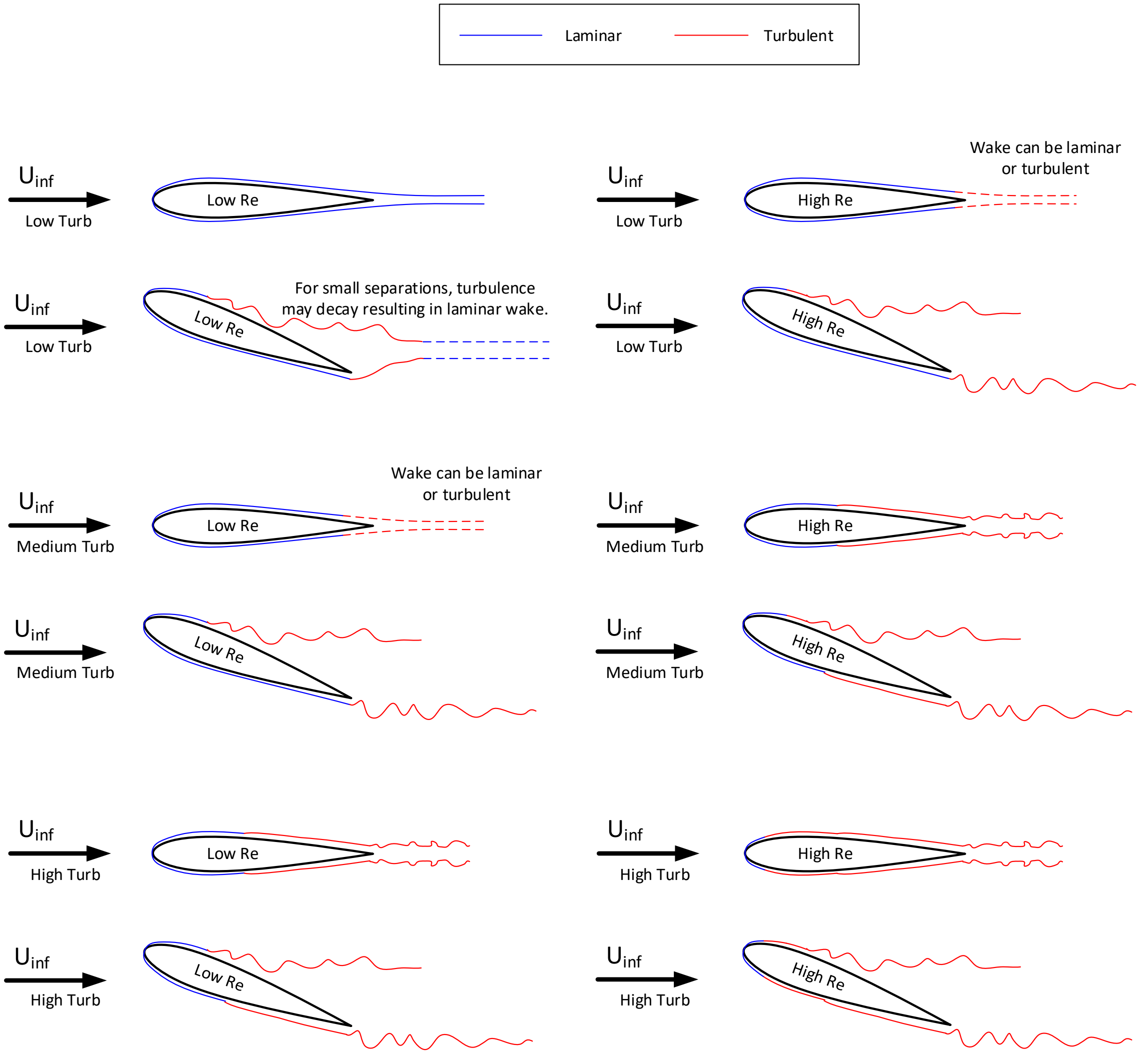
turbulent SA : RANS simualtion case using the Spalart Allmaras turbulence model.

turbulent : RANS simualtion case using energy based turbulence models.

While executing the above, also monitor the residuals to observe unsteady effects and convergence.

```
$ simpleFoam > log &
$ foamMonitor -l postProcessing/residuals/0/residuals.dat
```

When should flow be turbulent???



There is no simple answer or analysis method to determin when if flow is laminar or turbulent. The flow state depends on the flow history (where the fluid comes from) as well as the local geometry.
Reynolds number is a good indicator (i.e. high $Re \rightarrow$ turbulent) however there are many other effects that can influence transition. Some examples are adverse pressure gradients leading to separation, features (geometry) increases turbulence, previously existing free-stream turbulence, ...

Here are some example cases.

Energy Based Turbulence Models (1)

The simplest turbulence models are based on the assumption that there is a link between viscous and reynolds stresses. (E.g. turbulent stresses are an amplified version of viscous stresses)

- For newtonian fluids the stresses are proportional to fluid deformation

$$\underbrace{\tau_{ij}}_{\text{viscous}} = \mu \, s_{ij} = \mu \left(\frac{du_i}{dx_j} + \frac{du_j}{dx_i} \right) \quad \text{or } \tau_{xy} = \mu \left(\frac{du}{dy} + \frac{dv}{dx} \right) \quad \text{for 2-D}$$

- Experimental observations:
  - From energy cascade, turbulence decays, unless there is shear (feed of energy down the cascade)
  - Turbulence stresses increases as mean rate of deformation increases.

- Based on the above **Boussinesq** proposed the following relationship

$$\boxed{\underbrace{\tau_{t,\,ij}}_{\text{reynolds}} = -\rho \, \overline{u_i' u_j'} = \mu_t \left(\frac{dU_i}{dx_j} + \frac{dU_j}{dx_i} \right) - \frac{2}{3} \rho \, k \, \delta_{ij}}$$

Boussinesq Equation

where $k = \frac{1}{2} \left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2} \right)$.
The Kroenecker delta ($\delta_{ij} = 1$ if $i = j$; $\delta_{ij} = 0$ if $i \neq j$) in conjunction with second term ensures the correct values are obtained for the normal reynolds stresses (with $i = j$)

$$\tau_{t,\,xx} = -\rho \overline{u'^2}; \quad \tau_{t,\,yy} = -\rho \overline{v'^2}; \quad \tau_{t,\,zz} = -\rho \overline{w'^2}$$

Energy Based Turbulence Models (2)

- The necessity of 2nd term can be shown by considering the sum of the normal stresses (in absence of 2nd term)

$$2\mu_t S_{ii} = 2\mu_t \left(\frac{dU}{dx} + \frac{dV}{dy} + \frac{dW}{dz} \right) = 2\,\mu_t\, div\mathbf{U} = 0 \quad \text{from continuity}$$

- However in any flow the sum of the normal stresses is related to turbulent energy as follows

$$-\rho \left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2} \right) = -2\,\rho\, k$$

- Effectively to correct for this the 2nd term ($\frac{2}{3}\rho\, k\, \delta_{ij}$) equally distributes turbulent kinetic energy between the 3 normal stresses.
  - This makes an isotropic assumption

Effectively the turbulent kinetic energy is equally distributed between the x, y and z direction. This is generally not true (e.g. turbulence generated by jet is non isentropic). Nevertheless this is the best/accepted approach.

$\rightarrow$ Using the *Boussinesq* equation turbulent shear, $\tau_{x,y}$ can be related to the turbulent energy k.

k-ε Turbulence Model (1)

- Instantaneous kinetic energy is a sum of mean and turbulent energy

$$k(t) = K + k = \underbrace{\frac{1}{2}\left(U^2 + V^2 + W^2\right)}_{K} + \underbrace{\frac{1}{2}\left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}\right)}_{k}$$

- Define tensors for deformation s_{ij} and stress τ_{ij}, where

$$s_{ij} = \begin{bmatrix} s_{xx} & s_{xy} & s_{xz} \\ s_{yx} & s_{yy} & s_{yz} \\ s_{zx} & s_{zy} & s_{zz} \end{bmatrix}; \quad \text{with } s_{ij}(t) = S_{ij} + s'_{ij}$$

The idea of this turbulence model is based on modelling the turbulent energy, its transport and dissipation. Consider transport equations for K and k:

$$\frac{d(\rho K)}{dt} + div(\rho K \mathbf{U}) = div(-P\mathbf{U} + 2\mu \mathbf{U} S_{ij} - \rho \mathbf{U}\overline{u'_i u'_j}) - 2\mu S_{ij} \cdot S_{ij} + \rho \overline{u'_i u'_j} \cdot S_{ij}$$

$$\underbrace{\frac{d(\rho k)}{dt}}_{I} + \underbrace{div(\rho k \mathbf{U})}_{II} = div(-\underbrace{\overline{p'\mathbf{u}'}}_{III} + \underbrace{2\mu\overline{\mathbf{u}' s_{ij}}}_{IV} - \underbrace{\rho\frac{1}{2}\overline{u'_i \cdot u'_i u'_j}}_{V}) - \underbrace{2\mu\overline{s'_{ij} \cdot s'_{ij}}}_{VI} - \underbrace{\rho\overline{u'_i u'_j} \cdot S_{ij}}_{VII}$$

Rate of change of Φ	+	Convection of Φ	=	Transport of Φ by pressure	+	Transport of Φ by viscous stresses	+	Transport of Φ by reynolds stress	−	Rate of dissipation of Φ	+	Rate of production of Φ				

- Term VII is transfer of energy from K to k
- Term VI is the dissipation of kinetic energy, $\epsilon = 2\,\nu\,\overline{s'_{ij} \cdot s'_{ij}}$
- At high Re, term IV is always small compared to V and VI.

k-ε Turbulence Model (2)

Similar transport equations can be developed for other turbulence properties, such as rate of viscous dissipation ϵ. However these contain many unknown and unmeasurable terms.

- The standard k-ϵ model contains one equationf or k and one for ϵ.
- Define turbulent and velocity length scales: $v = k^{\frac{1}{2}}; \quad l = \frac{k^{\frac{3}{2}}}{\epsilon}$
- Eddy viscosity can then be define as $\mu_t = C\,\rho\, v\, l = \rho\, C_\mu\, \frac{k^2}{\epsilon}$ where $C_\mu = \text{constant}$
- k and ϵ are solved by two transport equations

$$\boxed{\begin{aligned} \frac{d(\rho k)}{dt} + div(\rho k \mathbf{U}) &= div\left[\frac{\mu_t}{\sigma_k} grad\, k\right] + 2\mu_t S_{ij}\cdot S_{ij} - \rho\,\epsilon \\ \frac{d(\rho\epsilon)}{dt} + div(\rho\epsilon\mathbf{U}) &= div\left[\frac{\mu_t}{\sigma_\epsilon} grad\,\epsilon\right] + C_{1\epsilon}\,\frac{\epsilon}{k}\,2\mu_t\, S_{ij}\cdot S_{ij} - C_{2\epsilon}\rho\frac{\epsilon^2}{k} \end{aligned}}$$

k-ε transport equations

- The 5 adjustable constants have been selected after extensive data fitting.

$$C_{mu} = 0.09; \quad \sigma_k = 1.00; \quad \sigma_\epsilon = 1.30; \quad C_{1\epsilon} = 1.44; \quad C_{2\epsilon} = 1.92$$

Once k is know, Reynolds stresses are calculated using Boussinesq equation:

$$-\rho\overline{u_i' u_j'} = \mu_t\left(\frac{dU_i}{dx_j} + \frac{dU_j}{dx_i}\right) - \frac{2}{3}\rho k \delta_{ij} = 2\mu_t S_{ij} - \frac{2}{3}\rho k \delta_{ij}$$

Using the k-ε Turbulence Model

Boundary Conditions:

- Inlet: Distribution of k and ϵ (`FixedValue`)
  Ideally use measured values, from literature; Otherwise try `http://www.cfd-online.com/Wiki/Turbulence_free-stream_boundary_conditions`
- Outlet / Symmetry: convective (`zeroGradient`)
- Walls:
  - $k = 0$ and $\epsilon = 0$ leads to indeterminate values as $\mu_t = \rho C_\mu \frac{k^2}{\epsilon}$
  - At high Re, k-ϵ use wall functions to avoid need to integrate equations all the way to the wall. I.e. assume log-law and calculate appropriate values for k and ϵ at wall.

    $$u^+ = \frac{U}{u_\tau} = \frac{1}{\kappa}\ln(Ey_P^+); \quad k = \frac{u_\tau^2}{\sqrt{C_\mu}}; \quad \epsilon = \frac{u_\tau^3}{\kappa y}$$

    With Karman's constant $\kappa = 0.41$ and wall roughness parameter $E = 9.8$ for smooth walls.
  - At low Reynolds numbers more elaborate corrections are required, as log-law is not accurate.

Extra info `http://www.cfd-online.com/Wiki/Turbulence_free-stream_boundary_conditions`

k-omega and k-omegaSST

The Wilcox k-omega ($k - \omega$) turbulence model is similar to k-epsilon, but omega is the specific dissipation rate.
Selected by `RASModel kEpsilon;`

The k-omega works well for boundary layer flows however the results show a dependence on the free-stream value of ω. Also for the free stream, where $\omega \rightarrow 0$ the transport equation for ω becomes indeterminate. This makes the k-omega model unsuitable for free-stream applications.

The Menter SST k-omega ($k - \omega SST$) is a combination of the k-epsilon and k-omega model.
Selected by `RASModel kOmegaSST;`

This hybrid model works by (i) trasnforming the k-epsilon model into a k-omega model close to the wall and (ii) using the k-epsilon model for the free-stream. The ϵ and ω equations are transformed by $\epsilon = k\,\omega$.

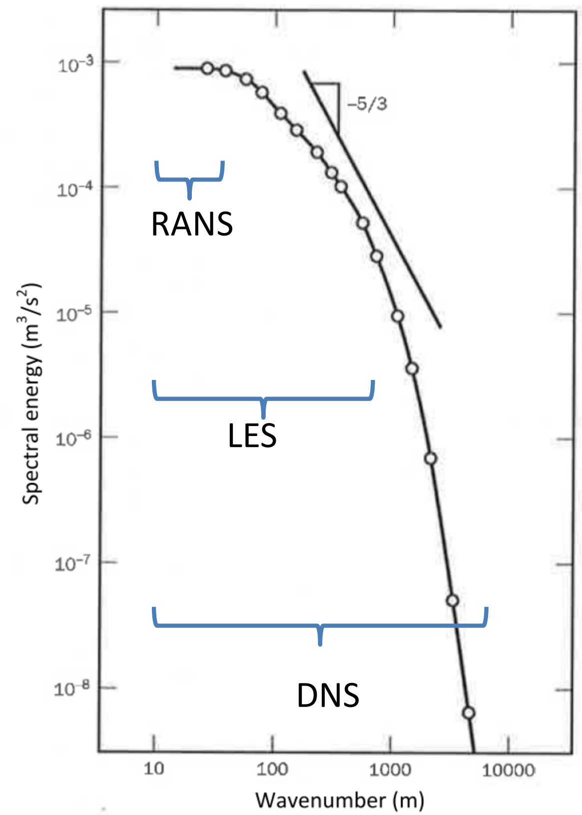
Other Turbulence Modelling Methods

Reynolds Averaged Navier Stokes Equations - RANS: Only largest eddies are resolved and solution is time time-averaged.

Unsteady RANS: Unsteady effects are simulated using the RANS framework.

Large Eddy Simulation LES: Time accurately simulate a certain fraction of turbulence length scales. Makes use of -5/3 gradient to model sub-grid-scale turbulence.

Direct Numerical Simulation DNS: Time accurately resolve all length scales and time-scales of turbulence



Required cell size is proportional to smallest resolved length scale.

Large Eddy Simulation - LES

LES simulation of fuel jet + plume inside scramjet.
Grid > 10 million cells
Source: Rolf Gehre

In LES, the governing equations are filtered, so that only spatial and temporal information above a cutoff length scale and time-scale are included.

$$\phi = \underbrace{\bar{\phi}}_{\text{filtered}} + \underbrace{\phi'}_{\text{sub-filtered}} \qquad \text{Does not fulfill Reynolds decomposition criteria!}$$

The resulting governing equations are:

$$\frac{d\rho}{dt} + \mathrm{div}(\rho\overline{\mathbf{u}}) = 0 \qquad \text{Continuity}$$

$$\underbrace{\frac{d\,(\rho\overline{u})}{dt}}_{I} + \underbrace{\mathrm{div}(\rho\overline{u}\overline{\mathbf{u}})}_{II} = -\underbrace{\frac{d\overline{p}}{dx}}_{III} + \underbrace{\mu\,\mathrm{div}(\mathrm{grad}(\overline{u}))}_{IV} - \underbrace{(\mathrm{div}(\rho\overline{u\mathbf{u}}) - \mathrm{div}(\rho\overline{u}\,\overline{\mathbf{u}}))}_{V} \qquad \text{x-momentum}$$

where I, II, III and IV for are the filtered equivalents to the RANS equation.
V is the term caused by the filtering process.

$$\mathrm{div}(\rho\overline{u\mathbf{u}}) - \mathrm{div}(\rho\overline{u}\,\overline{\mathbf{u}}) = \frac{d\tau_{ij}}{dx_j}$$

$$\tau_{ij} = \rho\overline{u\mathbf{u}} - \rho\overline{u}\overline{\mathbf{u}} = \rho\overline{u_i u_j} - \rho\overline{u}_i\overline{u}_j = \underbrace{(\rho\overline{\overline{u}_i\overline{u}_j} - \rho\overline{u}_i\overline{u}_j)}_{I} + \underbrace{\rho\overline{\overline{u}_i u'_j} + \rho\overline{u'_i\overline{u}_j}}_{II} + \underbrace{\rho\overline{u'_i u'_j}}_{III}$$

I are the *Leonards stresses*, $L_{i,j}$, caused by resolved scale.
II are the *cross-stresses*, $C_{i,j}$ caused by interaction between SGS eddies and resolved flow .
III are the *LES Reynolds stresses*, $R_{i,j}$, caused by convective momentum transfer due to interaction of SGS eddies.

- Sub-Grid-Scale, SGS (Sub-Filter-Scale) models are used to calculate the SGS stresses (just like RANS)
- Challenges arise, as SGS models must account for interactions between *all length and timescales*

As more of the eddies are resolved, results are more accurate (representative of real flow) than RANS. However finer meshes and longer running times are required to show "steady" flow solution.

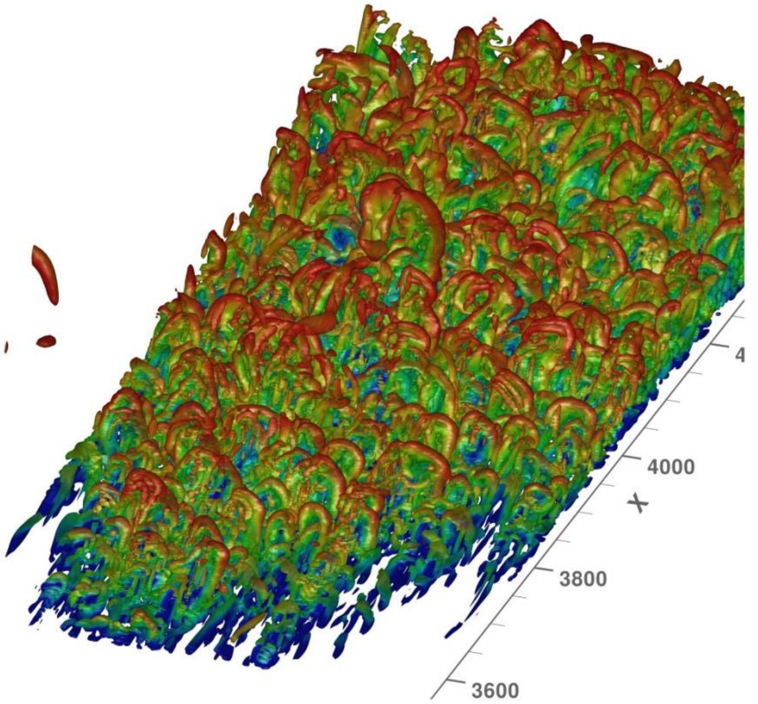
Direct Numerical Simulation - DNS

In DNS the Navier Stokes equations are solved without any turbulence model.

- All length scales need to be resolved by the mesh. $h < \eta$
  ($\eta =$ Length of smallest turbulent structure)
  - *Very* fine mesh required
- Time accurate simulation is required, even in smallest cells
  $$C = \frac{u' \, \Delta t}{h} < 1$$
  - Number of time-steps grows exponentially with Reynolds Number.

Due to high computational costs DNS is currently only viable for low Reynolds number cases.

Successfully used in fundamental research relating to turbulence. Numerical experiments using DNS provide insight/data that cannot be obtained in experiments. Such results help in the development of turbulence and Sub-Grid-Scale models.



DNS of zero-pressure-gradient flat-plate boundary layer (ZPGFPBL) by Xiaohua Wu and Parviz Moin. Image taken from DNS of the ZPGFPBL, which develops spatially from Re_theta = 80 at x=0.1 to Re_theta=1000 at x=3.5. **Grid: 209 million cells**.
http://web.stanford.edu/group/fpc/cgi-bin/fpcwiki/uploads/Main/Gallery/secondI_20to25_400_pb.jpg